[2A01] ANALYTICAL, EXPERIMENTAL AND NUMERICAL METHODS TO QUANTIFY THE PRESSURIZATION IN THE PISTON CHAMBER OF AXIAL PISTON MACHINES
*Markus Gaertner$^1$, Filipp Kratschun$^1$, Hubertus Murrenhoff$^1$ (1. Institute for Fluid Power Drives and Controls (IFAS), RWTH Aachen University)
9:00 AM - 9:16 AM

[2A02] A NEW PUMP DESIGN FOR GASOLINE DISPENSER AT THE SERVICE STATION
Yajun Liu$^1$, *Jiakun Ye$^1$, Wenhua Xie$^1$, Shuyan Zhan$^1$ (1. School of Mechanical and Automotive Engineering, South China University of Technology)
9:16 AM - 9:32 AM

[2A03] SIMULATION MODEL DEVELOPMENT TO PREDICT DYNAMIC PERFORMANCE OF VARIABLE DISPLACEMENT AXIAL PISTON TYPE PUMP
*Sung hun Kim$^1$, Sangkyu Lee$^1$, Jaechan Yoo$^1$ (1. Doosan Corporation Mottrol BG)
9:32 AM - 9:48 AM

[2A04] RESEARCH ON TRIBOLOGICAL BEHAVIOR AND LUBRICATING MECHANISM OF SLIPPER PAIR IN AXIAL PISTON PUMP UNDER THERMAL EFFECT
*tang hesheng$^1$, Ren yan, Xiang jiawei (1. wenzhou university)
9:48 AM - 10:04 AM

[2A05] THERMODYNAMIC ANALYSIS ON COMPRESSIBLE VISCOUS FLOW AND NUMERICAL MODELING STUDY ON PISTON/CYLINDER INTERFACE IN AXIAL PISTON MACHINE
*Lizhi Shang$^1$, Monika Ivantysynova$^1$ (1. Purdue University)
10:04 AM - 10:20 AM

[2A06] INTERACTION BETWEEN SWASH PLATE MOVEMENT AND COMMUTATION IN AXIAL PISTON MACHINES
*Florian Schoemacker$^1$, Hubertus Murrenhoff$^1$ (1. Institute for Fluid Power Drives and Controls (IFAS), RWTH Aachen University)
10:20 AM - 10:36 AM

[2A08] VARIABLE DISPLACEMENT ALTERNATING FLOW HYDRAULIC PUMP
*Kim Adair Stelson$^1$, Ryan Foss$^1$, Mengtang Li$^2$, Eric J. Barth$^2$, James D. Van de Ven$^1$ (1. University of Minnesota, 2. Vanderbilt University)
1:40 PM - 1:56 PM

[2A09] PERFORMANCE OF SPEED VARIABLE ASYMMETRIC PUMP CONTROLLED ASYMMETRIC HYDRAULIC CYLINDER
Long Quan$^1$, *Lei Ge$^1$, Bin Cheng Wang$^1$, Bin Li$^1$, Bin Zhao$^1$, Zhen Lu$^1$ (1. Key Lab of Advanced Transducers and Intelligent Control System of Ministry of Education, Taiyuan University of Technology)
1:56 PM - 2:12 PM

[2A10] SENSORLESS POSITION CONTROL OF DIRECT DRIVEN HYDRAULIC ACTUATORS
Tom Sourander$^1$, Matti Pietola$^1$, *Tatiana Minav$^1$, Henri Hänninen$^1$ (1. Aalto University)
2:12 PM - 2:28 PM

[2A11] HYDROSTATIC STEERING SYSTEM AND ENERGY SAVING EVALUATION IN IDLE REGIME
*Giorgio Paolo Massarotti$^1$, Pietro Marani$^1$, Massimiliano Ruggeri$^1$, Esteban Codina$^2$ (1. C.N.R. - Imamoter, 2. UPC. Universitat Politècnica de Catalunya, BarcelonaTech)
2:28 PM - 2:44 PM

[2A12] RESEARCH ON THE CONTACT PRESSURE CONTROL OF A DIE WEAR TESTER
*Chao Yang$^1$, Shigang Wang$^1$, Li Liu$^1$ (1. School of Mechanical Engineering, Shanghai Jiao Tong University)
2:44 PM - 3:00 PM

[2A13] NEW HIGH SENSITIVITY MEMS SENSOR FOR...
INDIRECT PRESSURE MEASUREMENT
*Massimiliano Ruggeri¹, Giorgio Massarotti¹, Esteban CODINA² (1. CNR-IMAMOTER, 2. Universitat Politècnica de Catalunya)
3:30 PM - 3:46 PM

[2A14] DYNAMIC CHARACTERISTICS OF THE PRESSURE-DRIVEN DEVICE BY CONSIDERING THE PRESSURE FLUCTUATIONS INDUCED BY THE PROCESS OF DROPLET FORMATION
*Wen Zeng¹, Hai Fu¹, Shuai Yuan¹, Songjing Li¹ (1. Harbin Institute of Technology)
3:46 PM - 4:02 PM

[2A15] A STUDY ON INTUITIVE CONFIGURATION OF JOYSTICK FOR OPERATOR IN FLATTENING TASK OF EXCAVATOR
*Quang Hoan Le¹, Soon Yong Yang¹ (1. University of Ulsan)
4:02 PM - 4:18 PM

[2A16] ONLINE PARAMETER ESTIMATION OF HYDRAULIC SYSTEM BASED ON UNSCENTED KALMAN FILTER
*Takashi Yamada¹, Yoshiharu Nishida¹, Akira Tsutsui¹ (1. Kobe Steel, Ltd.)
4:18 PM - 4:34 PM

[2A17] ON THE NONDIMENSIONALIZATION OF NOMINAL HYDRAULIC CYLINDER DYNAMICS
*Satoru Sakai¹ (1. Shinshu University)
4:34 PM - 4:50 PM

[2A18-22] H10 (Control & Measurements 3)
Chair: Kazuhisa Ito (Shibaura Institute of Technology), Wataru Kobayashi (Okayama University of Science)
5:00 PM - 6:20 PM Room A (ACROS Fukuoka)

[2A18] RESEARCH ON THE CHARACTERISTICS OF CONSTANT-SPEED STRETCH OF A HIGH-SPEED TENSILE MACHINE CONTROLLED BY THE ELECTRO-HYDRAULIC SERVO SYSTEM
*Enze Zhu¹, Guanglin Shi¹ (1. Shanghai Jiao Tong University)
5:00 PM - 5:16 PM

[2A19] DEVELOPMENT OF FLEXIBLE ELECTRO-HYDRAULIC CYLINDER FOR FLEXIBLE SPHERICAL ACTUATOR
*Hiroaki Tamaki¹, Shujiro Dohta¹, Tetsuya Akagi¹, Wataru Kobayashi¹, Yasuko Matsui¹ (1. Okayama University of Science)
5:16 PM - 5:32 PM

[2A20] HYDRAULIC RESONANCE CHARACTERISTICS OF THE HIGH-FREQUENCY EXCITATION SYSTEM CONTROLLED BY A 2D ROTARY VALVE
*Yan REN¹, Hesheng TANG¹, Jian RUAN² (1. Department of Mechanical and Electrical Engineering, Wenzhou University, 2. Department of Mechanical Engineering, Zhejiang University of Technology)
5:32 PM - 5:48 PM

[2A21] PERCEIVED STIMULI IN HYDRAULIC OPERATION LEVER OF CONSTRUCTION MACHINERY
*Hironao Yamada¹, Fumichika Okada², Katsutoshi Otsubo¹, Takuya Kawamura¹ (1. Dept. of Mechanical Engineering, Gifu Univ., 2. Toyota Motor Corporation)
5:48 PM - 6:04 PM

[2A22] A NOVEL INTEGRATED LOAD-SENSING ELECTRO-HYDRAULIC ACTUATOR FOR AIRCRAFT STRUCTURAL TESTS
*Yaoxing Shang¹, Xiaochao Liu¹, Zongxia Jiao¹, Jiaokang Wu¹, Liang Yan¹ (1. Beihang University)
6:04 PM - 6:20 PM
A LOW COST MOTION SERVO CONTROL SYSTEM
WITH PNEUMATIC MUSCLE ACTUATORS BASED ON PRESSURE OBSERVER AND HIGH SPEED ON/OFF VALVE
*Hao Liu¹, Xuping YAO¹, Jun TAO¹, Xinwei ZHOU¹, Pan LYU¹, Kun LIU¹ (1. State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University)

DEVELOPMENT OF PORTABLE REHABILITATION DEVICE USING FLEXIBLE EXTENSION TYPE SOFT ACTUATOR WITH BUILT-IN SMALL-SIZED QUASI-SERVO VALVE AND DISPLACEMENT SENSOR
*So Shimooka¹, Shujiro Dohta¹, Tetsuya Akagi¹, Wataru Kobayashi¹, Masataka Yoneda¹ (1. Okayama University of Science)

PERFORMANCE EVALUATION OF SUPPORTING ARM FOR REDUCING BODY LOAD USING SURFACE ELECTROMYOGRAPHY
*Tetsuro Miyazaki¹, Takuya Iijima², Yuuichi Hirahara², Kazushi Sanada² (1. Tokyo Medical and Dental University, 2. Yokohama National University)

A HUMAN-MACHINE COOPERATION CONTROL BASED ON ELECTROMYOGRAPHY FOR UPPER LIMB POWERED EXOSKELETON DRIVEN BY PNEUMATIC MUSCLE
*Jun Tao¹, Hao Liu¹ (1. State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University)

EVALUATION OF AIR COMPRESSING METHODS FOR DEVELOPMENT OF A PORTABLE PNEUMATIC POWER SOURCE
*Manabu Okui¹, Yuki Nagura², Shingo Ikikawa¹, Yasuyuki Yamada², Taro Nakamura² (1. Graduate School of Science and Engineering, Chuo University, 2. Faculty of Science and Engineering, Chuo University)

WRIST REHABILITATION SIMULATOR FOR P.T.
*Masahiro Takeiwa¹, Hiroyuki Imanaka¹ (1. Tokushima University)

DEVELOPMENT OF TENDON-DRIVEN CARE ASSISTANCE ROBOT ARM DRIVEN BY AIR PRESSURE CONTROLLING
*Daichi Kimura¹, Osamu Oyama² (1. first year master's student who belongs to Professor Oyama's laborator, 2. Meiji University)

THE CHARACTERISTIC ANALYSIS OF WATER SPRAY COOLING COMPRESSED AIR
*Guanwei Jia¹, Maolin Cai¹, Yan Shi¹, Weiqing Xu¹, Ziyue Du¹, Yuhua Li¹, Lianm Yang¹, Yaoying Shang¹, Dongkai Shen¹ (1. Beihang University, 2. Pneumatic and Thermodynamic energy storage and supply Beijing Key Laboratory)

IMPROVEMENT OF LIFTING FORCE IN VORTEX LEVITATION BY ATTACHING A CIRCULAR COLUMN
*Yuta Yamanouchi¹, Chikahisa Kawakami², Mitsuhiro Nakao¹, Minoru Fukuhara¹ (1. Kagoshima University, 2. Panasonic Co., Ltd.)

A NEW VACUUM GENERATOR BASED ON TORNADO-LIKE VORTEX FLOW
*Jyh-Chyang Renn¹, Jian-Siang Zeng¹ (1. National Yunlin University of Science and Technology)

MATHEMATICAL MODELING OF A PNEUMATIC VANE MOTOR IN MATLAB/SIMULINK
*Stephan Merkelbach¹, Joan Vidal Mas¹, Hubertus Murrenhoff¹ (1. RWTH Aachen University, Institute for Fluid Power Drives and Controls (IFAS))

NUMERICAL SIMULATION OF AIR JET
*Stephan Merkelbach¹, Joan Vidal Mas¹, Hubertus Murrenhoff¹ (1. RWTH Aachen University, Institute for Fluid Power Drives and Controls (IFAS))
IMPINGEMENT FOR ARCH BREAKING IN HOPPER
*Yige Fang1, Yajun Liu1, Cunyang Zuo1 (1. South China University of Technology, Department of Mechanical and Automotive Engineering)
6:04 PM - 6:20 PM

Room C

Oral Presentation | Oil hydraulics

[2C01-06] H6 (HST, Mobile Applications)
Chair: Xiangdong Kong (Yanshan University), Hideki Yanada (Toyohashi University of Technology)
9:00 AM - 10:36 AM Room C (ACROS Fukuoka)

[2C01] POSITION CONTROL OF VALVELESS HYDRAULIC CLUTCH ACTUATOR
*Chao Zhang1, Bingzhao Gao1, Xingjun Hu1, Yulong Lei1, Hong Chen1 (1. Jilin University, China)
9:00 AM - 9:16 AM

[2C03] DESIGN OF A POWER REGENERATIVE HYDROSTATIC WIND TURBINE TEST PLATFORM
Biswaranjan Mohanty1, Feng Wang2, *Kim A Stelson1 (1. University of Minnesota, 2. Zhejiang University)
9:32 AM - 9:48 AM

[2C04] WAVE POWER CONVERTER PENDULOR WITH HYBRID H.S.T.
*TOMII WATABE1, Prasanna GUNAWARDANE2, Hiroki MATSUMOTO3 (1. Director of T-Wave Consultant JAPAN (Inventor of Wave power converter Pendulor), 2. Senior Lecturer of Mechanical Eng. of Univ. of PERADENIYA SRLANKA (Reassembler of the Pendulor), 3. Lecturer of Mechanical Eng. of MURORAN I. T. JAPAN (Researcher on the wave propagation))
9:48 AM - 10:04 AM

[2C05] DISC BRAKE WITH HYDROMECHANICALLY CONTROLLED BRAKE TORQUE FOR RAILWAY APPLICATIONS
*Matthias Petry1, Ahmed Zak1, Hubertus Murrenhoff2 (1. Institute for Fluid Power Drives and Controls (IFAS), RWTH Aachen University)
10:04 AM - 10:20 AM

[2C06] RESEARCH ON THE EFFECTS OF DOUBLE ARC OIL GROOVE PARAMETERS ON TORQUE CHARACTERISTICS IN HYDRO-VISCOS DRIVE
YUANYUAN DENG1, Zisheng LIAN2, *Hongwei CUI2 (1. College of Mechanical Engineering, Taiyuan University of Technology, 2. Shanxi Key Laboratory of Fully Mechanized Coal Mining Equipment, Taiyuan University of Technology)
10:20 AM - 10:36 AM

Oral Presentation | Water hydraulics

Chair: Kazushi Sanada (Yokohama National University), Hideki Yanada (Toyohashi University of Technology)
1:40 PM - 3:00 PM Room C (ACROS Fukuoka)

[2C07] PERFORMANCE ANALYSIS OF LARGE FLOW SAFETY VALVE FOR POWERED SUPPORT
*YongChang Guo1,2, ZhiSheng Lian1,2, HongBing Yuan1,2, YaoYao Liao1,2 (1. College of Mechanical Engineering, Taiyuan University of Technology, 2. Shanxi Key Laboratory of Fully Mechanized Coal Mining Equipment)
1:40 PM - 1:56 PM

[2C08] EXPERIMENTAL RESULT FOR ENERGY-SAVING TECHNOLOGY IN WATER HYDRAULIC MOTOR SYSTEM
*Pha N. Pham1, Kazuhisa Ito2, Ryo Yagisawa2, Shigeru Ikeo3 (1. National Institute of Patent and Technology Exploitation, 2. Shibaura Institute of Technology, 3. Sophia University)
1:56 PM - 2:12 PM

[2C09] DESIGN AND EXPERIMENTAL RESULTS OF THE WATER HYDRAULIC DRIVE SYSTEM FOR NEUTRON BEAM SHUTTER PROTOTYPE AT CSNS
*Lixin Song1, Bing Xu1, Junhui Zhang1 (1. State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University)
2:12 PM - 2:28 PM

[2C10] STUDY ON ACTIVE CHARGE ACCUMULATOR FOR AQUA DRIVE SYSTEM (Effective Parameters on Boosting Performance)
*Satoru Takahashi1, Kazuhiisa Maeda2, Futoshi Yoshida3, Shoichiho Iio1, Ato Kitagawa4 (1. Shinshu University, 2. TOYOTA AUTO BODY, 3. KYB Corporation, 4. Tokyo Institute of Technology)
2:28 PM - 2:44 PM

[2C11] A NEW TYPE OF SPHERICAL MICRO PUMP
*Hao Pang1, Yinshui Liu1, Luyi Wang2, Zhuang Niu2 (1. Huazhong University of Science and Technology, 2. Hust-Wuxi Research Institute)
2:44 PM - 3:00 PM

The 10th JFPS International Symposium on Fluid Power 2017

©The Japan Fluid Power System Society
[2C12] ANALYSIS OF FLOW CONTROL VALVE IN HYDRAULIC SYSTEM USING PARTICLE EXCITATION
*Takahiro Ukida¹, Koichi Suzumori¹, Hiroyuki Nabae¹, Takefumi Kanda² (1. Tokyo Institute of Technology, 2. Okayama University)
3:30 PM - 3:46 PM

[2C13] COMPUTATIONAL ANALYSIS OF SOLENOID SPOOL VALVE CONSIDERED OF LEAKAGE FLOW
*Fumio Shimizu¹, Takahiro Tsukazaki¹, Takayuki Hori¹, Kazuhiro Tanaka¹, Tomohiro Yasuda², Masahito Watanabe² (1. Kyushu Institute of Technology, 2. Nidec Tosok Corporation)
3:46 PM - 4:02 PM

[2C14] A NOVEL PROPORTIONAL DIRECTIONAL VALVE WITH INDEPENDENTLY CONTROLLED PILOT STAGE
*Zhenyu Lu¹, Junhui Zhang¹, Bing Xu¹, Qi Su², Di Wang¹ (1. The State Key Lab of Fluid Power and Mechatronic Systems, Zhejiang University, 2. China Aerospace Science and Technology Corporation)
4:02 PM - 4:18 PM

[2C16] EXPERIMENT-BASED FLOW RATE INFERENTIAL MEASUREMENT METHOD OF HYDRAULIC VALVE
*Di Wang¹, Junhui Zhang¹, Bing Xu¹, Zhenyu Lu¹ (1. The State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University)
4:34 PM - 4:50 PM

[2C17] SIMULATION OF THE PRESSURE CONTROL VALVE IMPROVING RESPONSIVENESS AND STABILITY BY VARIABLE RESTRICT ORIFICE
*Seiei Masuda¹ (1. Control System Engineering Department, Aero-engine &Space operation IHI Corporation)
5:00 PM - 5:16 PM

[2C18] WORKING CHARACTERISTICS OF JET PIPE SERVO VALVE IN VIBRATION ENVIRONMENT
*yu wang¹, yao bao yin¹ (1. College of Mechanical Engineering, Tongji University)
5:16 PM - 5:32 PM

[2C19] CROSS-DOMAIN TOLERANCE DESIGN FOR DIRECTIONAL CONTROL VALVES
*Ralf TAUTENHAHN¹, Jürgen WEBER¹ (1. TU Dresden, Institute of Fluid Power)
5:32 PM - 5:48 PM

[2C20] THEORETICAL ANALYSIS ON SPOOL STUCK POSSIBILITIES OF ROTARY DIRECT DRIVE PRESSURE CONTROL SERVO VALVE
Yaobao YIN¹, Feiyi XIA¹, *Liang LU¹,², Jiayang YUAN³, Shengrong GUO³ (1. School of Mechanical Engineering, Tongji University, 2. State Key Laboratory of Fluid Power &Mechatronic Systems, 3. Aviation Key Laboratory of science and Technology on Aero Electromechanical System Integration)
5:48 PM - 6:04 PM

[2C21] VALIDATION OF AN ENHANCED MODEL OF STEADY-STATE FLOW FORCES FOR SPOOL VALVES
*Patrik Bordovsky¹, Hubertus Murrenhoff¹ (1. Institute for Fluid Power Drives and Controls (IFAS), RWTH Aachen University)
6:04 PM - 6:20 PM
Oral Presentation | Oil Hydraulics

[2A01-06] H5 (Hydraulic Pumps 2)
Chair: Randovan Peterovic (University Union Nikola Tesla of Belgrade), Toshiharu Kazama (Muroran Institute of Technology)
Thu. Oct 26, 2017 9:00 AM - 10:36 AM Room A (ACROS Fukuoka)

[2A01] ANALYTICAL, EXPERIMENTAL AND NUMERICAL METHODS TO QUANTIFY THE PRESSURIZATION IN THE PISTON CHAMBER OF AXIAL PISTON MACHINES
*Markus Gaertner¹, Filipp Kratschun¹, Hubertus Murrenhoff¹ (1. Institute for Fluid Power Drives and Controls (IFAS), RWTH Aachen University)
9:00 AM - 9:16 AM

[2A02] A NEW PUMP DESIGN FOR GASOLINE DISPENSER AT THE SERVICE STATION
Yajun Liu¹, *Jiakun Ye¹, Wenhua Xie¹, Shuyan Zhan¹ (1. School of Mechanical and Automotive Engineering, South China University of Technology)
9:16 AM - 9:32 AM

[2A03] SIMULATION MODEL DEVELOPMENT TO PREDICT DYNAMIC PERFORMANCE OF VARIABLE DISPLACEMENT AXIAL PISTON TYPE PUMP
*Sunghun Kim¹, Sangkyu Lee¹, Jaechan Yoo¹ (1. Doosan Corporation Mottrol BG)
9:32 AM - 9:48 AM

[2A04] RESEARCH ON TRIBOLOGICAL BEHAVIOR AND LUBRICATING MECHANISM OF SLIPPER PAIR IN AXIAL PISTON PUMP UNDER THERMAL EFFECT
*tang hesheng¹, Ren yan, Xiang jiawei (1. wenzhou university)
9:48 AM - 10:04 AM

[2A05] THERMODYNAMIC ANALYSIS ON COMPRESSIBLE VISCOUS FLOW AND NUMERICAL MODELING STUDY ON PISTON/CYLINDER INTERFACE IN AXIAL PISTON MACHINE
*Lizhi Shang¹, Monika Ivantysynova¹ (1. Purdue University)
10:04 AM - 10:20 AM

[2A06] INTERACTION BETWEEN SWASH PLATE MOVEMENT AND COMMUTATION IN AXIAL PISTON MACHINES
*Florian Schoemacker¹, Hubertus Murrenhoff¹ (1. Institute for Fluid Power Drives and Controls (IFAS), RWTH Aachen University)
10:20 AM - 10:36 AM
ANALYTICAL, EXPERIMENTAL AND NUMERICAL METHODS TO QUANTIFY THE PRESSURIZATION IN THE PISTON CHAMBER OF AXIAL PISTON MACHINES

Markus GÄRTNER*, Filipp KRATSCHUN*, Hubertus MURRENHOFF*

* Institute for Fluid Power Drives and Controls (IFAS)
RWTH Aachen University
Steinbachstraße 53, 52064 Aachen, Germany
(E-mail: markus.gaertner@ifas.rwth-aachen.de)

Abstract. In mobile and stationary applications, axial piston machines are often used as pumps or motors. The paper on hand deals with the pressurization behavior of swash plate axial piston units. The pressurization is influenced amongst others by the pressure on high and low pressure line \( p_{HP} \) and \( p_{LP} \) and the rotational speed \( \omega \), geometric properties, e.g. the dead volume \( V_{dead} \), the valve plate design and the fluid characteristics (density \( \rho \), viscosity \( \nu \), bulk modulus \( E \)). In the following a theoretical and an experimental analysis of the pressurization in the piston chamber and the resulting compression work is presented.

Keywords: axial piston machine, pressurization, bulk modulus, compression work

INTRODUCTION

In the past many researchers focused their work on the dynamic behavior, flow respectively pressure ripples and the acoustic emission of axial piston machines. Fiebig [1] made a holistic analysis of noise generation of hydraulic aggregates. Pumps are indicated as main noise source which is influenced by the commutation between high and low pressure and pressure pulsation. Schleihls et. al. [2] coupled pressure measurements in a cylinder block of a swash plate machine and a CFD analysis using the so-called Equilibrium Dissolve Gas Model to localize cavitation critical areas and to suggest a design optimization.

This paper deals with a detailed view on the effects that influence the pressurization in the piston chamber respecting the losses in the system consisting of piston chamber, valve and the connected piping. As a consequence of the complex resistance of the liquid column and the dissipated compression work a pressure loss and vibrations occur.

The presented work is divided into three parts. In the first part a single piston test rig from the IFAS laboratory is described. This test rig was built up for experimental research of the tribological behavior of the piston-bushing contact in swash plate machines [3]. Amongst other measured variables the dynamic pressure in the piston chamber was recorded. With a view to a more precise understanding and an optimization of the commutation, a 1D-simulation model of the single piston test rig was built up. Based on the test results the simulation is validated.

The third part deals with a theoretical study concerning the fluid characteristics and its effect on the compression work. The compression work is calculated respecting the relative amount of entrained air. For the theoretical study the pump and suction processes are compared to an ideal process cycle.

EXPERIMENTAL INVESTIGATION

At the Institute for Fluid Power Drives and Controls (IFAS), RWTH Aachen University, the tribological behavior of the piston-bushing contact of axial piston units in swash plate design is investigated mainly with experimental work, secondarily with simulation work, in the recent past [4, 5]. While analyzing the test results like friction work, piston rotation, etc. the pressurization in the piston chamber must be kept in mind.

Single Piston Test Rig

The measurement presented here is executed on a single piston test rig in the IFAS laboratory. This test rig was built up to analyze the radial and tangential friction work, leakage and temperature distribution in the piston-bushing contact. An overview of the test rig is given in figure 1.
FIGURE 1. Overview of the single piston test rig

This test rig consists of a central structure, located in the housing (see figure 2), which guides the piston via a bushing mounted in a force measurement plate. The force measurement plate is fixed to the housing with four piezo load cells. Via the compensation piston, the piston chamber is connected to the rotary valve outlet without causing a force shunt.

A wobble plate, driven by an asynchronous motor, generates the piston stroke. Due to the hold down device a piston slipper lift off is prevented.

FIGURE 2. Sectional view of the guiding unit

The piston chamber pressure, called $p_{\text{dyn}}$ in the following, is captured with a piezo-electric pressure transducer situated in the swivel plate. Figure 3 depicts the rotary valve that connects the piston chamber with the high or low pressure level ($p_{\text{HP}}, p_{\text{LP}}$), which is controlled with pressure control valves at the valve block. The rotary valve is synchronized with the wobble plate via an auxiliary shaft.

FIGURE 3. Sectional view of the rotary valve

A linking channel is machined into the sleeve, which is located on the fixed control journal and driven by a gear wheel. With the rotating linking channel in the sleeve, the valve outlet is connected with the high or low pressure line ($p_{\text{HP}}, p_{\text{LP}}$).
Regarding figure 1 and 2, the dead volume $V_{\text{dead}}$ of the entire pressurized chamber consists of the piston chamber and the valve outlet and is therefore much larger compared to commercial swash plate pumps on market. This leads to a delayed pressurization and a higher amount of compression work. Therefore, the pressurization can be studied easily.

## Test Results

The measurements are carried out at a speed of $n = 500 \ldots 2750$ rpm and a pressure $p_{\text{HP}}$ between 10 and 35 MPa. In figure 4 the measured chamber pressure $p_{\text{dyn}}$ is plotted across a full piston stroke at different rotational speeds. The high pressure level is held at 30 MPa and the low pressure level is adjusted to 5 MPa to lower the risk of cavitation.

### FIGURE 4. Measured pressurization

The measured pressure levels during suction and pump stroke differ from the set pressures $p_{\text{LP}}$ and $p_{\text{HP}}$. The commutation from low to high pressure is faster, caused by an early valve opening relative to the large capacity between piston and valve outlet. When the piston has its maximum speed at a stroke between 10 and 15 mm, a pressure increase during the pump stroke is visible. The same behavior can be seen vice versa during the suction stroke. This effect becomes more noticeable with increasing rotational speed $n$. It is caused by the total hydraulic resistance, the inductivity and the capacity of the liquid column in the piping between piston chamber and valve block outlet. The total hydraulic resistance is the sum of all hydraulic resistors such as pipes, elbows and control edges in the valve.

Two effects can be seen while commutating from high to low pressure. The pressure in the suction stroke tends to drop to zero as another consequence of the hydraulic resistance. Hence, the need to pressurize the low pressure line up to 5 MPa becomes clear. Furthermore, the pressure decrease is slower than it was expected using analytical calculations (compare fig. 8). This behavior is affected by leakage in the rotary valve that flows from the high pressure kidney to the low pressure kidney. When the linking channel is connected to the low pressure kidney, this volume flow induces such an altered pressure decrease.

## SIMULATION

To get a more precise understanding of the pressurization in the piston chamber, a 1D-simulation model was built up. This model considers the frequency dependent friction according to Theissen [6] and does not consider heat transfer. The operating parameters of the shown experiment (figure 4) and the simulation are the same. Figure 5 depicts the simulation results.

### FIGURE 5. Simulated pressurization
The model was parametrized appropriate to the geometric dependencies of the single piston test rig described above. From an abstract point of view, this model is a network of elements such as hydraulic resistors like pipes, elbows, gaps, cross section changes etc. In pipes and gaps the flow regime is supposed to be laminar. The sharp-edged resistors are assumed to behave like orifices (turbulent flow assumed). A deformation of valve parts caused by pressure fields and external forces is considered by a pressure dependent gap flow. Thermal effects like thermal strain or a temperature distribution in the valve are not implemented just as complex flow conditions e.g. turbulence or cavitation effects.

Simulation Results

The difference between experimental and simulation results are small. Hence the simulation model is validated by measured data. The simulation model can be used for several tests that would not be possible with experiments or would take much more time.

The valve timing was varied using the simulation model. So the already used adjustment of the rotary valve at the test rig is confirmed. Furthermore, an optimization of design features is possible.

THEORETICAL STUDY

The following theoretical study is aimed towards finding an analytical solution that quantifies the pressurization in the piston chamber and the compression losses of axial piston machines. In particular, the influence of the fluid’s compressibility and the valve timing on the pressurization within the piston chamber are regarded and compared to an ideal reference cycle.

Reference Cycle

The ideal reference cycle of a hydraulic swash plate machine is shown in figure 6 below. The valves’ positions represent the relative position of the valve plate to the piston chamber. At the beginning of the cycle (1) the cylinder is completely filled with the working fluid and the piston chamber is about to be disconnected from the low pressure port. Between (1) and (2) the pressurization takes place which means that the piston chamber is disconnected from both ports and the piston pressurizes the enclosed fluid. Having reached point 2 the high pressure port is connected to the piston chamber and the piston pushes the fluid out of the cylinder until it reaches the inner dead center (3). This represents the pump stroke wherein the effective work $W_{eff}$ is being released. Then the cylinder is being disconnected from the high pressure port and is being connected to the low pressure port (4) which induces a pressure drop within the piston chamber. Once connected with the low pressure port, the piston chamber is being filled with the working fluid until it is completely full and the cycle restarts from (1) again.

FIGURE 6. Ideal circular process

In a real machine, there are three main deviations from the ideal cycle. Firstly, the fluid is elastic, which results in additional compression work which has to be applied to pressurize the fluid. Secondly, the valve opening and closing process is of finite velocity. Thirdly there are friction and flow losses within the cylinder, which were examined in the previous chapter. The following chapter deals with losses due to the compressibility of the fluid.
Compression Losses

As mentioned before the fluid is rather elastic. Entrained air in the shape of micro bubbles reduces the stiffness (bulk modulus) of the fluid significantly, which is shown in this chapter. In literature, there are several models known for the calculation of the effective bulk modulus of hydraulic fluid with entrained air. Gholizadeh et. al. [7] summarized selected fluid models concerning the fluid bulk modulus disregarding the effect of entrained air, which is feasible at a high pressure. In [8], the effect of air in hydraulic cylinders on the eigenfrequency and the stiffness of the hydraulic system at pressures under 10 MPa are studied by analytic and experimental work. Kajaste [9], Kim [10] and Ruan [11] analyzed several models and examined experimental validations under different temperatures and different amounts of entrained air. According to Findeisen [12] dissolved air has no effect on the physical characteristics of hydraulic oil. In the given paper, the IFAS [13] approach is taken to calculate compression losses. Herein the fluid volume is regarded as a perfect composition of air volume and oil volume, as shown in (1).

\[ V = V_{oil} + V_{air} \]  

(1)

The bulk modulus for pure hydraulic oil without entrained air depends on the pressure \( p \) as follows in (2) [13]. Herein \( E_0 \) is the bulk modulus of pure oil at atmospheric pressure and \( m \) is a fluid specific material constant that needs to be identified by measurements. In this paper, these values are set to \( E_0 = 1500 \) MPa and \( m = 10 \) (see gray continuous line in figure 7).

\[ E_{oil} = E_0 + m \cdot p \]  

(2)

The bulk modulus of the entrained air is derived from the polytropic approach (9). Furthermore, the relative amount of entrained air \( \alpha \) is introduced and is defined by (3).

\[ \alpha = \frac{V_{air}}{V} \]  

(3)

The composed bulk modulus (IFAS model) can be derived using the equations mentioned above and is given by (4), considering the linear pressure dependency in equation 3 and the relative amount of entrained air \( \alpha \), the atmospheric pressure \( p_0 \), the current pressure \( p \) and a material constant \( m \). Kim verified this model with measurements [10].

\[
E_{oil+air} = \left(1 - \alpha\right) \left[1 + \frac{m \cdot p}{E_0}\right]^{\frac{1}{m}} + \alpha \left(\frac{p_0}{p}\right)^{\frac{1}{m}}
\]

(4)

Figure 7 shows the bulk modulus of the IFAS model compared to the models of Nykänen, Wiley and Yu, wherein the bulk modulus of the pure oil is considered to be constant [9, 11]. The influence of entrained air is obvious and cannot be neglected within the derivation of the compression work.
The elasticity of the fluid has to be implemented into the reference cycle, which can be seen in the figure below. Herein $W_C$ is the compression work needed to pressurize the fluid. $W_E$ is the expansion work, which is done by the fluid, once the low pressure port is connected to the pressure chamber. Since the stroke volume was exerted by the pump, only the dead volume ($V_3$) remains under high pressure and therefore the expansion work done by the remaining volume has to be less than the compression work, which had to be added to the pump. The difference of the compression work and the expansion work is defined as compression loss, since that amount of energy has to be added to the pump’s shaft.

**FIGURE 8.** Calculation of the compression work

The compression and expansion work can be calculated by solving equation 5, which holds true for both, oil and entrained air within the chamber volume. Therefore, a function $p(V)$ is needed for both phases.

$$W_{c1} = \frac{2}{1} \int p \, dV$$

(5)

For the oil phase equation (6), proposed by [10], is taken and inverted, which leads to equation 7. Integrating equation 7 leads to the compression work needed for the pressurization of the oil within the piston chamber.

$$V(p) = (1 - \alpha)V_1 \left(1 + \frac{mp}{E_o}\right)^{\frac{1}{\alpha}}$$

(6)

$$p(V) = \frac{E_o}{m} \left(\frac{V}{V_1}\right)^{\frac{1}{\alpha}} - 1$$

(7)

$$W_{C,\text{oil}} = \frac{V E_o}{m} \left[1 - m \left(\frac{V}{V_1}\right)^{1 - \frac{1}{\alpha}} - \frac{V}{V_1}\right]_{V_4}^{V_3}$$

(8)

For the air phase, the polytropic approach (9) is chosen, wherein $n$ is the polytropic exponent. For an isentropic process, $n$ is set to the specific heat ratio of air $\kappa = 1.4$ and for an isothermal process $n$ is set to unity. The integration of $p$ over $V$ leads to equation (10) for the isentropic case and (11) for the isothermal case.

$$p(V) = p_1 \left(\frac{V}{V_1}\right)^{n - 1}$$

(9)

$$W_{C,\text{air}} = \frac{\alpha V p_1}{1 - \mu} \left[\left(\frac{V}{V_1}\right)^{1 + \frac{1}{\alpha}} - 1\right]_{V_4}^{V_3}$$

(10)

$$W_{C,\text{air}} = \alpha V p_1 \ln \left(\frac{V}{V_2}\right)$$

(11)

Since the equations for the compression work of both phases are derived, the pressurization work of the entire fluid can be calculated. It is assumed that the compression of the entrained air takes place isentropically, but the thermal diffusion of the compressed entrained air leads to an immediate pressure loss within the bubbles. This
leads to energy dissipation of the compressed air, whose amount is the difference of the isentropic and isothermal process, which is derived by the subtraction of (10) and (11) and inserting (9) resulting in (12).

\[
W_{\text{C,at,dis}} = \alpha V_p \left( \begin{array}{c} \frac{P_2}{P_1} \end{array} \right)^{1 - \frac{\gamma}{\kappa}} - 1 - \ln \left( \frac{P_2}{P_1} \right) \tag{12}
\]

The derivation of the equations for the expansion work is done the same way it is carried out for the compression work and leads to (13) and (14).

\[
W_{\text{C,ex}} = \frac{V_p E_o}{m} \left( 1 - m \left( \frac{V}{V_i} \right) - \frac{V}{V_i} \right) \tag{13}
\]

\[
W_{\text{E,ex}} = \frac{\alpha V_p P_t (1 - m) \left( V \right)^{1 - \frac{\gamma}{\kappa}}}{1 - \frac{\gamma}{\kappa}} \tag{14}
\]

Finally the compression loss can be calculated by adding the compression and expansion work which results in (15).

\[
W_{\text{C}} = W_{\text{C,at}} + W_{\text{C,at,dis}} + W_{\text{E,ex}} + W_{\text{E,ex}} \tag{15}
\]

Figure 9 (left), depicts the normalized dissipated compression work \( W_{\text{AC,norm}} \) across the pressure for different amounts of entrained air. The center graph shows a detail of the left diagram at low pressure.

![Normalized dissipated compression work over pressure and dead volume](image)

**FIGURE 9.** Normalized dissipated compression work over pressure \( p \) and dead volume \( V_{\text{dead}} \) for different entrained air values

In figure 9 (right), the influence of the dead volume is plotted. It can clearly be seen that dissipation due to the compression is unavoidable by reducing the dead volume.

Compared to the losses caused by friction in the piston bushing contact, the dissipated compression work has the same order of magnitude in a wide operation range. The friction work in this contact has a very complex behavior and depends amongst others on the rotational speed of the hydraulic unit. Because of its complexity a more detailed analysis relating to the friction work is not part of this paper.

**Ideal Valve Timing**

The operating point that is characterized by the pressure level at the high and low pressure port, the rotational speed, the swash plate angle, the fluid temperature, the amount of entrained air and also the design of the pump (e.g. dead volume) influence the pressurization in the piston chamber. To achieve an efficient and low noise operating of the axial piston unit the valve timing needs to be adapted. For an ideal valve timing \( \phi_{\text{opt}} \) the valve opens when the piston chamber pressure has reached the set pressure level \( p_{\text{HP}} \) or \( p_{\text{LP}} \) (point 2 or 4 in figure 8). Most commutation devices in swash plate pumps are designed for a certain operational area. These non-adaptive devices show following effects when the valve opens too late (fig. 10 a) or too soon (fig. 10 c).
These effects are explained by means of the compression process. The expansion can be regarded analogously. A too late valve opening induces a pressure peak in the piston chamber followed by a short equalizing volume out-flow of the piston chamber. This not only leads to pressure pulsation and noise emission but also to a drive torque peak. When the valve opens too early, the pressure in the piston chamber needs to be adapted by a volume flow into the chamber, that is, contrary to the usual flow direction. As a consequence the effective volume flow of the pump is reduced.

CONCLUSION

Via an experimental investigation on a single piston test rig, a 1D-Simulation and an analytic study the pressurization of the piston chamber is examined. First measurements of the piston chamber pressure are presented and analyzed. Afterwards a 1D-simulation model is built up according to the dimensional properties of the single test rig. This simulation is validated with the measured data. Both analyses show similar results. Finally the compression losses are calculated and related to the friction losses. The calculations revealed that the compression losses lie within the same range as the friction losses and therefore cannot be neglected, which contradicts the common theory. Afterwards the effect of unfitted valve timing on the operating behavior of axial piston units is discussed. By means of experimental data and a 1D-simulation model the main effects on the pressurization in the piston chamber are analyzed. Mostly the complex resistance of the valve and the effective bulk modulus have an effect on the operating behavior regarding the efficiency and noise emission. A determination of the ideal valve timing can only be done respecting these characteristics.

REFERENCES

5. GÄRTNER, M., et. al., Comparison of analytical and experimental investigation of volumetric losses in the piston-bushing contact of axial piston machines, Antriebstechnisches Kolloquium, pp. 107-115, 2017

FIGURE 10. Over- and under-compression in the piston chamber due to different valve timing (c.f. [14])

\[ \phi_{\text{init}} > \phi_{\text{opt}} \]
\[ \phi_{\text{init}} = \phi_{\text{opt}} \]
\[ \phi_{\text{init}} < \phi_{\text{opt}} \]
A New Pump Design for Gasoline Dispenser at The Service Station

Yajun Liu*, Jiakun Ye*, Wenhua Xie*, Shuyan Zhan*

*School of Mechanical and Automotive Engineering, South China University of Technology
Guangzhou, Guangdong, 510640 China
(E-mail: corresponding.authors_yajun@scut.edu.cn)

Abstract: In order to improve the efficiency of gasoline dispenser, a new vane pump design is proposed. It is a hollow vane that aims to reduce the cavitation and increase the efficiency. In the paper explanation about force is given to account for the design. Contrast experiments are carried out to demonstrate the performance of the new pump. On the basis of the experimental data, the volumetric efficiency of pump and total efficiency of system are better than the old one. What's more, a mathematical model is established to investigate the effect of clearances' thickness on the distribution of dead-zone pressure. Results show that the clearance of stator-rotor is of great importance to the dead-zone pressure.

Keywords: Gasoline, New pump, Structure, Efficiency, Pressure

INTRODUCTION

Dispenser that mainly comprises of pump, motor, flowmeter and nozzle is a refueling terminal which injects fuel into tank of automobile at the service station. Compared to diesel, gasoline is a volatile medium with much lower viscosity causing low efficiency of its' pumping system. According to the experiments, the pumping efficiency of system is usually no more than 25% [1]. Even if the efficiency is low and thousands tons of fuel is refueled into vehicles' tank each day, a little investigation is focused on the efficiency of fuel dispenser. With the variable speed technique, the efficiency of system can be improved at last 20% by optimizing systems' control strategy[2]. In fact, the pump of dispenser plays an important role in the system, but there has been little research about fuel pump, since it was invented [3]. As for the pump used at the service station, it should keep the vacuum of pump between \[-40000 \text{ to } -30000\] Pa. Because of this, cavitation that results in noise and low efficiency occurs easily.

Moreover, the mechanical contact between pump and motor leads to leakage easily. The leakage of gasoline not only pollutes environment but also causes danger to people around the station. However, as the development of magnetic coupling, this kind of leakage can be avoided completely. The magnetic coupling that is non-mechanical contact does transmit torque with magnetic force. Because of the good performance of seal, the magnetic driven devices have been used in danger industry such as chemical industry[4]. Though the technique of magnetic coupling is mature, a few dispenser works with magnetic driven pump. Due to the advantage of magnetic coupling, it should be assembled into the pump.

Hence, to improve the volumetric efficiency, total efficiency of system and avoiding leakage, a new vane pump with magnetic coupling and unique structure is proposed for gasoline.

In addition, the thickness of clearance has great effect on the efficiency of pump. For one thing, the clearance influences the volumetric efficiency and vacuum of pump. For another, it relates to the inner pressure distribution[5]. About the internal pressure distribution study of vane pump, Jang et al.[6-8] mainly established the pressure ripple, flow ripple and kinetics model as well as testing verification. In gear pump, trapping is a
deadly problem which limits its performance. To solve the problem, many of works about damping tank were done [9-10]. In regard to investigating the thickness of clearance to reduce the trapping problem, it is insufficient. Though the efficiency of pump is higher, trapping occurs.

Therefore, in this paper, the structure of the pump will be introduced firstly, and explanation will be presented. The characteristic of pressure under different thickness of clearance will be investigated by mathematical model.

**DESIGN OF NEW PUMP**

**The Key Structure of Pump and Analysis**

As is showed in fig.1, there is the major structure of new pump and the inner structure of pump is changed thoroughly. The medium will go through from inner stator to hollow vane and at last fuel will full of the chamber, when the pump is driven by the motor with magnetic coupling. It is different from traditional vane pump that sucks fuel from outer stator.

![FIGURE 1. Primary Structure of New Vane Pump](image1)

The rotor is modified as showed in fig.1 and fig.2. The material of the rotor is copper and the material of inner stator is 41Cr4(ISO) as well as the outer stators’ material. The cooperating style of them is hard-soft-hard (hardness of material) making sure that they don’t stick each other as the thickness of clearance reduced. Experiments showed that if their cooperating style is hard-hard-hard, the pump will stick after it runs several minutes. Apart from vane-slot, there are four slots at the rotor. On the one hand, it is material saving. On the other hand, it helps to decrease the overshoot of dispenser. The weight of rotor is something to do with moment of inertia that affects the overshoot.

The vane is hollow like fig.2. A metal bar is assembled in the vane to add the weight of vane. The vane will paste to the wall of outer stator due to the centrifugal force and this distinguishes the common seal such as injecting the high pressure oil into the tail of vane or installing the follower ring at the rotor.

![FIGURE 2. Structure of Rotor and Vane](image2)

![FIGURE 3. Hydraulic Pressure of Vane](image3)

In the structure, the force of vane is optimized. A simplified model is established as fig.3. Compared to force
of solid vane, $F_1$ and $F_2$ are equal as well as pressure 1 ($P_1$) and pressure 2 ($P_2$). Hence, the force and pressure can be ignored, so pressure impact on vane is eased or removed. It also makes the vane pasted wall of outer stator by centrifugal force tightly. It not only seals well, but also relieves wear between vane-tip. Force of traditional vane is $F_1$ and that of new pump is $F_n$. Their comparison is showed as follow. Obviously, $F_n$ is smaller than $F_1$ . When calculating, pressure of dispenser is more than 250000Pa normally, and the length of vane is about 0.06m. The thickness of vane is 0.004m. For new pump, the speed of motor is about 22.5r/s, and radius of vane is 0.025m approximately.

$$F_1 = PS = 250000 \cdot \frac{4}{1000} \cdot \frac{60}{1000} = 60N$$  \hspace{1cm} (1)

$$F_n = ML\omega^2 = \frac{10 \cdot 9.8}{1000} \cdot \frac{25}{1000} \cdot \left(\frac{2\pi \cdot 1350}{60}\right)^2 = 49N$$  \hspace{1cm} (2)

What’s more, the new structure is beneficial for oil flowing into the chamber, once the volume of chamber increases. It is helpful to reduce cavitation, save energy and increase volumetric efficiency. As a positive displacement pump, the pressure of chamber will be negative at low pressure area, so the oil will be sucked into chamber. In the process, there are centrifugal force and negative pressure working on the oil. In the new pump, the direction of oils’ centrifugal force and force caused by negative pressure is the same, but it is opposite at the traditional vane pump. It means that more energy is consumed at traditional vane pump.

**Experiments and Results**

The optimization aims to increase the efficiency of pump. Experiments are carried out to certify the higher efficiency of new pump. The tested medium is testing fluid (D80) that the viscosity is the same as diesel. The motor integrated with magnetic driven pump is 21.2(t/s)/370W, while the motor driving the traditional vane pump is 15.5(t/s)/550W. The displacement of new pump is $4.96 \times 10^{-5}$ m$^3$/r, and that of the traditional vane pump is $1.5 \times 10^{-4}$ m$^3$/r. Both of pumps are driven directly. In the tests, input power, output flow and pressure differential are measured. Results are showed as follow.

![Graphs showing efficiency, output flow, and pressure differential results for both pumps.](image)

**FIGURE 4.** Relative Results of Tests
Curves at fig.4 (a) are the comparison of two pumps’ efficiency. The efficiency of new pump is higher than the other, caused by the increase of volumetric efficiency as curves at fig.4(b). It conforms to the investigation that as for low viscosity pumping system, the efficiency of system is depended on volumetric efficiency primarily. Though the output flow of traditional pump is larger than the new pump, the volumetric efficiency is less. Because larger flow leads higher pressure and outflow at the same pipe condition. It means more leakage. In all, no matter the total or volumetric efficiency, the performance of new pump is better than the rest.

CALCULATING THE DISTRIBUTION OF PRESSURE

Mathematical Model of Dead-Zone Pressure

To study the internal pressure distribution of dead-zone, a mathematic model including speed, working pressure and thickness of clearance has been established.

\[ l_1 = \sqrt{R^2 - e^2 \sin^2 \theta + e \cos \theta} \]  

(3)

Where, \( R \) is the radius of outer stator, \( e \) stands for eccentricity, \( \theta \) represents the initial angle of the vane.

On the basis of the geometrical relationship of \( l_1 \)and \( l_2 \), \( l_2 \) can be calculated. Therefore the changing rate of \( V_1 \) is:

\[ \frac{dV_1}{d\phi} = \frac{B}{2} (l_1^2 - l_2^2) \]  

(4)

Where, \( B \) is the length of eccentricity cavity.\( V_1 \) represents the volume of the chamber showed at fig.5(a).

There are two small triangles that included in \( V_1 \). The volume of them(\( V_2 \)) is:

\[ V_2 = \frac{B}{2} ((l_1 - r)^2 + (l_2 - r)^2) \times \cos \sigma \sin \sigma \]  

(5)

Where, \( r \) is the radius of rotor, \( \sigma \) stands for the angle between two sides.

According to fig.5 (b), the calculation of volume \( V_3 \) is Eq.(6).

\[ V_3 = B \left( \frac{1}{2} l_3^2 \varphi + r_l l_4 \sin((\tau - \varphi)/2) - \frac{1}{2} r_i^2 \tau \right) - V_v \]  

(6)

Where, \( r_i \) is the radius of inner stator, \( V_v \) is the volume of vane.

The schematic diagram is about the leakage of pump, and each of them is pressure-included leakage flow through the clearance. It is defined that leakage flow pushed into the chamber is negative. On the contrary it is
positive. Leakage flow is expressed by the following equation.

\[ \Delta q_1 = Q_2 - Q_1 = -\frac{B_i h_1^3}{12 \mu C_1} (P_0 - 2P) \]  

(7)

\[ \Delta q_2 = 2(Q_4 - Q_3) = -\frac{B_i h_1^3}{6 \mu C_2} (P_0 - 2P) \]  

(8)

\[ \Delta q_3 = Q_5 = \frac{B_i h_1^3}{12 \mu C_3} P \]  

(9)

\[ Q = \Delta q_1 + \Delta q_2 + \Delta q_3 \]  

(10)

\[ \frac{dV_L}{dt} = \omega \frac{dV_L}{d\phi} = Q \]  

(11)

Where, \( \omega \) is the rotating speed. \( \mu \) represents the dynamic viscosity of oil. \( h_i \) stands for the height of clearance, \( Bi \) is the width of clearance, \( Ci \) is on behalf of length of clearance.

According to the compressible of oil, the relationship between pressure and volume of oil is showed as follow.

\[ \frac{dp}{d\phi} = -\frac{k}{V_0} \frac{dV}{d\phi} \]  

(12)

Where, \( k \) is the bulk modulus of elasticity. \( V_0 \) is the initial volume.

Combining equations (4) to (12), there is

\[ \frac{dp}{d\phi} = -\frac{k}{V_0} \left( \frac{dV_1}{d\phi} - \frac{dV_4}{d\phi} + \frac{dV_3}{d\phi} + \frac{Q}{\omega} \right) \]  

(13)

LonggeKuta 45 is used to calculate in Matlab. Relative parameters are presented at table.1

<table>
<thead>
<tr>
<th>TABLE 1 Relative Parameter</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>( k )</td>
<td>Bulk modulus of elasticity</td>
</tr>
<tr>
<td>( \mu )</td>
<td>Dynamic viscosity</td>
</tr>
<tr>
<td>( \phi )</td>
<td>Angular velocity</td>
</tr>
<tr>
<td>( p_o )</td>
<td>Output pressure</td>
</tr>
<tr>
<td>( \varphi )</td>
<td>Dead-zone angle</td>
</tr>
<tr>
<td>( B )</td>
<td>Length of eccentricity cavity</td>
</tr>
<tr>
<td>( R )</td>
<td>Radius of outer stator</td>
</tr>
<tr>
<td>( r )</td>
<td>Radius of rotor</td>
</tr>
<tr>
<td>( r_1 )</td>
<td>Radius of inner stator</td>
</tr>
<tr>
<td>( e )</td>
<td>Eccentricity</td>
</tr>
<tr>
<td>( V_0 )</td>
<td>Initial volume</td>
</tr>
</tbody>
</table>
Results and Analysis

The clearance of stator-rotor is changed from $2 \times 10^{-5}$-4x$10^{-5}$m and others clearance included vane tip-outer stator, axial clearance of vane-rotor is changed as well. The thickness of clearance is commonly used at the design of fuel pump. The results are presented at fig.7.

![Pressure Distribution of Different Clearance](image1)

**FIGURE 7.** Pressure Distribution of Different Clearance

![Pressure Gradient at Clearance of Stator-Rotor](image2)

**FIGURE 8.** Pressure Gradient at Clearance of Stator-Rotor $2 \times 10^{-5}$ m

![Leakage Flow at Clearance of Stator-Rotor](image3)

**FIGURE 9.** The Leakage Flow at Clearance of Stator-Rotor $2 \times 10^{-5}$ m

As is showed at fig.7, no matter what clearance is in, the pressure rises up rapidly and increases steadily later. The pressure of clearance $4 \times 10^{-5}$m is larger than the other, but it is the lowest at last, when the clearance of stator-rotor is invariable. The maximum pressure decreases obviously as the clearance of stator-rotor increased. That is to say that the effect of stator-rotor’s clearance is greater than others. At each part of fig.7, curve of clearance $2 \times 10^{-5}$m will exceed the rest, but it will take more time.

On the basis of curves at fig.8, the pressure gradient of different clearance at the clearance of stator-rotor $2 \times 10^{-5}$m is quite different. Before 0.5°, the largest pressure gradient is at the clearance $4 \times 10^{-5}$m, while the smallest one is at the clearance $2 \times 10^{-5}$m. However, the situation inverts after 0.5°. The trend of the results at other stator-rotor clearance’s pressure gradient is the same as curves of fig.8.

The situation like curves at fig.7 and 8 is something to do with the leakage flow. The more leakage flow, the lower pressure it will be. At fig.9, from the start, the value of leakage flow is negative meaning that the leakage flow is pushed into the chamber. The leakage flow is pushed into the chamber easily, so the pressure changes quickly at the beginning, when clearance increasing. The absolute value of leakage flow of curve clearance $2 \times 10^{-5}$m is the smallest all the time so that the pressure is higher. The trend of leakage flow under other
stator-rotor clearance is the same as curves of fig.9.

CONCLUSIONS

This work has focused on the low viscosity pumping system. Based on the characteristic of gasoline and the gasoline pumping system, a new design pump has been put forward. Investigation is done to focus on the force and pressure. The following conclusions are drawn.

To improve the seal and reduce the noise of system, the magnetic driven coupling was used at the new design. Low viscosity and cavitation of gasoline causes the low volumetric efficiency of system. Aiming to that, the inner structure was redesign at the new pump, and explanations about the design were given based on force. Results of the comparison tests showed the good performance of the new pump. Although the tested fluid was D80, it proved that the new design worked. However, as a new pump, there was more investigation needed to be done, such as flow regime of pump at gasoline medium, the flow ripple and pressure ripple and so on. The clearance is of the great important to low viscosity pump. As the results of pressure model, the clearance of stator-rotor impacted the pressure greatly. Increasing the clearance could relieve the pressure impact effectively, but it also increased the inner leakage flow, and pressure gradient. It means that except the clearance of stator-rotor, other clearance of pump is as narrow as possible, if not, it has better to design a pressure groove to reduce the pressure impact.

REFERENCES

Simulation Model Development to Predict Dynamic Performance of Variable Displacement Axial Piston Type Pump

Sunghun Kim * and Sangkyu Lee ** and Jaechan Yoo ***

* Doosan Corporation Mottrol BG
184 Yeondeok-ro, Seongsan-gu, Changwon-si, Republic of Korea
sunghun2.kim@doosan.com
** Doosan Corporation Mottrol BG
sangkyu1.lee@doosan.com
*** Doosan Corporation Mottrol BG
jaechan.yoo@doosan.com

Abstract. Electronically controlled hydraulic components are recently required with growing the interest on energy efficiency of mobile hydraulic system. Especially, the pump for electronically controlled hydraulic system should have high dynamic performance to remove pressure peak and flow losses. Therefore in this paper, a simulation model for prediction of pump dynamic performance are introduced, and the influence factors (damping inside of cylinder, and swash moment) to be considered for an accurate simulation are discovered by the verification with tests.

Keywords: Swash moment, Pump dynamic performance, Pump simulation

INTRODUCTION

The world-wide efforts to lower emissions of carbon dioxide and energy consumption are requiring multiple actions. One of the actions is the application of an electronically controlled hydraulic system on excavator. Accordingly, many electronically controlled hydraulic system such as VBO (Virtual Bleed Off), IMV (Independent Metering Valve) and PCA (Pump Controlled Actuation), etc. are developed or being studied. For these electronic control systems, the hydraulic pump and also the main control valve (MCV) should be controllable via electric signal. Especially, the pump is required high dynamic performance to remove pressure peak and flow losses in electronically controlled hydraulic system. Therefore, our company “Doosan Mottrol” has developed a pump controlled by an electric proportional pressure reducing valve (EPPRV). In other words, the flow rate of pump is controlled by an EPPRV with mechanical feedback of swash angle as shown in Fig. 1.

But in the development of the pump, low dynamic performance in low pressure operating condition as illustrated in Fig. 2 was observed while the swash angle especially changes from maximum position to minimum position. For the analysis of this problem, a simulation model was necessary first of all and then it was developed.
So, this paper shows a simulation model to predict dynamic performance of a swash plate type variable displacement piston pump. Also, the main parameters effecting on pump dynamic performance are determined by sensitivity analysis and the influence factors to be considered for an accurate simulation are discovered by the verification with tests in this study.

![Simulation model for pump dynamics](image)

**FIGURE 2.** Response time of prototype pump depending on discharge pressure

**SIMULATION MODEL**

A simulation model to predict the dynamic performance of a pump is built using a 1D simulation program (LMS AMESim) as shown in Fig. 3. It includes a regulator controlled by EPPRV, regulating pistons, a swash plate and a simple hydraulic circuit with a pump. Each of the moving part such as spool, sleeve and swash plate has friction model calculated depending on pressure, and the hysteresis of springs is included in this simulation model. Especially, swash moment are considered through look-up table because the swash moment is a function of swash angle and discharge pressure. The swash moment is the resultant torque by pistons force toward swash plate.

By the sensitivity analysis on response time using this simulation model, it is determined that swash moment and swash plate friction are the main influence factors on the dynamic performance of pump. Especially, the swash moment is the most influence factor occurred the problem as shown Fig. 2. The swash moment effects on just one of the rising(qmin→qmax) and the falling(qmax→qmin) response time, while both of them increase with the increase of swash plate friction.

![Simulation model for pump dynamics](image)

**FIGURE 3.** Simulation model for pump dynamics

**EXPERIMENTAL VERIFICATION**

As mentioned above, to simulate the dynamic performance of an electronically controlled hydraulic pump swash moment should be simulated first of all. So, the swash moment are simulated using LMS AMESim pump model considering the inertia of pistons and the friction on tribology parts as illustrated in Fig.4. The inertia of
pistons is a major calculation factor of the swash moment if the pump has enough large displacement as shown in Fig. 5.

FIGURE 4. LMS AMESim pump model to simulate swash moment

(a) Without consideration for inertia and friction (b) With consideration for inertia and friction

FIGURE 5. Mean value of simulated swash moment depending on swash angle and discharge pressure

Fig. 6 shows the simulation results of the pump response time with the empirically assumed coulomb friction of 50 Nm at 350 bar and viscous friction of 2 Nm/rpm and the simulated swash moment. Unfortunately, the low dynamic performance in low pressure operating condition is not appeared in this simulation. There could be unexpected influence factors on the pump dynamic performance in real condition. However, they would be relevant to swash moment or swash plate friction.

FIGURE 6. Simulation results of pump dynamic performance with simulated swash moment and empirically assumed friction
One of the unexpected influence factors on swash moment could be found in a paper about piston shoe dynamics studied by Böinghoff, O., 1977. The position of resultant force on piston shoe could be eccentric by piston shoe tilting on swash plate as illustrated in Fig. 7. The eccentricity of the resultant force on piston shoe causes the change of swash moment. Deformation of rotary parts could also change the position of resultant force on piston shoe.

![Eccentricity of piston shoe rotation circle depending on piston shoe tilting](Böinghoff, O., 1977)

(a) simulation, (b) measurement
(da: 0.22m and di: 0.17m)

So, the swash moment was verified experimentally. As show in Fig. 8, the swash moment was indirectly calculated using static performance test results because of the limited space to directly install a force sensor on swash plate. The difference between discharge pressure and regulating pressure means the subtraction of coulomb friction from swash moment in case of this test pump when the swash angle increases. On the other hand, the coulomb friction is added to the swash moment if the swash plate changes from maximum angle to minimum angle. Therefore, the swash moment and the coulomb friction could be calculated by the comparison of regulating pressure when the swash angle increases and decreases as illustrated in Fig. 8.

![Measurement of swash moment and coulomb friction at discharge pressure of 200 bar](Fig. 9 shows the swash moment and the coulomb friction calculated from the static performance test results at discharge pressure of 80bar, 200bar and 300bar depending on swash angle. But the coulomb friction is just plotted depending on discharge pressure because it has nearly constant value although swash angle changes.)
The eccentricity of piston shoes rotation circle by piston shoe tilting and deformation of rotary parts is calculated by the comparison of simulation and test results as shown in Fig. 10. Fig. 10 shows that the eccentricity of piston shoes rotation circle changes depending on swash angle and discharge pressure, and the effect of swash angle is greater than discharge pressure.

Another reason for the difference of the simulation and test results on the dynamic performance of pump is the assumption of friction. The coulomb friction of swash plate is calculated from measurements with swash moment. So, the assumed viscous friction of swash plate should be finally verified. Viscous friction could be calculated by comparison of static performance test and dynamic performance test of pump as shown in Fig. 11.
The viscous friction is calculated with the assumptions as follows.
1. Viscous friction coefficient is constant whether swash angle increases or decreases.
2. Angular velocity of swash plate is constant while swash angle changes.
3. The influence of swash angle on viscous and the coulomb friction could be ignored.

But the viscous friction could not have the same value when swash angle increases and decreases because of the difference of damping inside of cylinders. The pressure inside of cylinders changes with the change of swash angle and it cause damping on swash plate. The difference of viscous friction due to damping inside of cylinders could effects on just one of rising and falling response time such as the swash moment.

Fig. 12 shows that the viscous friction in case of the decrease of swash angle is about 3 time larger than in case of the increase. The viscous friction between swash plate and supporting housing could be theoretically estimated with an assumption of gap height between them. So, it could be guessable with the difference of damping inside of cylinders and theoretically estimated viscous friction between swash plate and supporting housing that the viscous friction of swash plate in case of the decrease of swash angle is about 30% larger than in case of the increase.

FIGURE 12. Viscous friction by damping inside cylinders at discharge pressure of 50bar

The dynamic performance of pump is resimulated with the coulomb friction, the viscous friction and the swash moment calculated from static/dynamic performance test results. As shown in Fig. 13, the simulation model considered the eccentricity of piston shoe rotation circle changes and the damping inside of cylinders could predict the dynamic performance well.

FIGURE 13. Simulation results of response time with tested swash plate moment
CONCLUSION

This paper shows a simulation model for the prediction of pump dynamic performance and that swash plate moment and swash plate friction are main influence factors on the dynamic performance of pump. Especially, the flowing facts were found, based on the simulations and measurements:

1. Swash moment could change by the eccentricity of piston shoes rotation circle
   : The eccentricity of piston shoes rotation circle due to tilting of piston shoes and deformation of components should be considered in simulation of swash moment
2. Eccentricity of piston shoes rotation circle changes depending on swash angle and discharge pressure, and the effect of swash angle is greater than discharge pressure.
3. Viscous friction of swash plate could not has the same value when swash angle increases and decreases because of the difference of the damping inside of cylinders.
   : The viscous friction of swash plate in case of decrease of swash angle is about 30% larger than in case of increase for this test pump.
4. Swash moment and viscous friction causes the different dynamic performance of a pump between rising and falling of swash angle.

Several improvements to improve the dynamic performance of the prototype pump was founded using this simulation model, the low dynamic performance in low pressure operating condition could be solved with the improvements as shown in Fig. 14.

FIGURE 14. Test results of dynamic performance with improvements

REFERENCES

Research on tribological behavior and lubricating mechanism of slipper pair in axial piston pump under thermal effect

Hesheng Tang*, Yan Ren*, Jiawei Xiang*

* School of Mechanical Engineering, Wenzhou University, Wenzhou, 325035, China
(E-mail: tanghesheng321200@163.com)

Abstract. The present study is focused on accurate prediction of tribological behavior and lubricating mechanism of slipper pair in axial piston pump under thermal effects. The effects of oil film thickness and stiffness with screening material under thermal conductivity conditions were numerically investigated. In addition, a suitable material combination of the slipper pair was selected though a hydraulic test rig. Based on the theoretical and experimental studies, the combination of QT500-7 and ZY331608 under different surface modification technology was investigated. The results shows that the optimal combination of QT500-7 (nitrogen treatment) and ZY331608 (plasma-sprayed MoS$_2$) exhibits excellent self-lubricating and anti-wear properties. The lubricating transfer film, such as FeS or Cu$_3$S, shows the lowest friction coefficient and wear rate. Thus, this combination of screening material is suitable to improve remarkably wear resistance characteristic of slipper pair for axial piston pump under hydrodynamic lubrication condition in comparison with other screening materials.

Keywords: Axial-piston pump, Slipper pair, Lubricating gap, tribological behavior, thermal effect

INTRODUCTION

Axial piston pump is widely used in hydraulic systems because of its advantages, such as high power density, high limit pressure and long service life. There are several challenging issues associated with the pump, such as conflicts between lubrication and wear, and between sealing and leakage. For severe operating conditions, the temperature increase in lubricating region of friction pair may not be ignored. Particularly, slipper and swash plate form key frictional pairs, which will result in significant influences on the performance of pump.

In the earlier researches on the friction pairs of axial piston pump, some simulation models of the oil films were built with programming languages [1]. The most famous independent simulation tool is CASPAR developed by Wieczorek [2]. It was a no isothermal model and took the elastic deformation of friction pairs into consideration. Wang and Yamaguchi [3] theoretically investigated the load carrying capacity, power losses and stiffness of disk-type hydrostatic thrust bearings under eccentric load for elastic and rigid materials respectively. Koc and Hooke [4] had examined the design of hydrostatically balanced bearings used in the slippers of high pressure axial pumps, and outlined a design procedure whereby the slipper behavior, minimum film thickness could be established. In other work, Pang et al. [5] had experimentally investigated the deformation of a slipper bearing under load using photo-elastic measurement techniques. Bergada et al.[6] developed a set of useful analytic solutions for analyzing torque dynamics and barrel dynamics and measured the fluid film thickness, surface erosion and roughness to discuss the effects of oil pressure and temperature. Manring [7] analyzed the behavior of a hydrostatic thrust bearing in a stationary setting. They based their considerations on the fact that during normal operation, the bearing always deforms with either a concave or convex profile being observed. Harris [8] et al. built a dynamic model for slipper pads by bath$j^p$ simulation package that allows lift and tilt behavior to be predicted and used to examine the dynamic stability of slipper pads over the pumping cycle.

The friction and wear behavior of sliding bearings made from different material combinations under lubrication condition. Several comparison experiments with the bearings designed through the surface coating technology with different components were performed in the past decades. It was reported that solid lubricants (MoS$_2$) in the form of bonded coatings was an effective way to raise the load carrying capacity of slipper pair under complex lubrication condition. Lovell et al. [9] studied the frictional behavior of the MoS$_2$-coated ball bearings. The chemical and the physical features of coated thrust bearings are further investigated using scanning electron microscopy and chemical spectroscopy techniques. Wahl and Singer [10] studied the effect of different mass transfer flows in the performance of MoS$_2$ coatings by the sliding tests. For better understanding of the wear mechanism in slipper pair, it is necessary to observe the tribological behavior of screening material using standard pin-on-disk sliding test with tribometers.
This research will focus on tribological behavior and lubricating mechanism of slipper pair for axial piston pump. The outline of the current paper is as follows: (a) the oil lubricating mechanism of slipper pair under thermal effects will be investigated; (b) tribological performance of matching materials between the slipper and swash plate will be experimentally investigated, considering the effects of various material combinations and surface treatment method, and then the suitable material combinations of the slipper pair will be screened out.

ESTABLISHMENT OF MATHEMATICAL MODEL

2.2. Film thickness model

The external forces applied on the slipper pair is shown in Fig. 2. The inertial force of the piston bore caused by non-constant flow is ignored. The hydraulic clamping force caused by the piston structure. Axial hydraulic dynamics force in the oil film for slipper pair is mainly affected by the external force, including hydraulic force, the return spring force, the axial inertial force, the friction force caused by centrifugal force and cylinder bore friction reaction forces.

With the coordinate system of Figure 1, the force balance equation on the z-axis of slipper pair is listed.

\[ F_z \cos \beta - f (F_1 + F_2) - F_p - F_t - F_s = 0 \]  \hspace{1cm} (1)

where \( F_z \) is z-axial force, \( F_1 \) is hydraulic force, \( F_2 \) is return spring force, \( F_t \) is friction force caused by centrifugal force, \( F_s \) is axial inertial force, \( \beta \) is swash plate angle. \( F_1, F_2 \) are acting forces of piston, \( f \) is friction coefficient.

Therefore, the axial force acting on the slipper can be obtained

\[ F_z = \frac{f (F_1 + F_2) + F_p + F_t + F_s + F_x}{\cos \beta} \]  \hspace{1cm} (2)

![Figure 2. External forces applied on the slipper.](image)

In addition, the bearing force from oil film includes hydraulic force, thermal wedge bearing force and squeezing force. The thermal wedge bearing force is written as \[ F_t = \frac{\alpha_p \mu \omega^2}{Gcp \rho g \delta^4} \left[ 6R^2 \left(1 + \tan^2 \beta \cos \omega t \right) \left(R^2 - r_0^2\right) + \left(R^2 - r_0^2\right)^2 \right] \]  \hspace{1cm} (8)

where \( R \) is slipper outside radius, \( r_0 \) is slipper inside radius, \( \alpha_p \) is pressure coefficient, \( \mu \) is oil viscosity, \( G \) is equivalent power, \( \omega \) is angular velocity of slipper, \( \rho \) is oil density, \( c_p \) is fluid specific heat, \( g \) is acceleration of gravity, \( \delta \) is film thickness, \( t \) is simulation time.

The squeezing force equation at sealing land of slipper can be described as \[ F_s = \int_0^\pi 2 \pi r dr \left[ \frac{3 \pi \mu v L (R - r_0)}{2 \delta^3} \left( R^4 - r_0^4 \right) \left(R - r_0\right)^2 \ln \left( R / r_0 \right) \right] \]  \hspace{1cm} (9)

where \( F_s \) is squeezing force, \( p \) is film pressure, \( r \) is slipper radius.

The hydrodynamic bearing force is written as \[ F_h = \frac{3 \mu v L R r_0 (R - r_0)}{r_0 \delta^3 + (R - r_0)(l_i + \delta)} \]  \hspace{1cm} (10)

where \( F_h \) is hydrodynamic bearing force, \( v \) is slipper velocity, \( L \) is width of slipper sealing land, \( l_i \) is depth of groove.

Thus, submitting Eqs. (6)- (10) into Eq. (11), the slipper pocket pressure can be expressed as

\[ \text{(equation)} \]
\[ p_s = \frac{2 \ln \left( \frac{R}{r_o} \right) (F_c - F_i - F_s - F_h)}{\pi \left( R^2 - r_o^2 \right)} \] (11)

According to the principle of fluid continuity, the ratio of fluid pressure in slipper pocket can be written as

\[ \frac{p_s}{p_p} = \frac{1}{1 + \frac{64l}{3d_s^2 \ln \left( \frac{R}{r_o} \right)}} \] (12)

Substituting Eq. (12) into Eq. (11), the film thickness can be calculated by

\[ F_c - F_i - F_s - F_h = \frac{p_s}{2 \ln \left( \frac{R}{r_o} \right)} \frac{\pi \left( R^2 - r_o^2 \right)}{1 + \frac{64l}{3d_s^2 \ln \left( \frac{R}{r_o} \right)}} = 0 \] (13)

Since the thermal wedge bearing force \((F_i)\), squeezing force \((F_s)\) and hydrodynamic bearing force \((F_h)\) are obtained, the value can be submits into Eq. (13), in which the film thickness is investigated.

### 2.3 Temperature-dependent viscosity model

The control volume of fluid film within slipper pair is subjected to two major heat sources: the heat coming from the viscous dissipation associated with the fluid flow in the lubricating interfaces and in the heat generated due to the rotation of the slipper and swash plate in the oil-filled case. Thus, the overall oil film temperature on the slipper pair can be calculated by

\[ \frac{dT}{dt} = \frac{1}{c_p m} \left[ p_s \frac{dV}{dt} + \sum \dot{m}_{in} (H_{in} - H) + \sum \dot{m}_{out} (H_{out} - H) + \dot{m} \frac{dT}{dt} \right] \] (14)

In addition, viscosity is one of the important factors that affect the oil drive characteristics. Temperature has considerable influence on the oil viscosity. At present, ISO VG12 aviation hydraulic oil is usually used as the working medium for the pump, whose density is 900 kg/m³, and the dynamic viscosity is 0.018 Pa·s at 40 °C. Since the experimental data are discrete temperature points, the temperature-dependent viscosity is written as

\[ \mu = 2.961 \times 10^{-5} (T - 273)^2 - 1.827 \times 10^{-2} (T - 273)^2 + 2.835 \] (15)

Therefore, substituting Eq. (15) into Eqs. (8)-(9), the thermal wedge bearing force and thermal wedge bearing force are investigated.

### 2.4 Heat transfer model

In this section, the heat transfer process between oil film and slipper pair in axial piston pump is described. In order to select the appropriate heat transfer correlation during operation, the heat transfer regime of slipper pair is specified. When fluid flows from piston chamber into slipper central chamber, the convective heat exchange between the fluid film and slipper is written as

\[ \dot{Q}_1 = \frac{1}{h_{as} \pi r_s H_1 + k_i \pi H_1} + \frac{1}{h_{as} \pi r_s H_2 + k_i \pi H_2} \] (16)

where \(\dot{Q}_1\) is heat transfer rate of slipper, \(T_c\) is case temperature, \(H_1\) is slipper’s lug height, \(H_2\) is swash plate height, \(k_i\) is slipper thermal conductivity, \(h_{as}\) is convective heat transfer coefficient of slipper, \(h_{as}\) is convective heat transfer coefficient of swash plate.

The convective heat exchange between the fluid film and swash plate is written as

\[ \dot{Q}_2 = \frac{1}{h_{as} \pi R^2 + k_i \pi R^2} + \frac{1}{h_{as} \pi R^2} \] (17)

where \(\dot{Q}_2\) is heat transfer rate of swash plate, \(k_i\) is swash plate thermal conductivity.

The convective heat exchange between leakage fluid and case fluid is calculated by follows

\[ \dot{Q}_3 = \lambda \frac{2 \pi R \delta (T - T_c)}{h_{as} \pi R^2} \] (18)

where \(\dot{Q}_3\) is heat transfer rate of fluid, \(\lambda\) is fluid heat conduction coefficient.
Therefore, the total heat generation within control volume of fluid film through heat transfer mode can be written as

$$\dot{Q} = \dot{Q}_1 + \dot{Q}_2 + \dot{Q}_3$$

(19)

where $\dot{Q}$ is total heat transfer rate of fluid film.

Since the total heat generation within control volume of fluid film is obtained, the value can be submitted into Eq. (14), in which the oil film temperature is investigated.

2.5 Load carrying model

The support force that acts to resist the external load applied in slipper, denotes the load-carrying capacity of slipper pair. Thus, the load carrying capacity of slipper pair can be written as follows

$$J = \frac{d\left(\frac{\rho_s}{\rho_p}\right)}{d\delta} = \frac{32\pi l}{R^2 - r_0^2} \left[\ln\left(\frac{R}{r_0}\right)\right]^2 \left[1 + \frac{64l}{3\ln\left(\frac{R}{r_0}\right)d_s^3}\delta^3\right]$$

(20)

THEORETICAL RESULT

3.1 Effect of oil temperature

Fig. 10 illustrates effect of temperature on average film thickness. During the delivery stroke, the average film thickness is lower than that during the suction stroke. As the oil temperature increases from 50 °C to 90 °C, there is a prominent decrease in the average thickness of the lubricating oil film during the suction stroke. Especially when the oil temperature is 90 °C, the oscillation amplitude of average film thickness becomes smaller in the transition region, which in turn implies the breaking of the oil film. Two factors contribute to this influence, the squeezing force and thermal wedge bearing force acting on the slipper. The squeezing force increases dramatically as the oil temperature increases from 50 °C to 90 °C. At the same time, the thermal wedge bearing force becomes larger, which leads to increase viscous heat generation at the film gap between slipper and swash plate. These factors result in generating high clamping force acting on slipper that causes film thickness decrease. For high oil temperature condition, the slipper is unable to find equilibrium due to the larger thermal wedge bearing force, and the variation of oil film thickness is not compromised while the slipper is strongly unstable.

![Average film thickness vs. Shaft angle (°)](image)

Fig. 10 Effect of temperature on average film thickness

Figure 11 shows effect of temperature on load carrying capacity of slipper pair. The load carrying capacity of slipper pair periodically changes with shaft angle and increases as the film temperature increases. The physics phenomenon is related to the film stiffness that is defined as the gradient of load carrying capacity. This film stiffness changes with different film temperature. As the film thickness decreases, the load carrying capacity for high oil temperature increases faster than the low oil temperature, implying that the film stiffness increases as the temperature increases. There are two factors that affect the load carrying capacity. They are the film thickness and the oil viscosity. During delivery strokes, the load-carrying capacity increases with a high decline in the film thickness, while decreasing the fluid viscosity increases the load carrying capacity. As the film thickness drops, the temperature increases rapidly, the viscosity of the oil drops, and the overall result is a rather high increase in the load-carrying capacity.
3.2 Parameter influence

Fig. 11 Effect of temperature on load-carrying capacity of slipper pair

Fig. 16 shows the effect of slipper radius ratio on average film thickness and load carrying capacity. It can be seen that the film thickness decreases rapidly with increasing slipper radius ratio, but the load-carrying capacity becomes larger. This is because as the slipper radius ratio increases, there is less hydrodynamic bearing force on the sealing land area of slipper but larger squeezing bearing force, resulting in film thickness reduction. With increase of squeezing bearing force, the average film thickness becomes thinner and result in breaking of lubricating oil film and abrasion wear of slipper. Thus, although the load carrying capacity of slipper pair increases, there is no benefit of making slipper radius ratio larger than that corresponding to the minimum value of average film thickness. To take advantage of hydrodynamic bearing force and squeezing bearing force, the slipper radius ratio should be selected from 1.4 to 1.8.

Fig. 17 shows the effect of the orifice length diameter ratio on average film thickness and load carrying capacity. In Fig. 17, it can be found that average film thickness drops as the orifice length diameter ratio increases, but the load carrying capacity increase. Two factors contribute to this influence, the squeezing bearing force and the pressure drop along the orifice. As orifice length diameter ratio increases, the pressure drop along the orifice increase, which leads to high squeezing force generation, and result in the film pressure buildup reduction. Thus, the orifice length diameter ratio is very important in determining the load carrying capacity of slipper pair. For low orifice length diameter ratio, the average film thickness increase, but at the same time, the load carrying capacity will also decrease. For high orifice length diameter ratio, average film thickness becomes thinner, but the load carrying capacity will increase sharply. Considering all the above issues, it may be recommended that the orifice length diameter ratio should be selected from 4 to 5.

Fig. 17 Effect of orifice length diameter ratio on average film thickness and load carrying capacity
EXPERIMENT RESULT

Friction and wear tests were conducted on a MMS2A ring-on block test rig, as shown in Fig. 1. The tester connected to a computer was used to evaluate the friction coefficient and temperature of the friction pairs under oil lubrication. Fig. 2 shows the installation drawing of the friction pairs. The dimensions of upper specimen block was 6 mm (length) × 7 mm (width) × 3 mm (height). The dimensions of bottom specimen ring was Ø 16 mm (outer diameter) × Ø 40 mm (inner diameter) × 3 mm (height). The experiments were performed with rotational speed of 400 r/min, load of 400 N, and a period of 2 h. The test rig was powered in room temperature. Before the experiment, the water box was filled with enough seawater to immerse completely the upper and bottom specimens in lubricant oil. Friction coefficients and wear capacities are calculated by theoretical formulae, respectively. After the friction test, the worn morphologies of the specimens were observed under a laser scanning microscope.

4. 1 Friction Coefficient and Wear Rate

Fig. 1 shows the variation of the friction coefficient of two kinds of friction pairs with test time. At start stage (0~1200s), the surface roughness of pairing material is relatively high and the contact area is small so that the friction coefficient fluctuated around 0.125-0.135 and the amplitude of fluctuation is small. At the stable stage (1200~4800s), due to the transfer lubricating film formed on the friction surface of pairing material, the friction coefficient of A1 and A2 were fluctuated at about 0.135 and 0.125, respectively. During the stable stage, the friction coefficient is related to certain factors, such as the properties of material, load and sliding speed. At the intense stage (4800~7200s), the transfer lubricating film on the surface of pairing material ruptures, which directly contact the friction surface between upper and bottom specimens and cause oil temperature rise. In this stage, the friction coefficient of material change with the variation of oil temperature. Compared with the friction coefficient of A1 and A2, the friction coefficient of A1 was much higher than that of A2. The friction coefficient of A2 is 0.12~0.13, and the variation of friction coefficient is small. The sulfur atom in the MoS2 coating has a higher activity on copper element and iron element so that the surface of the dual material forms transfer lubricating film under the action of friction and heat stress. The transfer lubricating film is composed of Cu2S and FeS, which improves wear resistance of pairing materials and reduces friction coefficient.

Fig. 1 Wear measurement apparatus of type MMS2A and specimen

4. 1 Friction Coefficient and Wear Rate

Fig. 4 shows wear rate of pairing material surface under different surface treatments. Under the same load, the wear rate of the dual material in the group of A2 is the smallest. In the group of A2, the wear rate of QTS500-7 and ZY331608 were stable at about 2.17×10⁻⁶ mm²/N·m and 7.63×10⁻⁶ mm²/N·m, respectively. This result can be explained that the heat was accumulated on the friction surface of material due to sliding against each other, which lead to increase the contact surface temperature of friction pair. The shear strength of material decreases with the increasing surface temperature. In this condition, the micro-convexes of pairing pair surface is easier to wear, and increase the wear rate of pairing materials. In friction process, the surface of ZY331608 after plasma spraying MoS2 treatment is easy to form transfer lubricating film with Cu2S, which improve the hardness.
and shear strength of materials. The heat dissipation condition between QT500-7 and ZY331608 is improved. In addition, MoS$_2$ coating has strong adsorption capacity, and it has high bonding strength with metal material. Therefore, the ZY331608 with MoS$_2$ coating has better antifriction and anti-wear properties which is useful to decrease wear rate of material.

4.2 Worn Surface of Specimens

Fig. 6 shows the worn morphologies of QT500-7 (nitriding treatment) and ZY331608 before and after the tests. As shown in Fig. 6a, the white part of the surface morphology of QT500-7 after nitriding treatment is bright white layer of nitrided surface, which is composed of particle distribution. The hardness of nitrided layer is the biggest, and the wear resistance is the strongest. The black part of nitrided surface is the diffusion layer, which is relatively loose. The hardness of the material decreases with the increase of wear depth. When the wear depth of the material reaches the penetration layer and matrix of QT500-7, the wear resistance of the material decreases sharply. In Fig. 6b, a large number of spalling pits are formed on the friction surface of QT500-7. This result can be explained that the white layer with high surface hardness and the diffusion layer in the middle part of QT500-7 due to large shear stress are separated quickly in the friction process. Compared with Fig. 6c and Fig. 6d, the contact surface of ZY331608 has worn obvious furrow, and the direction of furrow parallels to the sliding direction. There are some debris in the furrow around. This phenomenon can be explained that the soft substrate of material in the local extrusion will form to small hard particles. In addition, the external force on the specimen surface can be divided into normal force and tangential force. On the one hand, the hard particles are pressed into the surface of the specimen under the normal force; On the other hand, the sample surface was cut in to furrow shape of wear trace by hard particles due to action of tangential force.

Fig. 6 Electron microscopy observations of QT500-7 (Nitriding treatment) and ZY331608

Fig. 7 shows the worn morphologies of QT500-7 (nitriding treatment) and ZY331608 (plasma spraying MoS$_2$ treatment) before and after the tests. Compared with Fig. 7a and Fig. 7b, a large spalling pit are formed along the frictional sliding direction on the friction surface of QT500-7. In the Fig. 7b, the number of spalling pits is decreased, and the depth of pit is shallow. Compared with 7C and 7d, it can be seen that the worn surface of ZY331608 is smooth. There are no obvious wear trace after the test, but the convex crystal particle are worn. The shedding phenomenon of worn face is not obvious. The adhesion wear and abrasive wear of materials are significantly reduced. The result can be explained that the MoS$_2$ coating can obviously improve the friction and wear resistance of the multi-element complex brass, which can be explained by the crystal structure particularity of MoS$_2$. The MoS$_2$ has the characteristics of six layered lattice with compact arrangement, which the deformation resistance of material is so small that slips easily along the surface. The copper element in the contact surface react with sulfur atoms to form the transfer lubricating film composed of Cu$_2$S, which plays a role in reducing friction and lubrication on the friction materials.
CONCLUSIONS

(1) As oil temperature increases, the film thickness decreases, but the load-carrying capacity will increase dramatically. The squeezing force and thermal wedge bearing force are main factors that affect film thickness and load carrying capacity. At high oil temperature, there is high viscous dispassion at the film gap, which leads to increase thermal wedge bearing force. Because the combination action of squeezing force and the thermal wedge bearing force become larger, the film thickness decreases with increasing clamping force, but load-carrying capacity of slipper pair will increase.

(2) Slipper radius ratio and orifice length diameter ratio have significant influence on the film thickness and load carrying capacity behaviors. If the slipper radius ratio increases, the film thickness decreases, but the load carrying capacity will increase sharply. As the slipper radius ratio increases, there is larger squeezing bearing force on the sealing land area of slipper, resulting in thinner film thickness.

(3) In friction process, the surface of ZY331608 after plasma spraying MoS$_2$ treatment is easy to form transfer lubricating film with Cu$_2$S, which improve the hardness and shear strength of materials.

ACKNOWLEDGMENTS

This paper is supported by the Notional Natural Science Foundation of China (No.51505338) and the Youths Science Foundation of Zhejiang (No. LQ16E050004 and No. LQ17E050003).

REFERENCES


Fig.7 Worn morphologies of QT500-7 (Nitriding treatment) and ZY331608(Plasma-spraying MoS$_2$)
THERMODYNAMIC ANALYSIS ON COMPRESSIBLE VISCOUS FLOW AND NUMERICAL MODELING STUDY ON PISTON/CYLINDER INTERFACE IN AXIAL PISTON MACHINE

Lizhi SHANG* and Monika IVANTYSYNOVA**

* Department of Agricultural and Biological Engineering, Graduate Research Assistant, Purdue University
1500 Kepner Dr., Lafayette, IN, 47905 U.S.A
(E-mail: shangl@purdue.edu)
** School of Mechanical Engineering & Department of Agricultural and Biological Engineering, Maha Professor of Fluid Power Systems, Director Maha Fluid Power Research Center, Purdue University
1500 Kepner Dr., Lafayette, IN, 47905 U.S.A
(E-mail: mivantys@purdue.edu)

Abstract. The fluid film behavior in the lubricating gap between piston and cylinder bore in axial piston machines is the main focus of this paper. The thermal behavior of the compressible viscous flow in the thin lubricating film formed between piston and cylinder bore plays a critical role on the interface performance and energy dissipation, therefore, deserves a thorough analysis. The temperature distribution in the fluid domain, as well as the heat flux from the fluid domain to the solid domain, follow the first, and the second laws of thermodynamics, however, are difficult to solve due to the constantly changing boundary conditions. The proposed fluid domain thermodynamic model calculates temperature distribution in the gap flow and the heat flux to the solid parts with a higher accuracy than the currently used fluid domain heat transfer model.

Keywords: Thermodynamic, Temperature distribution, Energy equation, Heat transfer, Piston/cylinder interface

INTRODUCTION

The temperature distribution in the fluid domain of the main lubricating interfaces of axial piston machine is always a key aspect of the fluid film behavior. Not only the fluid properties but also the heat flux applying on the running surface of the solid parts that form the fluid film have critical influences on the energy dissipation and the leakage occurring in this lubricating interfaces. In fact, the fluid temperature distribution is a result of the energy dissipation due to the viscous friction, the convection due to the gap flow, the fluid pressure changing with respect to time and space, the conductivity of the fluid, and the conduction between the fluid domain and the solid domain. A fluid and structure interaction problem is then created by the conduction and thermal deflection on the running surface of the solid domain. The temperature distribution in both the fluid domain and the solid domain interact with each other. And, the thermal deformation of the solid bodies under the thermal load changes the shape of the boundary of the fluid film, impacts the pressure build up in the fluid film, therefore the energy dissipation, the gap flow, and the fluid temperature.

The main objective of the paper is to study the fluid temperature distribution in the lubricating interface between piston and cylinder bore in axial piston machine. In the past, many researchers have tried to address this fluid structure and thermal interaction problem for piston/cylinder interface with more or less simplification of the complex physics. Ivantysynova [1][2] published a modeling approach that firstly solves the pressure distribution in the piston/cylinder interface using Reynolds equation for non-isothermal fluid. The viscosity in the fluid domain changes with the temperature, which is calculated considering conduction, convection, and the energy dissipation due to the viscous shear. Olems [3] solved piston micro motion fulfilling piston body force balance between the external load and the fluid film pressure force considering squeeze motion. His model was developed based on the previously published non-isothermal model proposed in [1][2]. Ivantysynova and Huang [4] added the elastic deformation due to pressure into Olems’s model. Their model captures the influence of the pressure deformation of the solid bodies on the fluid film behavior including the pressure distribution in the gap. Therefore, the fluid model and the solid parts deformation are solved using an iterative scheme. The pressure deformation in their model is calculated using an influence matrix approach. Pelosi and Ivantysynova [5] further improved the modeling ability of piston/cylinder interface by adding a solid domain heat transfer model and a thermal elastic deformation model. In their model, the fluid temperature distribution generates heat fluxes on both the piston and the cylinder bore running surface. The heat flux on each body then acts as thermal boundaries on a three-dimensional heat transfer model that solves the solid domain temperature distribution. This thermal load can be calculated from the resulting temperature fields, and be used to calculate the solid body thermal deflection. The deformation of the running surface of the solid body changes the shape of the fluid film.
The temperature on the solid body running surface controls the boundary temperature of the conduction calculation in the fluid domain. This fluid structure and thermal interaction problem is solved in their model using another iterative loop. Shang and Ivantysynova [6] published a port and case flow temperature prediction model that calculates the temperature in displacement chamber, inlet/outlet port volume, and the case volume. Those temperatures are essential thermal boundaries for the three-dimensional solid body heat transfer model. Their model put the last piece in the puzzle, enables the piston/cylinder interface fluid film behavior modeling without any support from measurements.

Shang and Ivantysynova [7] created a temperature adaptive piston design utilizing the previously described model. They proved that changes of the thermal behavior of the solid body including the thermal expansion coefficient and the heat transfer coefficient have critical impacts on the piston/cylinder interface performance. Their proposed bi-material piston showed improvement on energy dissipation with inlet temperatures in the range between -20°C and 100°C.

Currently, the fluid temperature modeling approach of the state-of-the-art piston/cylinder interface does not consider the temperature change due to the fluid pressure change with time, or gap flow through pressure gradient. However, Cheng and Sternlicht [8] and Cheng [9] already included the fluid velocity through pressure gradient in the source term of the energy equation to solve the thermal effect between two rolling and sliding cylinders. In their thermal analysis, the first law of thermodynamics was fulfilled. Burton [10] analyzed the thermodynamics of a viscoelastic film under shear and compression. His study combined the first and the second law of thermodynamic. Xu and Sadeghi [11] reported their thermal EHL analysis of circular contacts with measured surface roughness. A three-dimensional time dependent non-dimensional energy equation for the lubricating fluid was proposed which includes the local temperature change with respect to time.

In this article, a more completed numerical fluid film thermodynamic model is firstly proposed for the piston/cylinder interface for axial piston machine. The fluid temperature distribution is solved not only considering the convection, conduction, and energy dissipation, but also considering the temperature change with time, the load pressure changing with time, and the flow through pressure gradient. Both the first and the second law is fulfilled in this proposed thermal model.

In the first part of this paper, the energy equation which is the combination of the first and the second laws of thermodynamics is analyzed focusing on the compressible viscous flow. The concluded physical relationships, was then constructed with the finite volume method, resulting in a thermodynamic model for the given boundary conditions. In the second part of this paper, the proposed thermodynamic model was inserted into a fluid structure and thermal interaction model developed in authors’ research group, which allows the prediction of the fluid behavior in the main lubricating interfaces in a swash plate type axial piston machine considering the elastic deformation on the solid parts under both the pressure and thermal load, and the heat transfer in both the solid, and fluid domain. The proposed fluid domain thermodynamic model calculates the fluid domain temperature distribution and the heat flux to the solid domain with a higher accuracy than the currently used fluid domain heat transfer model. In the third part of the paper, the simulated flow thermal behavior are compared between the proposed and the current model. The simulated local temperature in the solid domain is also compared to the measured film temperature from a modified axial piston machine.

**TIME DEPENDENT ENERGY EQUATION FOR COMPRESSIBLE VISCOUS FLOW**

The foundation of the fluid film thermodynamic model is the energy equation that links the rate of local temperature change to the rate of local energy dissipation, to the rate of the local pressure change, and to the fluid conduction, the fluid convection, the fluid flow through pressure gradient. In order to solve the heat transfer problem numerically, the fluid domain between the piston and the cylinder bore is discretized to a three-dimensional structured numerical domain as shown in figure 1(a). For each three-dimensional finite volume as shown in figure 1(b), there are six faces labeled ‘t’, ‘b’, ‘w’, ‘e’, ‘n’, and ‘s’ separate the volume from its neighbors located at ‘T’, ‘B’, ‘W’, ‘E’, ‘N’, and ‘S’. The mass in the control volume is changing due to the mass flow rate through the faces ‘w’, ‘e’, ‘n’, and ‘s’. The fluid velocity on the gap height direction is assumed to be zero. The temperature and the pressure in the control volume are taken from the centroid. The local pressure as well as the pressure in its neighbors are changing due to the changing displacement chamber pressure, the piston sliding and spinning motion, and the piston micro squeezing motion. The pressure is calculated also considering the deformation of the solid bodies, and the fluid properties changing with pressure and temperature. The local temperature is changing due to the conduction on the six faces, the convection with the mass flow rate, the local pressure changing, and the mass flow rate through the pressure gradient. In order to model the previous described problem, an energy equation is used:
\[
H = H_0 + \Delta H
\]
\[
h \cdot m = h_0 \cdot m_0 + \left( \frac{dh}{dt} \right) \cdot m_0 + h_\text{i} \cdot \dot{m}_\text{i} - h_\text{m} \cdot \dot{m}_\text{m} + \mu \cdot \Phi_D \cdot dV + \sum \lambda_i \left( \nabla T \right)_i \cdot A_i \cdot dt
\]

The term on the left hand side of the energy equation expresses the total enthalpy of the control volume at the current time step. The terms on the right hand side are expressing the total enthalpy at the previous time step, the enthalpy changing with the in and out going mass flow, the energy dissipation due to the viscous friction, and the conduction on each face. The specific enthalpy in the proposed energy equation is a function of pressure and temperature:

\[
h = h_0 \left( T_0 \right) + \int_{t_0}^{T} c_p \, dT + \int_{p_0}^{p} T \left( \frac{\partial V}{\partial T} \right) \, dp + \int_{p_0}^{p} \nu \, dp
\]

Combining Eq. 1 and Eq. 2, the local temperature yields to a function of neighbor’s temperature and the source term:

\[
T_p = f \left( T_T, T_B, T_N, T_S, T_W, T_E, s \right)
\]

Where the source term is a function of the local previous temperature, current and previous pressure, and the neighbors’ pressure:

\[
s = f \left( T_s, P_s, P_p, P_T, P_B, P_N, P_S, P_W, P_E \right)
\]

The source term is pre-defined before the temperature distribution calculation. And the fluid domain heat transfer problem is solved through a Gauss-Seidel over-relaxed method with the solid bodies’ surface temperature as boundary conditions.

Comparing to the previous energy equation proposed by Ivantysynova [2], this presented thermodynamic model solves the temperature changing rate instead of absolute temperature. Therefore, an initial condition at time equal to zero is needed, and for the research study presented in this publication, it is set to be the linear temperature distribution between displacement chamber temperature and the case temperature.

**FLUID STRUCTURE AND THERMAL INTERACTION MODEL FOR PISTON CYLINDER INTERFACE UTILIZING THE PROPOSED THERMODYNAMIC MODEL**

The fluid structure and thermal interaction model developed in authors’ research group allows the prediction of the fluid behaviors in the piston/cylinder interface. As shown in figure 2, the fluid pressure and temperature are solved through a finite volume method. The Reynolds equation is discretized in a two-dimensional grid to calculate the pressure distribution in the gap area. The pressure deformation on the solid bodies is then calculated using an off-line influence matrix method. The fluid structure interaction problem can be solved using an iterative method. And the converged pressure and the fluid film thickness are used by the proposed thermodynamic model to calculate the fluid temperature distribution and update the fluid properties. The heat flux on the fluid/solid domain boundary can be calculated from the resulting fluid temperature gradient at each
time step. An average heat flux on the running surface of both piston body and cylinder block body for the whole revolution is obtained after each simulated revolution. At the end of each revolution, the solid bodies’ temperature is calculated based on the average heat flux, and the thermal deflection is then calculated based on the body temperature. This thermal deformation is used to update the fluid film shape for the next revolution, and the converged heat flux and body temperature can be obtained using an iterative method.

**FIGURE 2.** Simulation scheme for piston/cylinder interface.

Figure 3 shows a cross section of the resulting pressure and temperature distribution from the above-described model with the proposed thermodynamic model. The piston spins counter-clockwise relative to the cylinder bore and drags the fluid from zone C through zone B into zone A. As shown in the left-hand side of figure 3, the pressure is building due to the wedge shape of the fluid film and the piston spinning motion. The pressure of the fluid flow through zone B increases, as well as the fluid enthalpy. The high energy dissipation in zone A due to low film thickness also increases the fluid enthalpy. The increased enthalpy is represented by the fluid temperature as shown on the right-hand side of figure 3. The high temperature in zone C is due to the convection and conduction from zone B. Due to the low film thickness in zone B and zone A, more energy is transferred to the solid parts.

**FIGURE 3.** Cross section for pressure and temperature distribution in piston/cylinder lubricating interface
Comparing the temperature distribution in the piston/cylinder interface using the previous model that proposed by Pelosi and Ivantysynova (2012), the new thermodynamic model calculates higher fluid temperature thanks to the consideration of the pressure changing with respect to time and space.

FIGURE 4. Temperature distribution comparison between previous model by Pelosi and Ivantysynova [12] and the proposed new thermodynamic model.

COMPARISON OF SIMULATION RESULTS TO MEASUREMENT USING THE EHD TEST RIG

The EHD test rig has been designed and built by Ivantysynova et al. [12]. The specially designed pump for the EHD test rig, as shown in figure 5, is equipped with nine thermocouples and nine piezoelectric pressure sensors. The EHD pump is designed to achieve the same piston kinematics and dynamics as a commercial axial piston machine. Instead of a rotating block with nine pistons, the EHD pump is designed with a wobbling swash plate and a single piston stationary cylinder block. As shown in figure 5, the shaft rotates the swash plate and drives the piston in and out from the cylinder bore. The thermocouples are placed in the cylinder block pointing to the lubricating gap. The diameter of the tip of the thermocouple is 0.5 mm. The tip of the thermocouples are touching the fluid film, and therefore, forming part of the running surface of the cylinder bore. The nine thermocouples are placed 2.5mm, 3.0mm, 5.0mm, 8.0mm, 14.33mm, 20.66mm, 23.66mm, 25.66mm, and 26.16mm from the displacement chamber end of the running surface of the cylinder. The thermocouple closest to the case end is 2.5mm from the end of the running surface.

As shown in figure 5, the thermocouples are placed 45° apart. The first and the last thermocouple laid on the same circumferential position but 23.16mm apart axially. A locking device fixes the cylinder block at 180 different angular positions. Therefore, the temperature of the cylinder bore running surface can be measured at 1620 different locations.

The piston that was used for the measurement has a barrel shape as proposed by Ivantysynova and Lasaar [14]. This surface shaping has been considered in the simulation. The operating conditions of the measurement are
listed in Table 1. The temperature of the case flow and at the high pressure port were measured when the pump reached steady state conditions.

<table>
<thead>
<tr>
<th>Operating conditions</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Shaft speed</td>
<td>1000 [rpm]</td>
</tr>
<tr>
<td>Differential Pressure</td>
<td>150 [bar]</td>
</tr>
<tr>
<td>Case Temperature</td>
<td>55.0 [°C]</td>
</tr>
<tr>
<td>Temperature at High Pressure Port</td>
<td>45.0 [°C]</td>
</tr>
<tr>
<td>Temperature at Low Pressure Port</td>
<td>43.0 [°C]</td>
</tr>
</tbody>
</table>

When solving the solid body heat transfer problem, two types of boundary conditions are used on both the piston and cylinder of the EHD pump as shown in Figure 6. The mixed boundary applies heat flux on the surface of the solid bodies that is calculated from the environment temperature and the convection coefficient. The Neumann boundary applies heat flux that is calculated from the normal gradient of temperature at the surface.

![Figure 6. Boundary conditions for the solid body heat transfer model, Pelosi and Ivantysynova (2012)](image)

Figure 7 shows the comparison result between the measurement and simulation. The temperature distribution of both the measurement and simulation show similar absolute value and trends. The simulation is able to catch the hot spot near the case end. However, the measured hot spot at displacement chamber (DC) end is not represented in the simulation. Because of the unexpected derivation of the simulated temperatures from the measured profile, the authors very recently re-measured the surface of the cylinder and piston using a surface...
profilometer. These recent measurements did show that the surface profile of the cylinder bore has changed compared to the profile which was measured before the temperature measurements were carried out, however the original measured surface profile has been used for the simulation. The authors are still rerunning simulations and will present and discuss the new results at the conference.

CONCLUSION

The proposed thermodynamic model for the fluid domain in the piston/cylinder interface added the change of pressure with respect to space and time and rate of temperature change into model. The simulated temperature distribution comparison result between the previous and the proposed model show a significant difference. The previous model underestimated the temperature due to the fact of neglecting compression heat. The simulated temperature distribution using proposed thermodynamic model was compared to the measurement. The comparison result verified that the new model is capable of calculating the temperature distribution in the fluid domain of piston/cylinder interface with a very good accuracy.

REFERENCES

INTERACTION BETWEEN SWASH PLATE MOVEMENT AND COMMUTATION IN AXIAL PISTON MACHINES

Florian SCHOEMACKER* and Hubertus MURRENHOFF*

* Institute for Fluid Power Drives and Controls (IFAS)
RWTH Aachen University
Steinbachstr. 53, 52074, Aachen, Germany
(E-mail: florian.schoemacker@ifas.rwth-aachen.de)

Abstract. In a variable displacement pump the swash plate has the ability to oscillate because of the excitation of the piston pressure forces. The odd number of pistons and the alternating piston pressure produce a periodically changing torque load. The resulting swash plate movement alters the piston stroke and therefore interacts with the piston pressure gradient. In this study a simulation model of the dynamics of a variable displacement pump is developed and the interaction between the swash plate movement and the commutation is investigated. The torque load influence of different valve plate designs is analyzed and presented in this paper.

Keywords: axial piston pump, swash plate torque load, swash plate oscillations, valve plate design, commutation

INTRODUCTION

The system “hydraulic variable displacement pump” consists of mainly two components. One is the pump itself with the cylinder block, pistons, valve plate and swash plate. The other part is the pump controller including the control valve and control actuators. The design of the pump controller depends on the function of the pump system, for example pressure compensation or flow control. In order to change the displacement volume, the pump controller acts on the swash plate via the control piston. During steady-state operation, the resulting force of the control piston is in equilibrium with the torque load of the piston pressure forces acting on the swash plate. Because of the odd number of pistons and the alternating piston pressure, these forces produce a periodically changing torque. Current research has shown that the torque load results in a movement of the swash plate and a pressure variation in the control actuator. Achten et al. [1] showed that the torque load on the swash plate results in a movement of the swash plate. The study was based upon the floating cup principle, which is an axial piston pump with two cylinder blocks and two swash plates, and includes simulations as well as measurements. The results show that the oscillation of the swash plate angle eventually changes the sinusoidal movement of the pistons at the dead centers, which results in a variation of the piston pressure gradient during the commutation between the pressure levels. Furthermore the pump’s efficiency is reduced in controlled operation because of additional volume flow in the pump controller. The swash plate oscillations cause a pressure variation in the control actuator and therefore the flow in the pump controller.

Ericson [2] has investigated the swash plate oscillation for a conventional swash-plate type axial piston pump. The simulation of the pump model showed a variation of the piston stroke during commutation. Due to the dynamics of the swash plate, the amplitude of the oscillation depends on the pumps rotational speed. Slower speeds result in a higher amplitude because of the low natural frequency of the swash plate. However the pump’s output volume flow is only slightly affected by the swash plate oscillations. For the simulation of the swash plate oscillations a detailed model of the pressure built-up dynamics during commutation as well as a a pump controller model is needed. Manning and Johnson [3] published a mathematical description of the model for a variable displacement pump. Manning [4] also studied the forces acting on the swash plate and their variation due to the odd number of pistons. For the simulation of a control operated pump, Mandal et al. [5] developed a model of the pressure compensator for a variable displacement pump. The model was used for designing the pump controller according to the dynamics of the swash plate. Within a research project at IFAS, pump controller losses were investigated, showing an efficiency reduction of pumps in closed-loop control [6]. The power loss is almost constant for all swash plate angles and differs with the pressure level.
For their simulations all authors used valve plate geometries with silencing grooves. In this study the impact of the valve plate design on the torque load and therefore on the movement of the swash plate is investigated. The thesis is that the valve plate design has a strong impact on the torque load which then causes the swash plate movement. As stated before, the swash plate movement interacts with the commutation. In order to investigate the effect of the valve plate design on this interaction, different valve plate designs using silencing grooves and bores are simulated. Therefore a new simulation model with the ability to calculate pump dynamics including the swash plate torque load and oscillations has been developed.

**GEOMETRY**

For the analysis a swash plate-type axial piston pump has been chosen as a variable displacement controlled pump. The pump controller type is a pressure compensator which adjusts the output flow according to the hydraulic system’s need in order to hold constant pressure. The pump’s geometric data is derived from a fictional axial piston pump with a power output of about 30 kW. Fig. 1 shows the assembly of the pump system and a sketch of the hydraulic system used for displacement control. Forces acting on the swash plate are displayed as well.

![Pump system with pressure compensator](image)

**FIGURE 1.** Pump system with pressure compensator

The piston pressure forces act on the swash plate creating a periodically changing torque load. A spring provides the swash plate with an initial torque load for swiveling out if the pump is in unpressurised condition. The torque load on the swash plate is balanced by the control actuator’s force. The actuator is supplied by the pump controller which changes the swash plate angle in control operation.

The pump model consists of the piston assembly’s mathematical description and its commutation to the high and low pressure kidney. The mathematical calculation is based upon a Cartesian coordinate system which is shown on Fig. 1 and Fig. 2. The valve plates in Fig. 2 are displayed with their contact surface for the cylinder block.

![Different valve plate designs](image)

**FIGURE 2.** Different valve plate designs (VP 1: no commutation element, VP 2: grooves, VP 3: bores)

The used definition of the coordinate system and the rotational angle $\phi$ is chosen to be able to calculate using the vector product. Also this definition has the outcome that the zero angle for the rotational angle $\phi$ lies within the high pressure kidney. As stated before the design of the valve plate is taken into account for the study. The designs of the valve plates used for the simulation are shown in Fig. 2.
In order to compare the results of the calculation the opening area of the commutation between the piston bore and the valve plate is the same for the grooves and the bores. The starting angle of commutation is the same and the finally opened area has the same value. The different graphs for the valve plates are shown in Fig. 3.

![Figure 3](image-url)  
**FIGURE 3.** Commutation area opening for VP 1, VP 2 and VP 3

Valve plate 1 does not have any commutation element. Therefore the gradient of the opening area depends solely on the radius and the width of the kidneys. The gradient is almost constant. Especially at the beginning of the commutation a high backward volume flow from the high pressure port into the piston’s displacement chamber occurs. Therefore the pump’s output flow pulsates quite strongly.

The silencing groove of VP 2 provides a quadratic rising of the opening area. Therefore the area increases gradually. In hydraulic terms this means that the resistance for the backward-flow from the high pressure port into the displacement chamber is quite high at first and decreases afterwards while the pressure difference between piston chamber and high pressure port drops. Eventually an almost constant backward-flow occurs. Valve plate 3 with the silencing bore offers a quick opening of the entire bore as a commutation area which is held constant until the opening in the cylinder block hits the high pressure port. Therefore the resistance of the commutation is held constant and the backwards volume flow only depends on the remaining pressure difference.

**SIMULATION MODEL**

For the 1D-simulation the pump and controller components have been modelled. Each individual piston stroke is calculated via the mathematical function containing the rotational angle and the swashplate angle (1). Due to the aforementioned effects, the swash plate angle can no longer be taken as a constant value and leads to further terms in the derivation of the velocity (2) and the acceleration (3).

\[ s_i = R \cdot \tan(\beta) \cdot \sin(\varphi_i) \]  
\[ v_i = \dot{s}_i = R \cdot \left[ \sin(\varphi_i) \cdot \left( \dot{\beta} \cdot (1 + \tan^2(\beta)) \right) + \cos(\varphi_i) \cdot (\dot{\varphi} \cdot \tan(\beta)) \right] \]  
\[ a_i = \ddot{s}_i = R \cdot \left[ \cos(\varphi_i) \cdot \left( 2 \cdot \ddot{\beta} \cdot \dot{\varphi} \cdot (1 + \tan^2(\beta)) \right) + \sin(\varphi_i) \cdot \left( -\dot{\varphi}^2 \cdot \tan(\beta) + 2 \cdot \beta^2 \cdot \tan(\beta) \cdot (1 + \tan^2(\beta)) + \ddot{\beta} \cdot (1 + \tan^2(\beta)) \right) \right] \]

For each piston the stroke is calculated with \( \varphi_i = \varphi + i \cdot \Delta\varphi_i \).

The connection between the piston chamber and the kidney-shaped openings of the valve plate determines the pressure gradient which is calculated using (4) and (5).

\[ p_i(\varphi_i) = \frac{\dot{E}_{ol}}{V_i(\varphi_i)} \cdot \left( Q_i + A_{piston} \cdot v_i(\varphi_i) \right) \]  
\[ Q_i = \alpha_D \cdot A_i(\varphi_i) \cdot \sqrt{\frac{2 \cdot (p_{HD} - p_i(\varphi_i))}{\rho}} \]

The pressure is a periodically changing function which is displayed in Fig. 4 for all valve plates.
Furthermore the resulting piston pressure forces and torque $M_{\text{x,i}}$ on the swash plate are calculated.

$$M_{\text{x,i}} = R \cdot (1 + \tan^2(\beta)) \cdot A_{\text{piston}} \cdot p_i(\varphi_i) \cdot \sin(\varphi_i)$$  \hspace{1cm} (6)

In addition to the piston pressure forces, the spring and actuator forces act on the swash plate. Finally the angle and angular velocity of the swash plate are determined using Euler’s laws of motion.

$$J_{\text{SP}} \cdot \ddot{\beta} + d_{\text{SP}} \cdot \dot{\beta} + c_{\text{SP}} \cdot \beta = \sum M_{\text{x,i}} + h \cdot A_{\text{C}} \cdot (\alpha \cdot p_{\text{HD}} - p_A)$$  \hspace{1cm} (7)

The swash plate angle, its velocity and acceleration are then used as an input for the piston kinematics. For the pump controller a simulation model of a pressure compensator including the spool dynamics is used.

Following boundary conditions are used for the simulation: rotational speed 1500 rpm, high pressure 300 bar and a swash plate angle of 15°.

**SWASH PLATE TORQUE LOAD**

Utilizing the simulation, a detailed calculation of the forces acting on the swash plate is conducted. The torque load can be used as a result in order to compare different commutation designs. Fig. 5 shows the piston pressure and the resulting torque load of the piston on the swash plate $M_{\text{x,i}}$ for one revolution. The torque load is calculated for a mechanically locked swash plate to show the initial torque load. The graphs are displayed for the valve plate with the grooves and the bores.

As stated in (7) a positive torque leads to the swash plate swiveling out. Therefore the pressure distribution at the outer dead center has a significant impact on the torque load because it is the greatest decreasingly acting torque. The two commutation elements show a different pressure rise in the piston chamber and therefore an
altered torque load as well. The bore leads to a smoother pressure transition when getting connected to the high pressure kidney with the full area.

Fig. 6 shows the entire torque load of all pistons on the swash plate.

![Fig. 6. Torque load during one revolution (left VP2, right VP3)](image)

The calculation already shows a difference between the two valve plate designs. For VP 3 the difference between the peaks results from the unequal size of the bores for compression and decompression. This is done to lengthen the angle of high pressure in the piston chamber for a counter torque on the swash plate, as shown in Fig. 5, showing the possibility of adapting the design of a valve plate according to the means of a desired torque load. For example, the amplitude of the torque load could be reduced in order to minimize the excitation of the swash plate.

**SWASH PLATE MOVEMENT**

For a controlled pump the torque load is balanced with the control force of the actuator and other damping forces. A variation of the torque load leads to a movement of the swash plate. The control actuator is moved by the swash plate and therefore the chamber volume in the control actuator is changed. This causes a pressure variation which then acts as the control actuator force on the swash plate balancing the torque load. Therefore the swash plate needs to move in order to enable pressure built up in the control actuator. The oscillation of the swash plate angle for the two valve plate designs with commutation elements are shown in Fig. 7.

![Fig. 7. Swash plate angle (left: VP 2, right: VP 3)](image)

The amplitude of the oscillation is smaller than 0.1° and the curve for both valve plates do not differ significantly. The simulation is calculated without damping of the swash plate. Therefore the diagram shows the highest possible amplitude.

Achten et al. [1] and Ericson [2] stated a variation of the piston stroke especially in the dead centers. For this study, the piston stroke and the piston pressure during commutation is shown in Fig. 8 for VP 2.

![Fig. 8. Piston stroke and pressure during commutation for VP 2)](image)
The dashed lines display the stroke and pressure for a mechanically locked swash plate, for example in a constant displacement pump. For study of the swash plate oscillations this is equivalent to an ideal operation of the pump. The piston stroke is altered by the swash plate movement. This causes a change of piston pressure in two ways. On the one hand, the stroke variation alters the chamber displacement and therefore the piston pressure. On the other hand, the piston volume flow is changed by the altered piston velocity. Therefore the piston pressure gradient changes due to the altered flow through the commutation elements. At the inner dead center (on the left) the pressure gradient is significantly altered. According to Fig. 6 this reduces the torque load in decreasing direction of the swash plate angle.

FURTHER IMPACT AND EFFECTS

The torque load and resulting swash plate movement needs to be damped in order to stabilize the system. Because the torque is a force excitation on the swash plate, the realization of compensation via the force of the control actuator is rather complicated. In this case the pump controller would have to provide the pressure to the actuator at the exact time once the torque load occurs. Therefore the damping is realized mechanically by friction forces. The friction occurs in the swash plate bearing. Fig. 8 shows the calculated torque load with moderate damping for VP 2 (swash plate bearing: μ = 0.01, viscous friction: \( d_{SP} = 10 \text{ Nm} \cdot \text{s} \)).

The damping reduces the resulting torque load. Within the simulation a preferred value can be chosen but the damping also reduces the dynamic behavior of the pump system. So there is a conflict between a well damped, stabilized system and a high dynamic system which is needed for pressure compensation or flow control.
Ericson [2] states that the swash plate movement only minor affects the output volume flow of the pump. In order to prove this, the resulting flow at the high pressure output as a sum of all pistons’ volume flows is shown in Fig. 10.

![Fig. 10. Volume flow (above: locked, bottom: unlocked swash plate)](image)

For VP 1 a pulsation of the pump’s output flow for a locked swash plate occurs. Because the valve plate is not equipped with commutation elements, a high backward-flow from the high pressure port into the piston chamber results. The backward-flow then leads to a drop in the pump’s output flow which is shown in Fig. 10. This drop is reduced if the swash plate is unlocked. Furthermore the swash plate movement changes the output flow remarkably. The curves VP 2 and VP 3 show a difference in the output flow during commutation because of the different commutation area opening. The behavior is almost vice versa. The output flow of VP 2 and VP 3 is only slightly affected by the swash plate movement. Regarding the results of three valve plate, the impact of the swash plate movement of the pump’s output flow strongly depends on the valve plate design.

**CONCLUSION**

State of the art swash plate pumps operated with pump controllers show a different operating condition once the swash plate is not mechanically locked in its position. The torque load of the piston pressure forces is balanced with the control actuator force. The resulting movement of the swash plate eventually changes the sinusoidal stroke of the pistons and therefore the piston pressure gradient during commutation. This concludes that the intended design of the commutation behavior is altered once the pump works in controlled operation.

The torque load on the swash plate depends on the geometry and also on the piston pressure gradient during commutation. Silencing grooves and bores alter the pressure gradient differently which has a strong effect on the resulting torque load. The torque load needs to be compensated mechanically in order to reduce the swash plate movement. This also reduces the dynamics of the pump in controlled operation. The valve plate should be designed according to the usage of the pump. A constant displacement pump should be optimized for pulsation of output flow, but for a variable displacement pump the valve plate should be designed according to the control of the pump system. Especially for a displacement controller, the torque load does have a major impact on the function. Eventually, the study shows that there is still a high potential for further investigation into the design of valve plates for variable displacement pumps.
NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>( s_i )</td>
<td>Piston stroke</td>
<td>mm</td>
</tr>
<tr>
<td>( v_i )</td>
<td>Piston velocity</td>
<td>mm/s</td>
</tr>
<tr>
<td>( a_i )</td>
<td>Piston acceleration</td>
<td>mm/s²</td>
</tr>
<tr>
<td>( R )</td>
<td>Radius of piston</td>
<td>mm</td>
</tr>
<tr>
<td>( h )</td>
<td>Lever arm</td>
<td>mm</td>
</tr>
<tr>
<td>( V_i(\varphi_i) )</td>
<td>Piston displacement chamber volume</td>
<td>mm³</td>
</tr>
<tr>
<td>( A_i(\varphi_i) )</td>
<td>Opening area</td>
<td>mm²</td>
</tr>
<tr>
<td>( A_{piston} )</td>
<td>Piston area</td>
<td>mm²</td>
</tr>
<tr>
<td>( A_C )</td>
<td>Actuator area</td>
<td>mm²</td>
</tr>
<tr>
<td>( Q_i )</td>
<td>Flow rate of one piston</td>
<td>l/min</td>
</tr>
<tr>
<td>( p_{HD} )</td>
<td>High pressure</td>
<td>bar</td>
</tr>
<tr>
<td>( p_i(\varphi_i) )</td>
<td>Piston pressure</td>
<td>bar</td>
</tr>
<tr>
<td>( p_A )</td>
<td>Actuator pressure</td>
<td>bar</td>
</tr>
<tr>
<td>( E_{oil} )</td>
<td>Bulk modulus</td>
<td>bar</td>
</tr>
<tr>
<td>( a )</td>
<td>Area ratio</td>
<td>%</td>
</tr>
<tr>
<td>( \alpha )</td>
<td>Flow rate coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>( \beta )</td>
<td>Swash plate angle</td>
<td>°</td>
</tr>
<tr>
<td>( \varphi )</td>
<td>Rotational angle</td>
<td>°</td>
</tr>
<tr>
<td>( M_{Eigen,i} )</td>
<td>Torque load of piston</td>
<td>Nm</td>
</tr>
<tr>
<td>( J_{SP} )</td>
<td>Swash plate’s moment of inertia</td>
<td>kg·m²</td>
</tr>
<tr>
<td>( d_{SP} )</td>
<td>Swash plate damping</td>
<td>Nm·s/rad</td>
</tr>
<tr>
<td>( c_{SP} )</td>
<td>Spring ratio</td>
<td>Nm/rad</td>
</tr>
</tbody>
</table>

REFERENCES

Oral Presentation | Oil hydraulics

[2A08-12] H7 (Control & Measurements 1)
Chair: Hironao Yamada (Gifu University), Wataru Kobayashi (Okayama University of Science)
Thu. Oct 26, 2017 1:40 PM - 3:00 PM Room A (ACROS Fukuoka)

[2A08] VARIABLE DISPLACEMENT ALTERNATING FLOW HYDRAULIC PUMP
*Kim Adair Stelson¹, Ryan Foss¹, Mengtang Li², Eric J. Barth², James D. Van de Ven¹ (1. University of Minnesota, 2. Vanderbilt University)
1:40 PM - 1:56 PM

[2A09] PERFORMANCE OF SPEED VARIABLE ASYMMETRIC PUMP CONTROLLED ASYMMETRIC HYDRAULIC CYLINDER
Long Quan¹, *Lei Ge¹, Bin Cheng Wang¹, Bin Li¹, Bin Zhao¹, Zhen Lu¹ (1. Key Lab of Advanced Transducers and Intelligent Control System of Ministry of Education, Taiyuan University of Technology)
1:56 PM - 2:12 PM

[2A10] SENSORLESS POSITION CONTROL OF DIRECT DRIVEN HYDRAULIC ACTUATORS
Tom Sourander¹, Matti Pietola¹, *Tatiana Minav¹, Henri Hänninen¹ (1. Aalto University)
2:12 PM - 2:28 PM

[2A11] HYDROSTATIC STEERING SYSTEM AND ENERGY SAVING EVALUATION IN IDLE REGIME
*Giorgio Paolo Massarotti¹, Pietro Marani¹, Massimiliano Ruggeri¹, Esteban Codina² (1. C.N.R. - Imamoter, 2. UPC. Universitat Politècnica de Catalunya. BarcelonaTech)
2:28 PM - 2:44 PM

[2A12] RESEARCH ON THE CONTACT PRESSURE CONTROL OF A DIE WEAR TESTER
*Chao Yang¹, Shigang Wang¹, Li Liu¹ (1. School of Mechanical Engineering, Shanghai Jiao Tong University)
2:44 PM - 3:00 PM
VARIABLE DISPLACEMENT ALTERNATING FLOW HYDRAULIC PUMP


*Department of Mechanical Engineering, Vanderbilt University
2301 Vanderbilt Place, Nashville, TN 37235, USA
**Department of Mechanical Engineering, University of Minnesota
111 Church St. SE, Minneapolis, MN, 55356, USA
(E-mail: vandeven@umn.edu)

Abstract. The variable displacement pump is a key component to eliminate metering valve throttling losses through displacement control. Yet existing variable displacement pumps are heavy, have poor efficiency at low displacements, and are axially long making common-shaft mounting challenging. This paper proposes a novel alternating flow hydraulic pump based on Constantinesco’s wave transmission theory. The alternating flow (AF) hydraulic pump varies the displacement by phase shifting pairs of oscillating pistons. The AF pump is highly efficient across a wide range of operating conditions, power dense, and axially short—allowing multiple pumps to be mounted on a common shaft.

Keywords: Variable Displacement Pump, Efficiency, Alternating Flow Hydraulics

INTRODUCTION

Traditional hydraulic circuits use metering valves to control the load, resulting in precise control and fast response. However, metering valve control dissipates significant energy, which is largely the reason that the average efficiency of hydraulic system is as low as 21% [1]. Load sensing (LS) circuits are widely used in industry to control the pressure source to match with the highest pressure load to minimize the throttling loss. An LS system does improve the system efficiency but when the required pressure levels from different loads are not close to each other, LS still suffers from throttling loss and poor efficiency. Instead of throttling the fluid and transforming the unnecessary power into heat, a more efficient alternative is to drive each actuator with a variable displacement pump and control the actuator with the pump displacement, termed displacement control. The modularity of displacement control brings another benefit. It makes the whole system more reliable than a single pump circuit with throttling valves. If one pump fails, the other axes are decoupled and still fully functional. Critical to displacement control is the performance and efficiency of the variable displacement pumps across a wide range of displacements and pressures. For multi-actuator circuits, where multiple pumps are driven on a common shaft, the package size and weight of the pumps is also important.

Significant research on existing variable displacement pumps has improved the maximum efficiency. However, the efficiency of conventional variable displacement pumps is poor at low volumetric displacements because the largest energy losses do not scale with output power [2]. In many applications, the pumps operate at low displacement for a large portion of the duty cycle, resulting in significant energy loss. Additionally, existing variable displacement pumps do not package well on a common shaft due to the long aspect ratio of the axial piston pump and the lack of a through shaft on a bent axis unit. These challenges motivate the development of a more efficient variable displacement pump with a form factor amenable to a common shaft.

ALTERNATING FLOW

Alternating flow (AF) hydraulics is a novel approach for hydraulic power transmission proposed by Constantinesco in 1922 [3]. In an AF system, a periodically varying pressure or flow source, with no net fluid flow, is used to transmit power. The most famous invention that utilizes the alternating flow concept is the propeller aircraft mounted machine gun synchronizer, which allows pilots to shoot between the spinning blades of the propeller. Constantinesco’s machine gun synchronizer became standard weapon equipment for aircraft in
the later part of World War I. Extensive research has been conducted in the past and the study of AF hydraulics reached its summit in the 1960s. A good summary of the history and research results of AF hydraulics is presented in [4]. The previous studies are focused on the applications with vibration or oscillatory actuators such as rock drills, road surface fatigue testing, combine harvesters, and riveting machines [5]. The lack of research interest in AF hydraulics today probably results from the sufficiency of conventional “DC” hydraulics. Nevertheless, the authors believe that the AF hydraulic power system deserves more attention, as has been the case with AC electric power transmission. With more research and proper design, AF hydraulic systems may reveal significant advantages leading to more widespread use.

ALTERNATING FLOW HYDRAULIC PUMP

Basic Concepts of AF Hydraulic Pump

A novel alternating flow (AF) hydraulic pump is proposed based on the theory of sonics originated by Constantinesco [3]. The AF hydraulic pump uses periodically varying flow sources with rectification to generate unidirectional flow. The AF hydraulic pump can be visualized as two sinusoidal oscillating pistons of equal displacement with a fluid conduit directly connecting the two cylinders, as shown in Figure 1. When the motion of the two pistons is in phase, the flow is combined. When the motion of the pistons is 180 degrees out of phase, they shuttle flow back and forth, resulting in zero output flow. Figure 2 illustrates three ideal cases where the phase-shift angles, \( \phi \), are 0°, 120° and 180°, respectively. The upper three figures show the total flow rate of the AF pump and the lower three figures are for two individual pump units forming the AF pump. Positive flow rate means output fluid from the AF pump while negative flow rate means inlet fluid from the tank.

\[
X = \cos \frac{\phi}{2}
\]

(1)

Two radial piston pumps form the configuration for the AF pump, Figure 4, where pairs of cylinders between the pumps are connected and the inlet and outlet flows are controlled with check valves. The two pumps share a common shaft. Rotating the case of one pump creates the phase shift angle. While check valves are used in early experimental work, active valves will allow operation as a pump or a motor. Motoring operation allows the unit to absorb regenerative energy to transfer through the common shaft to pumps driving other degrees of freedom, reducing the load on the prime mover and further improving system efficiency.

FIGURE 1. Alternating Flow Hydraulic Pump Concept.
The most efficient positive displacement hydraulic pumps are axial piston pumps, bent axis pumps, vane pumps and radial piston pumps. The axial piston pump does have a through shaft but its axially long shaft makes it difficult for multiple pumps to be mounted on common shaft. The variable swash plate axial pump has the same problem and it requires complex components such as a swash-plate and ball bearings. The bent axis pump is a
variation of the axial piston pump with low side loading, but cannot provide a through shaft. Vane pumps can only operate at relatively low pressure levels. The radial piston pump has an axially short through shaft. It can work efficiently across a wide range of conditions. However, very few variable radial piston pumps are currently commercially available except the Moog RKP.

An alternative to a mechanically variable displacement pump is to vary the flowrate of a fixed displacement pump through high speed switching of digital valves, termed digital displacement. The most common approach to digital displacement is flow diverting, where the actively controlled tank valve is held open for a portion of the upstroke of the piston, returning the fluid to tank. At a specified displacement fraction of the piston stroke, the tank valve is rapidly closed and the pressure valve is opened, sending flow to the load. While this approach eliminates the leakage and friction of the port plates of an axial piston or bent axis pump, it has several drawbacks. The valve transitions occur at high piston velocity, resulting in throttling energy loss across the partially open tank and pressure valves for a fraction of the piston stroke and generating water hammer creating noise and large flow pulsations [6, 7]. Viscous flow losses are incurred by pumping the unused flow back to tank, and there is a lack of digital valves with reasonable energy consumption that can switch fast enough for high-speed pumps.

The radial piston configuration of the AF pump provides a number of benefits. It is highly efficient and simple, requiring no port plate or swash plate. The radial piston pump has a short axial package, allowing multiple units to be mounted on a common shaft for multi-actuator displacement control, shown in Figure 5. Extremely high displacement density can also be achieved in a radial pump by using a multi-lobe cam to create multiple pumping strokes per revolution. This improves compactness, balances radial forces on the pump axle, and reduces the pump case rotation angle required for changing displacement.

FIGURE 5a. Multi-actuator Displacement Control System with Multiple AF Pumps Mounted on a Common Shaft.

FIGURE 5b. Three-actuator Displacement Control System with Three AF Pumps Mounted on a Common Shaft.
PROTOTYPE DESIGN

First Generation Prototype of AF Hydraulic Pump

To demonstrate the basic concept of this novel alternating flow hydraulic pump and validate the dynamic model developed for it [8], a first generation prototype was established and tested as shown in Figure 6 [9]. The first generation pump was constructed from two Cat Pump® inline triplex 3CP1120 piston pumps. Each pumping cylinder pair is connected via a rigid fluid pipe. The connecting pipe is designed with large diameter and short length to minimize energy loss and avoid complex hydraulic phenomena from reflected pressure waves.

![First Generation Prototype of AF Hydraulic Pump Based on Cat Pump® 3CP1120 Pump](image)

**FIGURE 6a.** First Generation Prototype of AF Hydraulic Pump Based on Cat Pump® 3CP1120 Pump: Connecting Pipe Structure.

![First Generation Prototype of AF Hydraulic Pump Based on Cat Pump® 3CP1120 Pump](image)


For simplicity, sprockets and chains were used to control the phase shift between the pumps. The sprocket-chain transmission allowed accurate measurements of the phase shift angle, but varying the displacement requires disassembling and reassembling the transmission.
Future Generation Prototype of AF Hydraulic Pump

To create a more compact AF hydraulic pump and shift the phase angle in real-time, a second generation prototype is proposed and designed based on the Bosch® PR4-3X pump, as shown in Figure 7. With modifications, the pump cylinders units can be separated from their common channel, creating the same configuration as the first generation prototype. To achieve variable displacement, the solid camshaft of the PR4-3X pump will be replaced with one where the two cam lobes can rotate relative to each other. One of the lobes will be driven by a hydraulic rotary vane actuator, which itself rotates at the same speed of the driving motor.

FIGURE 7. Second Generation Prototype of AF Hydraulic Pump Based on Bosch® PR4-3X Pump.

CONCLUSION

The basic principles of a novel alternating flow (AF) hydraulic pump are presented in this paper. The advantages of the AF hydraulic pump are compared with other efficient positive displacement hydraulic pumps. A first generation prototype has been built and tested. Its working principle is introduced and its disadvantages described. To overcome the shortcoming of the first generation prototype, a second generation AF pump is proposed with design details.

ACKNOWLEDGMENTS

This project is sponsored by the Center for Compact and Efficient Fluid Power (CCEFP) under NSF grant #0550834 with funding from the National Fluid Power Association (NFPA) Education and Technology Foundation. We also thank Cat Pump® for donating the experimental prototype.

REFERENCES

PERFORMANCE OF SPEED VARIABLE ASYMMETRIC PUMP CONTROLLED ASYMMETRIC HYDRAULIC CYLINDER

Long Quan, Lei Ge, Chengbin Wang, Bin Li, Bin Zhao, Zhen Lu

*Key Lab of Advanced Transducers and Intelligent Control System of Ministry of Education, Taiyuan University of Technology, Taiyuan, P.R.China
(E-mail: quanlong@tyut.edu.cn)

Abstract. Valves controlled cylinder are applied in many industry and mobile equipments due to its compact structure, high dynamic performance and easily to drive more than one actuators by a single pump. However, the energy efficient is low, which caused by the large throttling loss and the simultaneous actuation of the valves meter-in and meter-out control edges. These losses can be avoided by replacing the valve controlled system by a pump controlled system. To a typical pump controlled asymmetric cylinder system, a complex and big flow circuit is necessary to compensate the flow difference between the working ports of the cylinder. And also, the operation stability of the system will become worse when the load force direction changed frequently and the control chamber changing alternatively. This paper addresses the stability problem of pump controlled asymmetric hydraulic cylinder especially which works under four quadrants, and proposes a novel solution for it. The system under consideration utilizes a new designed asymmetric pump which can match the differential area of an asymmetric cylinder basically and was used to control the arm cylinder of excavator. To verify the feasibility of the new circuit, a multi-body dynamic model of the excavator with symmetric and asymmetric pump is constructed. Furthermore, the operating characteristics and energy efficiency characteristics of the arm with the new scheme based on the designed open-loop and closed-loop strategies are studied on a real excavator. The results show that there is no obviously velocity fluctuation with the asymmetric pump and the position controlled precision is satisfied. Compared with the independent metering circuit with pump and valve accordance control, the energy-saving ratio reaches to 75.3% during a working cycle.

Keywords: Asymmetric pump, Four-quadrant operating, Pump-controlled cylinder, Energy efficiency

INTRODUCTION

Hydraulic systems have been widely used in industry and mobile machines by virtue of high-power density and high speed of response with fast start, stop and speed reversal possible. It can be classified as the circuit type into valve controlled system and pump controlled system. Valve controlled system are applied in many machines due to its compact structure, high dynamic performance and easily to drive more than one actuators by a single pump. However, the advantage of such a simple hydraulic drive structure is always connected to its disadvantage of large energy losses, such as throttling loss. It is reported that only 35% of the pump output power was transferred to the actuators [1]. There are therefore many years’ efforts to develop hydraulic systems without throttle losses. Pump controlled systems also called displacement controlled system, can eliminate throttling loss completely and have proven themselves in practice for a long time [2-3]. The pump controlled system consists of pump controlled double rods hydraulic cylinder (motor) and pump controlled single rod hydraulic cylinder, Zhongyi Q. has given a review of the latest development of these circuit and control technologies [4]. It can be seen that pump controlled double rods hydraulic cylinder has been well-developed and has been an inherent part in industrial applications, for example in Airbus A380 [5-6]. Due to the small installation space requirements and big output force, at least 80% of the electro-hydraulic control system adopts the single rod cylinder as actuator. Additional challenge for pump controlled system is to compensate the unbalanced flow rates due to the asymmetry in the single rod cylinder.

There are several solutions for the unbalanced flow rates compensation problem: use of hydraulic transformer [7], use of secondary pump [8], and use of a single pump combined with pilot operated check valves, shuttle valves or solenoid valves. These systems are reported that the energy efficiency can be significant improved [9-10]. Long Q. put forward a circuit in which the cylinder was controlled by two variable speed pumps, and sum pressures control strategy was put forward to pressurize the cylinder chambers to improve the dynamic performance [11-12]. The circuit with a single pump combined with compensation system is a relatively simple solution to compensate for unbalanced flow rates compared with the two pumps scheme [13-15]. In recent years, professor Monika Ivantysynova and her team, studied the static and dynamic behavior, and control strategy of the pump
controlled single rod cylinder circuit. \cite{16-21} It could be shown that the circuit solutions utilizing a single pump combined with pilot operated check valves, shuttle valves or solenoid valves are able to achieve a good energy efficiency. However, these systems suffer from undesired and uncontrolled pressure and velocity oscillations, when the load is light or the load force direction changed frequently. To overcome this backward, some solutions are proposed, such as a predictive observer to provide sufficient lead time for feedforward control of actuator pressure \cite{22}, adopting two solenoid directional valves controlled with singular point perturbation theory\cite{23}, and adopting a pair of check valves and an On/Off valve to compensate the flow difference, a counterbalance valve to control the chamber pressure of cylinder \cite{26}. In summary, the two pumps scheme can compensate the unbalanced flow basically, unfortunately the auxiliary components will increase the cost and complexity of the overall system. And the asymmetrical volume flow cannot be easily compensated by a single pump and the asymmetrical flow will increase energy exchange in the check valve. Also, it suffers from undesired and uncontrolled pressure and velocity oscillations, when the load is light or the load force direction changed frequently. In order to eliminate these disadvantages, an innovative pump controlled architecture with a new designed asymmetric pump which can match the unbalanced flow of the cylinder is put forward. The new system is used to control the excavator arm cylinder which works under four quadrants. Innovative research and major contributions of this articles is that the smooth control of hydraulic cylinder position and velocity can be achieved by only a new designed asymmetric pump without complex compensation circuit and control strategies.

This paper is structured as follows. Firstly, the circuits principle of the single rod cylinder controlled by symmetric pump and asymmetric pump are given in Section II. In Section III, the multi-body simulation model is constructed and the dynamic working performance of symmetric and asymmetric pump controlled arm cylinder is analyzed based on the model. In Section IV, the strategy of position closed loop combined with velocity feed forward control is designed with only displacement feedback. Experiments are conducted to prove the working of the whole system and the energy efficiency of the new system is compared with a separate metering in and separate metering out system. Conclusion is made in Section V.

**WORKING PRINCIPLE OF PUMP CONTROLLED SINGLE ROD CYLINDER**

Recently, a single pump controlled single rod cylinder in the study region is one of the hot topic in pump controlled systems due to its simple structure, low costs and big output force. In case of these systems, there exist unequal flow rates at two ports of the cylinder due to its asymmetric structure. And when a conventional pump is used to control this type cylinder, either a deficient or excess flow rate is always formed in the closed circuit. Hence researches have been focused on the compensation method of the unequal flow rates and on improving the stability of the system. There are several solutions for the differential flow compensation problem: use of secondary pump combined with accumulator and valves, use of hydraulic transformer. Both these concepts and structures have the drawbacks of high investment costs, increased number of control element, and required complex control effort. This paper proposes a novel compensation solution based on a new designed asymmetric pump and the system stability can be improved without much control efforts and additional element.

**2.1 Conventional Pump Controlled Single Rod Cylinder System**

The circuit of the symmetric pump controlled single rod cylinder is shown in Fig.1. A servo motor is used to regulate the drive speed of a constant pump. The two ports of the pump are directly connected to the single rod cylinder, the cylinder velocity can be regulated by changing the speed of the motor. An essential part of this circuit is the low pressure compensation system, which consists of a small pump and an accumulator. Its functions include compensating for the cylinder unbalanced flow, compensating for the volumetric losses in the closed circuit and cooling of the hydraulic fluid, over two back to back connected hydraulic controlled check valves. And pressure relief valves are utilized to limit operating pressure.

![FIGURE 1. Pump controlled single rod cylinder with check valve balancing the flow](image-url)
According to the working condition and job demands, many hydraulic systems are required to operate in 4-quadrants. That means that the cylinder and pump can both work as a hydraulic pump or motor. Hence the working process of the pump controlled single rod cylinder system under 4 quadrants is described as follows.

Consider extending the cylinder under resistance load, the pressure in the cylinder rodless chamber is high than it in the rod chamber. So, the rodless chamber is named as control cavity. The pump sucks oil from cylinder rod chamber through port B, and then the pump discharge oil to cylinder rodless chamber though port A. The pilot operated check valve B will be open by the pressure in the rodless chamber. The compensation system supplies low pressure oil to cylinder rod chamber. \( D \) and \( n \) are the displacement and rationing speed of the pump, \( v \) is the velocity of the cylinder, \( A \) and \( \alpha A \) represent the area of the rodless and rod chamber of cylinder. Without considering the system leakage, the flow rate output the pump, \( q_A \), and the flow rate into the pump, \( q_B \), can be written as \( D n \). The flow rate input the cylinder rodless chamber, \( q_{A0} \), is about \( A \cdot v \), and it is equal to the flow rate of the pump port A, and so the velocity of the cylinder can be written as \( v = D n / \alpha A \). The flow rate output the cylinder rod chamber, \( q_B \), is about \( \alpha A \cdot v \). The compensation system should deliver a flow rate, \( q_{in} \) of \((1-\alpha)\cdot A \cdot v\) to the main line F. Consider retracting the cylinder under resistance load, the control cavity is the cylinder rodless chamber, the flow rate input the cylinder rod chamber is equal to the flow rate of the pump port B, and the velocity can be written as \( v = D n / (\alpha A) \). The compensation system should store a flow rate, \( q_{out} \) of \((1-\alpha)\cdot A \cdot v\) to the main line E. And also, consider extending the cylinder under overrunning load, the control cavity is the cylinder rod chamber, \( v = D n / (\alpha A) \), \( q_{in} = (1-\alpha) \cdot A \cdot v \). Consider retracting the cylinder under overrunning load, the control cavity is the cylinder rodless chamber, \( v = D n / A \), \( q_{out} = (1-\alpha) \cdot A \cdot v \).

It can be concluded that when the displacement and rotating speed of the pump is a constant valve, the compensation system should deliver a flow rate of \((1-\alpha) \cdot A \cdot v\) during extending process and store the same flow rate during the retraction process. And also, the cylinder velocity will change if the load condition changes.

2.2 Asymmetric Pump Controlled Single Rod Cylinder System

As mentioned above, for the conventional pump controlled single rod cylinder, due to the asymmetry structure of the system, the relationship between the cylinder velocity and pump rotating speed depends on the load conditions, so the cylinder velocity stability is worse when the load changes frequency. In order to solve this problem, we have introduced the principle of asymmetric valve controlled asymmetric cylinder system to the pump controlled single rod cylinder system based on a new designed asymmetric pump. Benefiting from the asymmetric structure of the pump, the flow rates of the cylinder and pump can be balanced basically. Fig.2 gives the working principle and photograph of valve plate and cylinder block of the new designed asymmetric pump.

![FIGURE 2. Working principle of the Asymmetric Pump and Photograph of the Valve Plate and Cylinder Block](image)

It can be seen that there are four assignment windows on the valve plate, named A, B, C and D. A and C are on a circle with a radius of \( R_1 \). B and D are on a circle with a radius of \( R_2 \). Windows A and B are connected to each other by the port \( A_1 \) on the pump end shell cover. Windows C and D are one-to-one correspondence with pump ports \( B_1 \) and \( C_1 \). There are 10 plunger chambers and they are divided into two groups averagely. At the bottom of the cylinder block, there are an inner annular array and an outer annular array. The pitch radiuses of these two annular arrays match with the slots A, B, C and D on the valve plate. As shown in Fig.2(b), the plunger chambers identified as \( S_i \) correspond to the outer annular array, and these five pistons suck and discharge oil from the outer annular array only. The plunger chambers identified as \( S_j \) correspond to the inner annular array, and these five pistons suck and discharge oil from the inner annular array only. So benefiting from the asymmetric structure of the flow distribution, the flow rates ratio of the three ports of the pump is about 1:0.5:0.5. And we can match the area ratio of single rod cylinder by changing the piston diameter or \( R_1 \) and \( R_2 \) to change the flow rate ratio.

The principle of asymmetric pump controlled single rod cylinder is given in Fig.3. A servo motor regulates the drive speed of the new designed asymmetric fixed displacement pump. The ports A and B of the pump
are directly connected to the single rod cylinder, and port \( C_1 \) is connected to an accumulator directly and tank though a check valve. And also, the accumulator is used to compensate the differential flow rate caused by leakage and to pressurize the low pressure chamber over the hydraulic controlled check valves. And pressure relief valves are utilized to limit operating pressure, respectively.

During the extending process, the pump sucks oil from cylinder rod chamber though port \( B_1 \), and from tank or accumulator though port \( C_1 \). Then the pump discharge oil to cylinder rodless chamber. During the retraction process, the pump sucks oil from cylinder rodless chamber though port \( A_1 \). Then the pump discharge oil to cylinder rod chamber and to accumulator though port \( B_1 \) and \( C_1 \). \( n \) is the rationing speed of the pump, \( v \) is the velocity of the cylinder, \( A \) and \( \alpha A \) represent the area of the rodless and rod chamber of cylinder. \( D_A, \gamma \) \( D_A \) and \( (1-\gamma) \) \( D_A \) are the displacements of the pump ports \( A_1, B_1 \) and \( C_1 \), and \( \gamma \) is the displacements ratio between the port \( B_1 \) and \( A_1 \). Consider extending the cylinder under resistance load and retracting the cylinder under overrunning load, the control cavity is the rodless chamber, and the rod chamber is pressurized by the accumulator though port \( C_1 \). Under these conditions, the cylinder velocity can be written as \( v = D_A \, n/\alpha A \). Consider extending the cylinder under overrunning load and retracting the cylinder under resistance load, the control cavity is the rod chamber, and the rodless chamber is pressurized by the accumulator though port \( C_1 \), the cylinder velocity can be written as \( v = \gamma D_A \, n/(\alpha A) \). As the displacement ratio \( \gamma \) is designed to equal to the area ratio of the rodless and rod chamber of cylinder \( \alpha \), the flow rates in and out of the pump and cylinder match to each other basically.

**MULTI-BODY DYNAMICS SIMULATION**

Because of the requirements on the installation space and output force, at least 80% of the electro-hydraulic control system adopts the differential cylinder as actuator, such as excavator and press machine. In many applications and working conditions, both the value and the direction of load tend to change during working process, such as excavator arm cylinder. For the four quadrants working cylinder, in the valve controlled system, the method to improve the control performance, is increasing the pressure of return oil which will cause a large energy loss. So the new asymmetric pump is applied to control the excavator arm cylinder which is a typical four quadrants operating actuator. The control performance and energy efficiency are studied in this paper.

### 3.1 Pump Controlled Arm Cylinder Model

Unlike the working condition of the conventional hydraulic cylinder, the load exerted on the cylinder used in the mobile machine varies in large range and is unpredictable. It is difficult to establish the dynamic mathematical model of such system. In order to have a good knowledge about the working performance of the new designed system and the actuator, benefiting from the computer simulation technique, a multidisciplinary and multi-body dynamics model of the machine is constructed. And also, the performance of symmetric pump controlled system is studied for comparing.

A simulation model is created in order to analyze the symmetric pump controlled system and to implement new concepts into the existing machine on a virtual level. A multi-body model coupled with the hydraulic model determines the forces that act on the actuators. The moment of inertia and mass of the mechanical are taken into account and also the force acting on the bucket can be transmitted to other actuators in real time. The simulation model is based on SimulationX, which is based on the open source language Modelica. The model can be seen in Fig.4 below.

![Figure 3. Principle of Asymmetric Pump Controlled Single rod cylinder](image-url)
As shown in Fig.4, the prototype consists of the hydraulic excavator mechanical structure such as boom, arm, bucket and swing and so on, and electro-hydraulic system is constructed based on the circuit shown in Fig.3. For a real variable speed pump controlled system, the response of the servo motor is enough and it can be simplified to a simpler model to reduce the simulation time without much compact on the performance. The new designed asymmetric pump is a detail model based on detailed geometry model using single piston structure. And a small fixed gear pump combined with accumulator is used to balance the unbalanced flow rates caused by the leakage and volume loss. Parameters of the shuttle valve are included based on the real structure in the system model, rather than treating it as an ideal switching element as handled in literature. The model can be modified to a symmetric pump controlled system by replacing the asymmetric pump and increasing the displacement of the gear pump. The mechanical structure and asymmetric model has been verified in the team’s previous research work [25-26].

In the model, the arm cylinder stroke is about 720 mm, and the diameters of the piston and piston pole are 85 mm and 55 mm. The displacements of the asymmetric pump are 40 mL/r, 20 mL/r, 20 mL/r. The flow rate of the compensation system is about 5 L/min.

### 3.2 Simulation Results

Fig.5 gives displacements, velocities, pressures of the arm cylinder controlled by symmetric pump and asymmetric pump. As shown in Fig.5 (a), during 0~1 s, 6.5~8 s and 13.5~15 s, there is control signal and the cylinder does not work. At the time 1 s, the motor speed is set as 700 r/min. During 1~3.6 s, the arm cylinder extends under overrunning load, and the control chamber is the rod chamber, the velocity is about 154 mm/s. Along with extending, the load decreases, and at the time 3.6 s, load goes to 0 kN, and the pressures in the cylinder two chambers are equal which can not open the shuttle valve. Then the velocity slows down to about 76.5 mm/s. And then the cylinder works under resistance model. At the time 8.0 s, the motor speed is set as -700 r/min. During 8.0~9.2 s, the arm cylinder retracts under overrunning load, and the control chamber is the rodless chamber, the velocity is about 77.5 mm/s. Along with retraction, the load decreases, and at the time 9.2 s, load goes to 0 kN, and the pressures in the cylinder two chambers equal compensation pressure. Then the velocity goes up to about 76.5 mm/s. And then the cylinder works under resistance model.

Fig.5(b) gives displacement, velocity, pressures of the arm cylinder controlled by asymmetric pump. It can be seen that the cylinder velocity is comparatively steady during extending and retracting. And also, hydraulic cylinders are still running at the same velocity even the control chamber changes.
Mention above, the asymmetric pump can balance the unbalanced flow rate of the asymmetric pump, and also the velocity and pressure oscillations caused by the changing of the control chamber can be avoid. However, it is difficulty to match dynamic flow of the pump and cylinder due to the leakages. So the matching characteristics between the pump and the cylinder is studied based on the model by changing the displacement of the port B. Fig.6 gives the working performance of the system when $\gamma_1=95\% \gamma_2$ and $\gamma_1=105\% \gamma_2$. According to the curves in Fig.6, when $\gamma_1<\gamma_2$, the pressures and velocity are comparatively steady, and when $\gamma_1>\gamma_2$, the pressures is similar to the pressures of symmetric pump controlled system.

### CONTROL STRATEGY AND EXPERIMENTS

#### 4.1 Control Strategy

Velocity control is widely used in hydraulic excavator. The operator gives the velocity command by operating the joystick. And then the displacement or rotational speed of the pump changes. The actuator works at the demand velocity. When the cylinder approaches the demand location, the operator relieves the joystick. The actuator stops.

But only adopting an open loop to control arm velocity may result in no position accuracy and poor anti-interference ability, the working performance relies on the operator. And it is difficult to realize automatic excavation. For these problems, a strategy of displacement closed loop control is designed to realizes accuracy control of the cylinder, as shown in Fig.7. However, the velocity can not be controlled using only displacement closed loop control. So the displacement closed loop control combined with velocity feedforward is put forward.

On the base of position control system, the velocity feedforward is introduced to control the velocity. When the difference between the desired position and the real position is large, the velocity feedforward control plays an important role. And position feedback plays an important role when the difference is small.

If the excavator is in artificial operation condition, it can get rid of the position closed loop and just use velocity feedforward to realize velocity control. In this way, the operator gives sets the velocity demand by operating the joystick, the signal is calculated and gives to the motor driver to control the rotational speed of the electric motor and so the pump output the demand flow. If the excavator works under automatic operation condition, when the target position is given, the target velocity signal will be formed by velocity regulator in position closed loop according to the position difference between the target and actual position.
As shown in Fig.7, the operator gives the desired position signal $x_{set}$, then the controller calculates the desired velocity signal $v_{set}$ and $v_x$ to control the speed of the servo motor. And then the velocity and position of the cylinder can be realized. If the velocity feedforward is cut off, then the system works as position control. And also, the controller can be modified to velocity open-loop control if the displacement feedback is cut off.

### 4.2 Experimental System

In order to provide compared data about working performance and energy efficiency, the test of arm cylinder with separate metering in and separate metering out system driven by an inverter motor is implemented first\(^\text{[27-28]}\). After the test of the arm cylinder controlled by separate metering system, the asymmetric pump controlled arm cylinder test rig is constructed, as shown in Fig.8.

As shown in Fig.8, the arm cylinder is controlled by an asymmetric pump driven by a servo motor. A small gear pump is introduced to compensate the unbalanced flow caused by leakage. And also, there are additional instruments such as displacement sensor in the actuator, pressure sensors on the pump and cylinder, power sensor and rotational speed sensor on the motor which are employed to detect the corresponding variables. The circuit control concepts of the position closed-loop combined with velocity feedforward are being realized by the hardware in the loop computer control system ds1103. The test rig and instruments are shown in Fig.9.

### 4.3 Experimental Results

1. **Velocity open-loop control**

   Fig.10 presents the experimental results of the outlet pressure curve of the pump, the pressure curves of the arm cylinder chambers, the velocity curve and displacement curve of the arm cylinder.
During 0-5.16 s, 9.59-13.44 s, 17.33-20.00 s, the operator does not operate the joystick and the motor does not work. During 5.16-9.59 s, the joystick output a positive control command, the hydraulic cylinder extends out. In this process, the pressure of the rod chamber reduces gradually. When the direction of the load force is changing, the pressure of the rodless cavity increases. During 13.44-17.33 s, the joystick output a negative control command, the hydraulic cylinder retracts. In the retracting process, the pressure of the rodless increases gradually. During the whole work time, there velocity and pressure have no significant fluctuations.

(2) Position closed-loop combined with velocity feedforward

Fig. 11 and Fig. 12 show the dynamic response with the closed-loop position control strategy combined with velocity feedforward. At the time of 2.41 s, the demand position is set to as 600 mm with the keyboard. The servo motor accelerates from zero to the saturation value and the arm cylinder extends out. The motor speed set point is determined by the velocity and position closed loop. During the extending process, the hydraulic cylinder moved moves from 50 mm to 599.68 mm for 5.84 s with an average speed of 94 mm/s. At 18.23 s, the given position is set to as 100 mm via the keyboard, the motor accelerates reversely, and the cylinder retracts and the cylinder moves from 599.85 mm to 100 mm for 4.69 s with an average speed of approximately 106.60 mm/s. The control error of the strategy is small, only about 0.10 mm, which can satisfy the requirements.

(3) Energy consumption characteristics

Fig.13 shows the electrical power input and consumption of electric energy curves comparison between the separate metering in and metering out system and the new designed circuit at an average run speed of 100 mm/s. As shown in the Fig.13, during 2.41 s-6.98 s, the cylinder extends, the average electric input power of the independent control system is about 2.83 kW, and which is about 0.74 kW of the asymmetrical pump system. The energy saving ratio reaches up to 87.0%. During 19.02 s-22.81 s, the cylinder retracts, the average electric input power of the separate metering in and metering out system is about 11.7 kW, and which is about 5.8 kW of the asymmetrical pump system. The energy saving ratio reaches up to 50.4%. In the whole course of the work, the arm displacement from 50 to 600 mm, the separate metering in and metering out system consumes electricity about 127.1 kJ. And asymmetric pump control system consumes only 30.95 kJ. The energy saving ratio can reach up to 75.3%. The whole machine is more efficient with the design of the asymmetric pump control system.
(1) Compared with the system controlled by symmetrical pump, with an asymmetric pump, the load value and direction changing has no influence on the cylinder velocity without any feedback.
(2) The position closed-loop control combined with velocity feedforward is adopted, the error of steady-state control is small, only about 0.10 mm. The precision of position control is high enough to meet the automatic mining requirements.
(3) Compared with the separate metering in and metering out system, during cylinder extending, the energy saving ratio can be increased up to 87.0%. During cylinder retraction, the energy saving ratio can be increased up to 50.4%. And in a whole working cycle, the saving ratio can be increased up to 75.3%.

ACKNOWLEDGEMENT
The author(s) disclosed receipt of the following financial support for the research, authorship, and/or publication of this article: This work was supported by the National Natural Science Foundation of China (Grant Nos. U1510206 and 51575374).

REFERENCES
[14] 佐藤寛. 油圧装置から制御、配管を取り去ったらどのような形になむか、油圧技術. 2001, 6: 36-42
SENSORLESS POSITION CONTROL OF DIRECT DRIVEN HYDRAULIC ACTUATORS

Tom Sourander, Matti Pietola, Tatiana Minav, Henri Hänninen

*Department of Mechanical Engineering, Aalto University School of Engineering Espoo, Finland

(E-mail: tom.sourander@aalto.fi)

Abstract. This study investigates sensorless position control of valveless pump-controlled hydraulic actuators for non-road mobile machinery (NRMM). The utilized hydraulic systems are direct driven hydraulics (DDH), a type of electrohydrostatic actuators (EHA), which uses an electric servomotor to drive hydraulic pumps of a single actuator. The advantages of DDH over traditional valve-controlled hydraulics are increased energy efficiency due to elimination of the valve losses and improved controllability. The servomotor driven pumps provides a possibility for sensorless position control of hydraulic cylinders without need for sensors. The sensorless position control was realized by simulating the interaction of DDH units and hydraulic cylinders of a testbed prototype hybrid mining loader. Measured data from a test work cycle was used to test the accuracy of the simulation. The results demonstrated that accuracy with maximum error of about 30 mm could be achieved with no load and with 1040 kg load.

Keywords: Sensorless position control, Virtual sensors, Direct driven hydraulics, Electrohydrostatic actuator

INTRODUCTION

In efforts to reduce operating costs and harmful emissions of hydraulic machinery, there has been increasing research to find solutions for replacing and improving conventional systems. In non-road mobile machinery (NRMM), hybrid electric powertrains have gained popularity due to the high efficiency of electric motors. In hydraulic systems, electric motors have generally been utilized to drive pumps for conventional valve-controlled systems that suffer from throttling losses.

To replace valve-controlled systems, electric servomotor driven pump-controlled systems have been increasingly researched. Pump-controlled systems that operate a single actuator are generally called electro-hydrostatic actuators (EHA). In this study, a variation of EHA, direct driven hydraulics (DDH) is utilized. The advantages of DDH over traditional valve-controlled systems is mainly increased energy efficiency from elimination of valve throttling and idling losses as well as reduced oil cooling requirements. Direct pump control by servomotor provides improved and efficient actuator control with no need for servovalves. The most typical problem with EHA-like systems is caused by the difference in the volume between variable piston and rod sides in single-rod cylinders and thus the required flow rate ratio. This has been commonly solved by using double-rod symmetrical cylinders, however, this solution requires more space [1]. Solutions for single-rod cylinders involve pilot operated check-valves that compensate for the flow difference or separate pumps for either side that ideally have the same displacement ratio as the cylinder [1, 2].

With a trend of energy efficient machinery becoming more automatized, implementation of advanced control technologies and increased utilization of various sensors is required. Several common types of internal and external cylinder position sensors exist with their advantages and disadvantages [3]. However, these sensors can be expensive and challenging to utilize in applications that require reliability in harsh environments. Therefore, measuring the cylinder position with indirect means has gained research interest. Sensor virtualization is becoming more researched and utilized in various industrial applications, for example, car window DC motor [4] and steel sheet rolling mill [5]. With DDH, it is possible to realize sensorless position control of hydraulic cylinders by utilizing only torque and rotational speed data of the servomotor while accounting errors caused by hydraulic leakage loss.

Indirect position estimation of DDH actuated cylinders has been previously researched in Aalto University. In [6] the position estimation of a DDH actuated cylinder was studied by measuring the pump leakage in the locked position and calculating a slip coefficient that accounted for all leakages. In [7] sensorless position control in an electro-hydraulic forklift was researched. The position estimation in these studies was based on measurements and temperature changes were not accounted and also constant pump efficiencies was assumed. The results in
both research showed error in the range of 1-3 % during lifting and lowering. Also in [6], accumulation of error was observed in repeated lifting-lowering cycles due to inaccuracy in pressure estimation.

As a part of project El-Zon, sensorless position control of DDH actuated cylinders is investigated based on simulation of DDH units installed into a testbed prototype mining loader [8]. The cylinder position calculation method is based on Matlab/Simulink simulation model of the DDH units [9]. In the study described in [9] the position calculation accuracy was tested by comparing the calculated cylinder position to the actual position within the simulation. The research of this paper utilizes measurement data of the real DDH units to test the accuracy of the sensorless position algorithm in test work cycles. The following section describes the test setup followed by results of test measurements and finally conclusion and discussion.

**DESCRIPTION OF TEST SETUP**

The test platform for the DDH units is a prototype mining loader that has been converted to diesel-electric serial hybrid. The front section consists of a boom and bucket. The boom is actuated by two parallel cylinders and the bucket by one cylinder. The bucket cylinder is attached to a linkage mechanism that keeps the bucket angle stationary during the lifting of the boom. The loader has two DDH units, which drive the boom and bucket cylinders. Figure 1 presents a simplified model of the boom and bucket mechanism of the mining loader. Table 1 presents dimensions and utilized parameters of the boom and bucket cylinders of the loader. Here, A-side to the piston side and B-side to the rod side of the cylinder. Cylinder frictions were based on previous measurements with smaller cylinders [10] and linearly scaled up to the loader cylinder dimensions [9].

![FIGURE 1. Simplified model section of the loader boom and bucket.][1]

**TABLE 1. Parameters of the boom and bucket cylinders of the loader.**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Boom cylinder</th>
<th>Bucket cylinder</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stroke (mm)</td>
<td>311.15</td>
<td>850</td>
</tr>
<tr>
<td>A-side area (mm²)</td>
<td>10261</td>
<td>18050</td>
</tr>
<tr>
<td>B-side area (mm²)</td>
<td>7096.9</td>
<td>11847</td>
</tr>
<tr>
<td>Area ratio</td>
<td>1.446</td>
<td>1.524</td>
</tr>
<tr>
<td>Static friction (N)</td>
<td>570</td>
<td>760</td>
</tr>
<tr>
<td>Coulomb friction (N)</td>
<td>170</td>
<td>220</td>
</tr>
<tr>
<td>Viscous friction (N)</td>
<td>41000</td>
<td>41000</td>
</tr>
</tbody>
</table>

The DDH units were dimensioned to provide similar movement speed performance as the original system and the displacement ratio of the A- and B-side pumps was chosen to be as close as possible to the cylinder area ratios. The DDH units consist of an electric motor, motor controller, internal gear pumps [11], belt reduction transmission to better adjust the displacement ratios, pressure relief valves, anti-cavitation valves, cylinder safety valves and oil reservoir. Figure 2 presents a hydraulic diagram and the components of the boom DDH unit. The bucket unit is identical with the exception of different size pumps and a single cylinder. Data acquisition consists of the pressure, the oil temperature and draw-wire cylinder distance sensors. The motor is powered by a 60 Ah, 96V lithium-titanate battery pack [12]. Table 2 presents relevant parameters of the pumps.

Control network for the motor controllers and sensors is implemented utilizing CAN. The control system for the DDH units consist of a main computer unit MicroAutoBox II, which runs a Matlab Simulink-based control

---

[1]: #FIGURE 1. Simplified model section of the loader boom and bucket. [9]

---
software compiled to C++ - code with dSPACE developed CAN toolboxes. This program handles all CAN messages of the sensors, devices and the DDH motor controllers connected to the MicroAutoBox. Graphical monitoring and control of the program is implemented with dSPACE ControlDesk software. The Simulink program also includes data logging features for measurements. The motor controllers can be operated by a joystick or a predetermined test cycle input for more accurate and repeatable control. For testing the accuracy sensorless position control, a realistic test cycle is used, which consist of first lifting the bucket the lifting the boom, dumping and lifting the boom and finally lowering the boom. The cycles were controlled by PID position controllers with the input being the real cylinder position received from the draw-wire distance sensors and the output the required motor torque. When controlled by the joysticks, the motors are operated in speed control mode.

![Boom hydraulic circuit](image)

**FIGURE 2.** Hydraulic circuit and components of the boom DDH unit. [9]

Table 2. Parameters of the DDH pumps [11]. Notice that x2 in pump name means that the pumps are double chambered and the displacement in the name is for one chamber and X2 at the end means two parallel pump units, which will work as a single pump unit.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Boom pumps A-side PGI100-008x2 X2</th>
<th>Boom pumps B-side PGI100-013+011</th>
<th>Bucket pumps A-side PGI100-016x2 X2</th>
<th>Bucket pumps B-side PGI100-022x2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total displacement (cm³/rev)</td>
<td>15.8 * 2 = 31.6</td>
<td>24.2</td>
<td>31.6 * 2 = 63.2</td>
<td>44.4</td>
</tr>
<tr>
<td>Maximum circuit work pressure (bar)</td>
<td>270</td>
<td>70</td>
<td>70</td>
<td>172</td>
</tr>
<tr>
<td>Nominal volumetric efficiency at 250 bar and 1450 rpm</td>
<td>0.93</td>
<td>0.94</td>
<td>0.95</td>
<td>0.95</td>
</tr>
<tr>
<td>Nominal hydromechanical efficiency at 250 bar and 1450 rpm</td>
<td>0.91</td>
<td>0.91</td>
<td>0.92</td>
<td>0.93</td>
</tr>
<tr>
<td>Maximum rotational speed (rpm)</td>
<td>4200</td>
<td>4000</td>
<td>4000</td>
<td>3600</td>
</tr>
<tr>
<td>Gear ratio, motor to pump</td>
<td>28/41</td>
<td>41/47</td>
<td>28/44</td>
<td>44/47</td>
</tr>
<tr>
<td>A- and B-side geared flow ratio</td>
<td>1.416</td>
<td>1.542</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The sensorless position control is based on estimation of cylinder piston positions with a simulation model of the bucket arm hydraulic systems [9]. The model was created in Matlab/Simulink using ready-made Simscape fluid power blocks for hydraulics. In the DDH system, the pump speed affects the fluid flow rate and thus the cylinder speed and torque affects the pressure that causes driving force for the cylinder. Multiple sources for non-linearities and errors exist, which is caused by the nature of hydraulics. The most prominent is the volumetric efficiency of the pumps, which affects the leakage flow mainly in function of pressure and oil viscosity. Hydromechanical
efficiency of the pump is also important but less significant factor in the total efficiency and error in the produced flow [13]. Other significant affecting factors are the efficiency of the cylinders, bulk compressibility of the oil and pressure losses in the pipelines. These non-linearities make it challenging to calculate the positions of the cylinders. In addition, parameters of some components are based on estimations and simplified calculations as measuring them would be impractical. As the manufacturer of the pumps only provides efficiency curves for nominal operating conditions in which pump speed and oil viscosity are constant [11], it is not possible to directly form efficiency tables for various speed, pressure and temperature conditions. Measuring these was not possible, thus the simulation model calculates pump efficiencies based on Hagen-Poiseuille laminar pipe flow model using the nominal efficiencies as reference points [9]. Due to the pump leakages being mostly pressure dependent, more cylinder load requires higher pump pressure and thus torque, which leads to increased leak flow. This leads to the cylinder movement for a given fluid volume being ultimately dependent on the pump torque, speed and oil viscosity. Thus, it is possible to estimate the movement of a cylinder from the torque and speed of the electric motor. Figure 3 summarizes a diagram of the interaction between different parts and factors related in the sensorless position calculation process.

![Diagram of sensorless position calculation process]

The simulation of the interaction of the motor and the cylinders was performed with cylinder load increasing from a small load to pressure relief valve limit, oil temperature increasing from -5 °C to 100 °C and the motor run at three different speed ranges. The results were saved as lookup tables of cylinder movement in mm per motor revolution in function of motor torque at various temperatures and three motor speed ranges. Since the simulation was run at variable time step, the amount of motor revolutions per step varies. In each time step, the cylinder movement per motor revolution at the current torque is looked up from the tables. This is then compared to the current motor speed to calculate the cylinder movement during the step. The movement at each step is then cumulatively summed to calculate the current cylinder position. For more detailed description of the simulation process, refer to [9].

Figure 4 provides a flowchart explanation of the sensorless control idea of a DDH actuated cylinder. The user in this case is the high-level input source and can be either human or an automated process that provides control inputs to the control software that in turn produces input to the motor controller. The motor controller in turn feeds back torque and speed data from the motor sensors, which are used to calculate the estimated cylinder position. This position data is then fed back into the control software. This way the cylinder position calculation functions as a normal position sensor for the user. In this research, the draw wire position sensor is utilized for the reference positions. In further applications simple proximity sensors, for example Hall-effect, can act as reference points.
RESULTS

First, Figure 5 presents examples of simulation based boom and bucket cylinder movement look-up tables at four different oil temperatures and maximum motor speeds. The bucket motor torque is opposite to the boom due to the bucket cylinder rod-side being the lifting side and thus the motor is driven in negative direction. These figures illustrate that increasing cylinder load and pressure, and thus torque results in less cylinder movement per pump revolution during lifting due to increasing pump leakage and vice versa for lowering. Rapid drop at high torque is caused by the pressure reaching pressure relief valve limit. At higher temperatures, the oil viscosity is lower and thus causing more pump leakage and lower cylinder movement ratio. The sensorless position calculation is based on the lookup tables consisting of these graphs at various temperatures and speeds. The oil temperature during the measurements varied between 25 and 30 °C. The parameters of the oil are based on Shell Tellus T 32 [14], which is VG32 equivalent hydraulic oil.

![Graphs showing cylinder movement per motor revolution in function of torque at various temperatures](image)

**Figure 5.** Simulation results: Boom (left) and bucket (right) cylinder movements per motor revolution in function of torque at various temperatures [9].

The measurements were performed by running a lifting-lowering cycle of the boom and bucket at four different speeds and with no load weight at the bucket and with a 1040 kg payload. Results for no load and with load at the highest speed are presented. The maximum speed cycle lasts for about 25 seconds and the motors operate at about 5000 – 6000 rpm.

---

**FIGURE 4.** Flowchart of the sensorless control method of the DDH cylinders. [9]
Figure 6 left part shows the calculated boom cylinder position at no load and maximum cycle speed validated by measurements (marked as real position in the Figure) and the right part shows the error between the measured and calculated values. The calculated positions are as is and are only limited by soft upper and lower limits.

During the lifting movement, the error accumulation is minimal and is apparently mainly caused by the beginning acceleration and end deceleration. When the cylinder real position has reached the top position, the calculated position continues to creep upwards. This is can be explained as the boom motor maintains a speed of about 60 rpm (Figure 7), which does not affect the cylinder speed however it manages to deceive the position calculator. This is ultimately caused by low accuracy PID control to achieve smoother movement. Faster responding PID would result in better accuracy at the cost of sharp torque spikes and motor speed oscillation, which are detrimental to the position calculator.

Figure 8 left part presents the measured and calculated positions of the bucket cylinder during the same cycle and the right part shows the difference between these.
Figure 8. Bucket cylinder cycle with no load. Left: Bucket cylinder measured real and calculated. Right: Difference between the measured real and calculated boom cylinder position.

Figures 9 and 10 present results of the same cycle, however the position calculator now utilizes the end and middle points of the measured position as references. The boom cylinder has a stroke of 311.15 mm so the middle point is rounded to 155 mm. The bucket cylinder middle point is at 425 mm. These movement cycles were not allowed to reach the upper limits due to workspace limitations. In theory, utilizing the reference point compensation should result in less cumulated error over successive cycles.

Figure 9. Boom cylinder cycle with reference point compensation and no load. Left: Boom cylinder measured real and calculated positions. Right: Difference between the measured real and calculated boom cylinder position.
Figure 10. Bucket cylinder cycle with reference point compensation and no load. Left: Bucket cylinder measured real and calculated positions. Right: Difference between the measured real and calculated boom cylinder position.

Figures 11 and 12 present results for the same cycle performed with a 1040 kg payload attached to the bucket. Figures 13 and 14 add the reference point compensation. The upward creep of the boom cylinder calculated position is also present in these results.

Figure 11. Boom cylinder cycle with a bucket payload of 1040 kg. Left: Boom cylinder measured real and calculated positions. Right: Difference between the measured real and calculated boom cylinder position.

Figure 12. Bucket cylinder cycle with a bucket payload of 1040 kg. Left: Boom cylinder measured real and calculated positions. Right: Difference between the measured real and calculated boom cylinder position.

Figure 13. Boom cylinder cycle with reference point compensation and a bucket payload of 1040 kg. Left: Boom cylinder measured real and calculated positions. Right: Difference between the measured real and calculated boom cylinder position.
Figure 14. Bucket cylinder cycle with reference point compensation and a bucket payload of 1040 kg. Left: Boom cylinder measured real and calculated positions. Right: Difference between the measured real and calculated boom cylinder position.

Based on the results, the sensorless position calculator manages to achieve an accuracy with the maximum error being about 30 mm. The results demonstrated that the position accuracy is does not vary with payload variation. This accuracy is possibly enough for NRMM applications. The accuracy is slightly worse than in earlier simulation results [9], which could be expected since the hydraulic system in the simulation cannot exactly match the real system.

CONCLUSIONS

This study investigated sensorless position control of valveless pump-controlled hydraulic actuators in an NRMM test platform. The sensorless position control was realized by simulating the hydraulic units and forming look-up tables for the interaction between the cylinder movement and motor torque and speed at various oil temperatures. Measured data from a test work cycle was utilized to test the accuracy of the simulated results. The results demonstrated that accuracy with maximum error of about 30 mm could be achieved with no load and with 1040 kg payload.

The cylinder position calculation still has some limitations. The process is not capable of determining the true start position of the cylinder and will always regard the current starting position as the zero position. While software end limits are implemented, the algorithm is not directly capable of detecting if a cylinder piston has reached an end point. Theoretically, detecting the pressure limit from motor torque is possible however in practice, there can be many situations where the pressure reaches the relief valve limit but the cylinder has not reached an end point. Therefore, reference points at least at the cylinder ends are needed to synchronize the position calculator. Preferably a middle point reference would also be needed since it can be possible that a cylinder is never driven to the end stops. The results show that during a single lift-lower cycle, the reference point compensator does not significantly affect the error and in some cases is able to increase the error. However as mentioned in the previous section, the reference point compensation can help to avoid cumulative error during multiple movement cycles, which improves the average accuracy.

When considering that some simulation parameters of the hydraulic system were based on rough calculations and guesses, the achieved accuracy can be considered sufficient if an error of few centimeters is acceptable for an application. Higher accuracy could be achieved by more thoroughly measuring the relevant parameters and forming a more accurate cylinder movement tables in function of motor torque, speed and oil temperature.

ACKNOWLEDGMENTS

This research was enabled by the financial support of Tekes, the Finnish Funding Agency for Technology and Innovation, (project EL-Zon) and internal funding at the Department of Mechanical Engineering at Aalto University. The cooperation with Lehmuspelto Teemu and Tammisto Otto is highly appreciated.
REFERENCES


Copyright (C) JFPS All Rights Reserved.

- 2A10 -
HYDROSTATIC STEERING SYSTEM AND ENERGY SAVING EVALUATION IN IDLE REGIME

Giorgio Paolo MASSAROTTI*, Pietro MARANI*, Massimiliano RUGGERI*, Esteban CODINA**

National Research Council of Italy – Institute for Agricultural and Earthmoving Machines
Via canal bianco, 28, 44124 Ferrara Italy.
** UPC. Universitat Politècnica de Catalunya. BarcelonaTech
Lab.Mec.Fluids Planta Baixa 0.10 C. Colom, 11 08222 Terrassa Spain.
(E-mail gp.massarotti@imamoter.cnr.it)

Abstract. The main goal of modern steering systems is to ensure the feeling of control whilst keeping an high safety level since steering is one of the most critical function of a mobile machine. In manifold mobile machines the architecture include priority valves that on one hand increase safety, on the other increase the energy demand. In the present work a conceptual study of a novel steering system is outlined. With this architecture some reduction of fuel consumption can be obtained avoiding flow supply in idle condition. The system is separated from the rest of the hydraulic system thus the priority valve is not necessary. The aim of the system is to have the same characteristics of a classical system (with steering unit), with a better energy behavior in idle regime. Several simulations will be analyzed with the aim of studying the steering performance with emphasis on the steering wheel control characteristics in both normal and emergency operation.

Keywords: Hydrostatic Steering, Energy consumption, Negative Control Displacement.

INTRODUCTION

Over the years, with the aim of reducing the energy consumption of devices on machines, several attempts have been made. Many researchers and specialists in the industry have focused on various aspects of the vehicle's steering system. The point of view between a researcher and other specialist can change the understanding knowledge available to each one. In the work [1] a study of a rack steering system with hydraulic power suction was made. The system has also been studied in dynamic terms and, given its simplicity, can be widely applied in the automotive field. Dynamic information management was dealt with by avoiding the use of electronic systems as in the case of the system in [2]. This entails a greater rigidity of the system but certainly an uncontested reliability. The author examined the utility of a modular system with the aim of reducing energy consumption due to machine shift actions [3], and the system treated in this work remains in line with this philosophy. The work developed in [9] can be defined as an improvement study on the oil flow generation feed system. The author focused on the functional analysis of the pump in order to avoid energy consumption when not required. The [10] is focused on the development of an electro-hydraulic system, very trendy at this time, with the aim of obtaining optimal management in the various situations of circuit use. However, not only energy saving is important in the case of steering systems, an example comes from work [11] where the use of accumulators has been carried out with the aim of studying the system in different work environments. The functional scope of the steering systems must go hand in hand with the energy saving part. The steering device is a feature that is not only limited to have vehicles on the ground but also to the craft that need to correct the offshore sea lane and the mooring operations in the harbor. In the work [12] a rudder handling system efficiency survey, under normal operating conditions, using electro-hydraulic implementation (as for [10]) provided an overview of the system adjustment range by expressing also the effective torque values. The response of these systems to rapid action is of paramount importance for safety purposes, so a dynamic study was addressed in the work [13]. In this paper, only the correlation between the speed of the steering wheel and the displacement of the power pump piston will be considered. The system tends to reduce, in addition to energy losses, the problems arising from high pressure fluid in the steering unit inside the cabin. The steering unit study was done with the aim of reducing noise as well, as these units are located inside the cabin directly in contact with the user. The noise reduction in [14] was made using CFD simulations on the inner steering unit distributor.
STEERING SYSTEM IN CLOSED LOOP

The aim of the proposed system is to achieve a lower consumption energy in idle regime this is obtained by means of a closed loop hydrostatic unit. In Fig.1 an hydrostatic unit, coupled with the main actuator, provides the energy to turn the wheels. In this system a fixed displacement pump (gear pump for example) with a spool type distributor aimed at the displacement control of main pump is provided, instead of a traditional steering unit.

The aim of the system is to provide a steering wheel feeling (reaction mode) that is similar to the one of current mobile machinery. If the steering wheel is turned on the left a main valve will be displaced on the right and the balancing valve for the displacement control of a main control will connect the side with the tank. If the steering wheel (i) is not turned the pressure drops down because the actuation line is connected with the tank through a calibrated orifice.

It is possible to have an active or reactive steering type, changing the underlap spool of the distributor (g). This aspect will not be discussed in the following work.

Negative Displacement Control

Displacement control is actuated by means of a negative signal of the control steering pump (Fig.1), unbalancing the displacement control system of pressure pump (e) according to the pressure drop between the sides of displacement piston pump.

FIGURE 1. Steering system in closed loop.
The system is controlled by means of the negative pressure between the orifice k pull down the right pressure on b-c line. In this way the main pump displacement will be the function of the steering pump speed rate. In the system shown in Figure 1 there are orifices in the boost pump control lines. The system in Fig.2 must be designed in such a way as to have a performance match between the control pump (i) and the variable displacement pump characteristics regulation. This aspect would not be trivial as the regulation pump may have pressure drops such as not guaranteeing a regular torque to the steering wheel [6]. The resistance against the steering wheel must be such that it does not exceed the limit values described in the standards. This must be especially in the case of an emergency or in the case of no power supply from the power pumps.

**Functional description of the system**

The proposed system involves adjusting the displacement of the variable displacement pump by operating remotely from the steering pump, an open center distributor. The aim is to define a relationship so as to describe, depending on the steering wheel rotation, the displacement of the steering piston. The system considered is shown in Figure 5 and the mathematical description start from the next figure:

Turning the steering will allow the flow of oil that will act from right to left causing a decrease in the pressure identified with $p_r$. The left-hand zone is powered by the $p_n$ boost pressure and the relationship that describes the maintenance of the flow between pump and orifice is as follows:
\[ p_n - p_r = \rho \left( \frac{rD_m}{2C_dA_h} \frac{d\theta_m}{dt} \right)^2 \]  

(1)

The equation was obtained by an analogy made with [8] not considering the leakages and losses. A further drip in the system analysis is done by considering the balance of the open center spool for the variable displacement pump adjustment. By balancing the translation we will have:

\[ p_rA_p + k_p(x_t - x_c) - k_p(x_r - x_c) - p_nA_p = 0 \]  

(2)

From where we get:

\[ x_c = \frac{(p_n - p_r)A_p}{2k_p} \]  

(3)

Examining the example of the steering system in [7] we can resume the following equation for an open center valve (g):

\[ Q_L = \frac{Q_s}{U} x_c - \frac{Q_s}{2p_n} p_t \]  

(4)

In the case considered being \( Q_L = 0 \) and considering a stationary condition (where the rate of implementation has constant speed) we have:

\[ p_{lb} - p_{2b} = \frac{K_1}{K_2} x_c \]  

(5)

Where \( K_1 \) and \( K_2 \) are the characteristics of the valve. The same consideration as for the open-center valve can be made for the displacement piston of the variable displacement pump. The equilibrium will be made according to figure 4:

\[ x = 0 \Leftrightarrow p_t = p_r = p_n \]

**FIGURE 4.** Static balance for pump displacement variation.

The translation equation determines an equation similar to the (3):

\[ x_{cp} = \frac{(p_{lb} - p_{2b})A_{cp}}{2k_{cp}} \]  

(6)

Inserting into (6) the (5), (3), (1), we will have:

\[ x_{cp} = \frac{A_{cp} A_p K_1 \rho \left( rD_m \frac{d\theta_m}{dt} \right)^2}{4k_{pp}^2 k_p K_2} \]  

(7)

The equation (7) shows a quadratic relationship between the speed of steering piston pump displacement and the steering wheel speed (the latter proportional with the equation: \( D_m = k \cdot \phi = k_1 \cdot x_{cp} \) [7a]). However, this relationship is more marked at the beginning of the implementation and lower later. For simplicity, the system in
Figure 5 has been considered with the aim of understanding the mathematical correlation between the speed of the steering wheel and the position of the spool.

System Simulations for Evaluating Basic Parameters

According to the system shown in Figure 1, a simulation was performed with the Amesim 12.1.0 / Rev12 SL1 software to highlight the functional connection between the speed of the steering wheel and the piston displacement of the hydrostatic pump. The following figure shows the system being studied:

![System Simulation with Amesim software](image)

**FIGURE 5.** System Simulation with Amesim software.

The data for the various components of the simulation are as follows:

<table>
<thead>
<tr>
<th>Component characteristic</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steering pump cylinder capacity</td>
<td>3.2</td>
<td>cc</td>
</tr>
<tr>
<td>Orifice Diameter</td>
<td>0.7</td>
<td>mm</td>
</tr>
<tr>
<td>Diameter of the control distributor</td>
<td>10</td>
<td>mm</td>
</tr>
<tr>
<td>Underlap of the passing lights</td>
<td>0.12</td>
<td>mm</td>
</tr>
<tr>
<td>Stiffness springs of the distributor</td>
<td>9</td>
<td>N/mm</td>
</tr>
<tr>
<td>Boost pressure</td>
<td>5 – 25</td>
<td>bar</td>
</tr>
<tr>
<td>Diameter of pump piston</td>
<td>35</td>
<td>mm</td>
</tr>
<tr>
<td>Rotation speed of the steering wheel</td>
<td>120</td>
<td>g/min</td>
</tr>
</tbody>
</table>

The pump in position g represents the pump connected to the steering wheel, while the spool represents the piston for adjusting the displacement power pump. The distributor c is an open center spool with a negative overlap and the same for all ports (symmetrical overlap). The system works by maintaining constant pressure on one side of the steering pump while on the opposite side the pressure drop for setting the center open spool is achieved by reducing the pressure from the boost. Between the supply and the aspiration of the steering pump there is a reduction in pressure relative to the boost pressure such that the pressure control conditions for the variation of the power pump capacity are met. The steering system has been designed with reference to [6] so as to have a pressure drop between the supply and suction of the steering pump so that the torque requires the steering wheel to be standardized and does not exceed the threshold in the emergency condition where the power supply alliance is absent (or the boost signal from the boost pump is empty).
In Figures 6, the movement of the power pump piston can be observed depending on the speed of the steering wheel. The blue and pink curves are the displacement piston of the hydrostatic pump. There is an increase in displacement as a function of boost pressure. The displacement of the piston changes if there is a steering action and an change in boost pressure. This aspect is most important as it allows to management of power loss (decrease of pressure) in idle regime, but also to manage the feel of the steering wheel in the state of high speed and low speed of the vehicle. In fact, in this case, the steering response will be higher in the case of 25 bar (low vehicle speed) and will be lower in the case of 5 bar (high vehicle speed). At high speed the vehicle does not require high steering sensitivity (high gain). Instead, in the low or zero speed of the vehicle it is more useful to have a high gain of steering gear so that you do not have to make too many steering turns. The condition with a 5 bar pressure boost implies the saturation of the power pump setting as the minimum pressure reached to the second 5.5 is the least achievable (environment pressure).

CONCLUSION

This system, unlike the priority valve management systems, saves power especially under idle conditions. The idle regime can be identified as the time when the machine is off and does not work but the engine has come to a minimum speed. This situation is important because for a large slice of overall machine life the machine is on but does not work. If this happens (condition obtainable by interrogating the vehicle's CAN network) then strategies could be adopted for which the boost pressure is limited to a minimum threshold value that is useful only for the lubrication of the power pump. If fact in the idle condition the power pump is null displaced so ideally it is not burning power from the motor and the only active power is that derived from the boost pump for which it is possible to make energy-efficient strategies.

$$P_{steering} = P_{boost} + P_{power}$$  \hspace{1cm} (8)

In the equation (8), the two power inputs needed for the steering operation can be evaluated: the first factor provides the power needed from the steering to the wheels; It implies the speed of steering and static support due to the weight of the machine and to the various frictions present in the mechanical chain. In addition, there is a portion of power required for the functioning of the control system which is obtained by the appropriate boost system. It can be handled according to the machine's use (high speed and low speed) and can also be controlled in the idle state condition and maintained appropriately low. The proposed system tends to be an alternative to systems using the priority valve. The treatment tends to highlight the possibility of reducing the energy used during the idle phase of the machine. The actuation of the hydrostatic pump and the boost pump can be done by means of an electric motor, in order to release the steering gear from the diesel engine. This
possibility could lead to energy benefits in terms of system management. The development of this system is in the embryonic phase, and in addition to the will of a dynamic system discussion, a project proposal and a prototype will be proposed in the near future.

RESEARCH ON THE CONTACT PRESSURE CONTROL OF A DIE WEAR TESTER

Chao Yang*, Shigang Wang*, Li Liu*

*School of Mechanical Engineering, Shanghai Jiao Tong University
800 Dong Chuan Road, Shanghai, 200240, P.R.China
(E-mail: yangchaoycx@163.com)

Abstract. In order to study the effect of the contact pressure on die wear during the stamping process of advanced high strength steels, this article analyzed the electro-hydraulic proportional control system on the contact pressure of a die wear tester, established the transfer function of the proportional-valve-controlled asymmetric hydraulic cylinder system, introduced PID control and simulated the control system in Simulink. The simulation results showed the control system responded fast and the errors were satisfactory. Then this article simulated the “equal” synchronous control strategy and the “master-slave” synchronous control strategy of the contact pressure and the wear displacement. The results showed that under the “master-slave” control, the deviation between the actual pressure and the target pressure was less and the control effect was better.

Keywords: Die Wear Tester, Contact Pressure, Electro-hydraulic Proportional Control, Synchronous Control.

1. INTRODUCTION

The application of advanced high strength steels (AHSS) in automobile shell forming, which achieves the same strength with thinner plates, is an important measure for lightweighting of automobile, energy saving and emission reduction. But the increase of sheet strength causes serious die wear problems during the stamping process, which results in surface quality problems of the sheet [1]. Therefore, a die wear tester was developed to research the die wear situations during the AHSS stamping process by simulating stamping conditions, which would help to select appropriate kinds of die material and predict the die life.

Among those conditions of stamping, the changes of the contact pressure between the sheet and the die have an important influence on the wear and the fatigue of the die. The typical change curves of the contact pressure with the stamping stroke are shown in Figure 1, it can be seen that there is a transient state and a steady state in U-shaped part stamping process. In the transient state, the contact is not close enough, so the pressure is high and changes drastically, the peak value has a significant effect on the wear situation during the stamping process [2]; in the steady state, the die and the sheet attaches closely, so the pressure becomes lower and stable. Thus, how to simulate the dramatic changes of the contact pressure with the stamping stroke correctly is the sticking point to develop the die wear tester.

In view of the feature that contact pressure changed drastically with the stamping stroke, so in the tester, an electro-hydraulic proportional control system was adopted to control the pressure, which had high control precision, high power density, fast dynamic responses and suitable costs [3]. But the dead zone and the flow nonlinearity of the proportional valve have a great influence on the system control characteristics. Therefore, the choice of control methods and control strategies is the key to realize the contact pressure control and the synchronization of the contact pressure and the wear displacement.
2. THE COMPOSITION OF THE DIE WEAR TESTER AND THE WORKING PRINCIPLE

After detailed analysis of the various conditions during the stamping process, as shown in Figure 2, an electro-hydraulic system was designed as the composition of the wear tester. The hydraulic schematic diagram is shown in Figure 3.

The blank holder is to fix the plate on the worktable. When the plate is put on the worktable, there is a hydraulic cylinder above each side of the plate. The PLC outputs control signals to the directional valves, changes the oil circuit and makes the hydraulic cylinders moving downward. The plate is fixed. The oil pressure is measured back to the PLC by pressure sensors. When the pressure reaches the pre-set value, the valves return to the median position. The part of the right blank holder cylinder is separated from the left part and has slide rails underside. The piston rod end of the drawing cylinder on this side is connected with the right part. The PLC outputs analog voltage to the proportional directional valve. The size and the direction of this voltage control the speed and the direction of the drawing cylinder. The movement position is input to the PLC by a displacement sensor. After reaching the pre-set position, the PLC controls the cylinder to stop moving.

The movement of a hydraulic cylinder controls the contact pressure. The cylinder is installed vertically and there is a normal force sensor on the underside. The die grinding head is at the bottom. When wearing, the grinding head contacts with the plate. The PLC outputs analog voltage to the proportional directional valve to control the vertical motion of the cylinder, which changes the contact pressure between the grinding head and the plate. The normal force is returned to the PLC by the force sensor. The servo motors and ballscrews complete the horizontal movement of the grinding head relative to the plate along X-axis and Y-axis. The position control module of the PLC controls the horizontal movement.
After moving a pre-set distance with pre-set conditions, the grating scale is used to measure the wear height. It converts the height to a number of pulses, which will be counted by a counting card. The upper computer software runs on the industrial computer. The computer sends operation information to the PLC through the Ethernet, displays sensor information and motion status collected by the PLC in the software interface and converts the count value of the counting card to the actual wear height. By setting different contact pressures and wear speeds, researchers can study the effect of the pressures and the speeds on the wear situation. By changing grinding heads made of different materials, researchers can compare the wear resistance of different materials. By simulating the actual conditions during the stamping process, researchers can predict the die life and direct the production.

3. MODELLING OF THE ELECTRO-HYDRAULIC PROPORTIONAL CONTROL SYSTEM ON CONTACT PRESSURE AND CONTROL STRATEGIES

Considering that the contact pressure between the plate and the die has a great influence on the wear situation and the pressure changes drastically during the stamping process, it is necessary to model the oil circuit about the contact pressure and put forward reasonable control strategies.

3.1 Modelling of the Electro-hydraulic Proportional Control System on Contact Pressure

The oil circuit about the contact pressure can be seen as a proportional-valve-controlled asymmetric hydraulic cylinder system. The external voltage input determines the displacement of the valve core, which determines the displacement of the cylinder and the contact pressure. In the system modelling, when the hydraulic natural frequency of the hydraulic power mechanism is low, the transfer function can be considered as a first-order object. When the hydraulic natural frequency is high, the transfer function can be seen as a second-order object. In the article, the proportional valve was expressed as a first-order object. The transfer function between the valve core displacement ($X$) and the voltage input ($U$) is:

$$
\frac{X(s)}{U(s)} = \frac{1}{K_x T_v s + 1}
$$

Here $K_x$ is the proportional feedback gain of the valve core displacement, $T_v$ is the equivalent time constant of the valve.

Then the valve-controlled asymmetric hydraulic cylinder system shown in Figure 4 was analyzed.

![FIGURE 4. The schematic diagram of the valve-controlled asymmetric hydraulic cylinder system](image)

The hypotheses were assumed before analyzing the system [4]:

1. The valve was an ideal zero-opening four-port slide valve and four throttle windows were matching and symmetrical.
2. The flow at these throttle windows was turbulent.
3. The supply pressure ($P_0$) was constant and the return pressure ($P_0$) was zero.
4. The valve had ideal response ability, which meant that the flow change caused by the valve core displacement and the change of valve pressure drop completed instantaneously.
Based on these hypotheses, the article used the valve flow equation to establish the relationship between the valve flow ($Q_L$), the valve core displacement ($X$) and the load pressure ($P_L$). It is as follows:

$$Q_L = K_qX - K_cP_L$$

(2)

where $K_q$ is the flow gain and $K_c$ is the flow-pressure coefficient.

Then established the relationship between the proportional valve and the hydraulic cylinder by the flow equation of the hydraulic cylinder:

$$Q_L = C_tP_L + \frac{V_eP_L}{4\beta_e} + A_{me}\dot{y}$$

(3)

Here $\dot{y}$ represents the piston velocity, $C_t$ is the total leakage coefficient of the cylinder, $V_e$ denotes the equivalent volume of the cylinder, $\beta_e$ is the effective volume elastic modulus and $A_{me}$ represents the average piston area. $V_e$ was computed by equation (4) and $A_{me}$ was computed by equation (5).

$$V_e = \frac{(1 + \eta^2)A_1L}{1 + \eta^2} = A_eL$$

(4)

$$A_{me} = \frac{A_1 + A_2}{2}$$

(5)

$A_1$ represents the area of the piston no-rod side and $A_2$ represents the area of the piston rod side. $\eta$ is the flow ratio of the left and the right cavities, its value is $\frac{A_2}{A_1}$. $L$ stands for the maximum stroke of the cylinder and $A_e$ is the equivalent working area of the cylinder.

In order to find the relationship between the piston rod displacement and the valve core displacement, the load pressure equation of the cylinder[5] (the relationship between the load pressure, the piston rod displacement ($y$), the disc spring elastic force ($F_d$) and the external disturbing force ($f$)) was introduced:

$$P_L = \frac{(M\ddot{y} + F_d + f - f_{ad})}{A_e}$$

(6)

Here $M$ represents the total mass of the piston and the load, $f_{ad}$ is an intermediate quantity produced in the calculation, it could be expressed as follows:

$$f_{ad} = \begin{cases} 
\frac{A_1\eta^2(1 - \eta)P_e}{1 + \eta^2}, & \dot{y} \geq 0 \\
\frac{A_1(1 - \eta)P_e}{1 + \eta^2}, & \dot{y} < 0 
\end{cases}$$

(2)

The end of the piston is equipped with a normal force sensor, several series disc springs and a grinding head. The buffer springs can make the displacement greater under the same contact pressure, which helps to adjust the pressure. Considering that the mass, the velocity and the acceleration of the grinding head is relatively small, the article thinks the contact force ($F$) equals to the disc spring elastic force:

$$F = F_d = K_{y_F}y_r$$

(8)

Where $K_{y_F}$ is the elastic coefficient of the disc springs, $y_r$ is the piston rod displacement relative to the plate after that the springs begins to be compressed, which is determined by the piston rod displacement and the plate fluctuation ($y_s$):

$$y_r = y - y_s$$

(9)

By uniting these equations, the transfer function of the proportional-valve-controlled asymmetric hydraulic cylinder system was obtained as follows:
\[
Y(s) = \frac{K_q}{A_{me} K_x (T_v s + 1)} \frac{1}{U(s)} + \left( \frac{K_c + C_c + \frac{V c}{4 \beta e}}{A_{me} A_e} \right) \left[ K_{yf} Y(s) - f(s) + f_{ad}(s) \right] \\
\] 

\[
Y(s) = \frac{K_q}{A_{me} K_x (T_v s + 1)} \frac{1}{U(s)} + \frac{K_{ce}}{A_{me} A_e} \left( \frac{V c}{4 \beta e} s + 1 \right) \left[ K_{yf} Y(s) - f(s) + f_{ad}(s) \right] \\
+ \frac{V e M}{4 A_{me} A_e \beta e} s^3 + \frac{K_{ce} M}{A_{me} A_e} s^2 + \left( \frac{V e K_{yf}}{4 A_{me} A_e \beta e} + 1 \right) s + \frac{K_{ce} K_{yf}}{A_{me} A_e}
\] (10)

Where \( K_{ce} = K_c + C_c \).
According to equation (10), the block diagram of the transfer function is shown in Figure 5.

3.2 The Contact Pressure Control and the Synchronous Control

3.2.1 The PID Control of the Contact Pressure Execution System

Considering that PID control was widely used in the engineering, PID control was adopted to control the contact pressure. The control block diagram is shown in Figure 6.

\[ F_d \] represents the target normal force.
In order to simulate and verify the control system, the values of these parameters were assigned as table 1.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Meanings</th>
<th>Values</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>( K_x )</td>
<td>Proportional feedback gain of valve core displacement</td>
<td>1.2</td>
<td>V/m</td>
</tr>
<tr>
<td>( T_v )</td>
<td>Equivalent time constant of the valve</td>
<td>0.012</td>
<td>s</td>
</tr>
<tr>
<td>( K_q )</td>
<td>Flow gain of the valve</td>
<td>0.0171</td>
<td>(m³/s)/m</td>
</tr>
<tr>
<td>( A_{me} )</td>
<td>Average piston area</td>
<td>0.1231</td>
<td>m²</td>
</tr>
<tr>
<td>( K_{ce} )</td>
<td>Total flow-pressure coefficient</td>
<td>( 5.86 \times 10^{-8} )</td>
<td>m³/(s · Pa)</td>
</tr>
<tr>
<td>( A_e )</td>
<td>Equivalent working area of the cylinder</td>
<td>0.1341</td>
<td>m²</td>
</tr>
<tr>
<td>( V_e )</td>
<td>Equivalent volume of the cylinder</td>
<td>0.0262</td>
<td>m³</td>
</tr>
<tr>
<td>( \beta_e )</td>
<td>Effective volume elastic modulus</td>
<td>( 6.5 \times 10^8 )</td>
<td>Pa</td>
</tr>
<tr>
<td>( K_{yf} )</td>
<td>Elastic coefficient of the disc springs</td>
<td>( 6.47 \times 10^4 )</td>
<td>N/m</td>
</tr>
</tbody>
</table>
CONTINUED TABLE 1

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Meanings</th>
<th>Values</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>M</td>
<td>Total mass of the piston and the load</td>
<td>20</td>
<td>kg</td>
</tr>
<tr>
<td>A₁</td>
<td>Area of the piston no-rod side</td>
<td>0.1407</td>
<td>m²</td>
</tr>
<tr>
<td>A₂</td>
<td>Area of the piston rod side</td>
<td>0.1055</td>
<td>m²</td>
</tr>
<tr>
<td>η</td>
<td>Flow ratio of the left and the right cavities</td>
<td>0.75</td>
<td>1</td>
</tr>
<tr>
<td>Pₛ</td>
<td>Supply pressure</td>
<td>1</td>
<td>MPa</td>
</tr>
</tbody>
</table>

The PID control system of the contact pressure in Simulink is shown in Figure 7.

![PID control system diagram](image)

**FIGURE 7.** The simulation model of the contact pressure PID control system

The target normal force was set to a step signal with a steady value of 10000N or a sinusoidal signal with an amplitude of 5000N, the external disturbing force \( f \) was a white-noise signal and the plate fluctuation \( (y) \) was a sinusoidal signal.

The simulation results are shown in Figure 8 (the dotted line is the target force; the solid line is the actual force).

![Simulation results](image)

(a) Step signal response curve  
(b) Sinusoidal signal response curve

**FIGURE 8.** The simulation results of the contact pressure PID control system

It can be seen from Figure 8 that the step response has a certain overshoot, which means the system responds fast, and stabilizes within a certain range around the target pressure. The sinusoidal signal response tells that the follow effect of the system is great.

3.2.2 The Synchronous Control Strategies of the Contact Pressure and the Wear Displacement

The “equal” control and the “master-slave” control are two kinds of basic synchronous control strategies [6]. Under the “equal” control, every execution part tracks its own pre-set aim value to achieve the synchronization. Under the “master-slave” control, the output of a “master” execution part is considered as an ideal output, other parts track this output to...
achieve the synchronization. These two synchronous control strategies of the contact pressure and the wear displacement are shown in Figure 9.

![Diagram](image)

**FIGURE 9.** The schematic diagrams of these two synchronous control strategies

Under the “equal” control, this article set the target normal force and the target displacement corresponding to the same time based on the known relationship between the contact pressure and the stamping stroke. The force and the displacement were controlled separately. Under the “master-slave” control, because of the pressure fluctuation during the stamping process, the same force may correspond to different displacements. So this article took the displacement control as the ideal output and computed the pre-set force based on the actual displacement, which directed the force control later.

In order to compare the control effect of these two control strategies, simulations were carried out in Simulink. Among these simulations, a AC servo motor position control system was modelled to represent the displacement control system referring to the modelling process in the article [7]. The ball screw pitch is 10mm. The model is shown in Figure 10, $X_d$ represents the target position and $X$ represents the actual position.

![Diagram](image)

**FIGURE 10.** The position control system

The horizontal velocity was set as 5mm/s. The relationship between the force and the displacement is shown in Figure 11.

![Graph](image)

**FIGURE 11.** The change curve of the normal force with the displacement

The “equal” control simulation model is shown in Figure 12. MATLAB Function is the relationship in Figure 11.
FIGURE 12. The “equal” control simulation model

The simulation results can be seen in Figure 13.

(a) The displacement error under the “equal” control
(b) The comparison between the target force and the actual force under the “equal” control
(c) The force error under the “equal” control

FIGURE 13. The “equal” control simulation results

The “master-slave” control simulation model is shown in Figure 14.

FIGURE 14. The “master-slave” control simulation model
The simulation results are as follows:

![Graph (a)](image1.png) The displacement error under the “master-slave” control

![Graph (b)](image2.png) The comparison between the target force and the actual force under the “master-slave” control

![Graph (c)](image3.png) The force error under the “master-slave” control

FIGURE 13. The “master-slave” control simulation results

It can be seen from Fig.13 (a) and Fig.15 (a) that the displacement control results under these two control strategies are almost identical. The maximum dynamic error between the actual displacement and the target displacement is 0.1563mm.

Fig.13 (b) and Fig.15 (b) displays that these two control strategies did not adjust in time at the beginning. But after x=5mm, the adjustment improved greatly.

Fig.13 (c) and Fig.15 (c) shows that the overshoot under the “master-slave” control is lower and after the overshoot, the errors under the “master-slave” control are obviously less than those errors under the “equal” control.

Therefore, the “master-slave” control strategy was chosen to the synchronous control of the contact pressure and the wear displacement.

4. CONCLUSIONS

The die wear tester designed in this article simulates the important conditions of the stamping process, which achieve the blank holder, the plate drawing and the contact pressure control by hydraulic valves and cylinders and achieve horizontal movement by servo motors and ballscrews. Considering that the contact pressure during the stamping process changed drastically and had an important influence on the die wear, the article focused on the modelling and the computer simulation of the contact pressure electro-hydraulic proportional PID control system. The simulation results show that the response is fast and the errors are satisfactory.

In view of the fact that the correspondence relationship between the contact pressure and the wear displacement could influence the wear rate, this article studied the synchronous control strategies of the pressure and the displacement. The comparison of the simulation results under the “equal” control and the “master-slave” control show that under the “master-slave” control which sees the displacement control result as the ideal output, the errors between the actual normal force and the pre-set normal force are smaller. Therefore, the “master-slave” control is more suitable for the synchronous control.

5. ACKNOWLEDGMENTS

I would like to express my gratitude to all those who helped me during the writing of this article.
Firstly, special thanks should go to Professor Wang and Professor Liu, my teachers, who gave me important advice and helped me find my research direction. Without their help, this article could not be so organized and detailed. Secondly, I am deeply indebted to my colleagues in my laboratory for their help and encouragement. Without them, I might spend much time in lots of useless work and could not carry on my research.

6. REFERENCES

Oral Presentation | Oil hydraulics

[2A13-17] H8 (Control & Measurements 2)
Chair: Yasunori Wakasawa (National Institute of Technology, Toyota College), Kazushi Sanada (Yokohama National University)
Thu. Oct 26, 2017 3:30 PM - 4:50 PM  Room A (ACROS Fukuoka)

[2A13] NEW HIGH SENSITIVITY MEMS SENSOR FOR INDIRECT PRESSURE MEASUREMENT
*Massimiliano Ruggeri, Giorgio Massarotti, Esteban CODINA (1. CNR-IMAMOTER, 2. Universitat Politècnica de Catalunya)
3:30 PM - 3:46 PM

[2A14] DYNAMIC CHARACTERISTICS OF THE PRESSURE-DRIVEN DEVICE BY CONSIDERING THE PRESSURE FLUCTUATIONS INDUCED BY THE PROCESS OF DROPLET FORMATION
*Wen Zeng, Hai Fu, Shuai Yuan, Songjing Li (1. Harbin Institute of Technology)
3:46 PM - 4:02 PM

[2A15] A STUDY ON INTUITIVE CONFIGURATION OF JOYSTICK FOR OPERATOR IN FLATTENING TASK OF EXCAVATOR
*Quang Hoan Le, Soon Yong Yang (1. University of Ulsan)
4:02 PM - 4:18 PM

[2A16] ONLINE PARAMETER ESTIMATION OF HYDRAULIC SYSTEM BASED ON UNSCENTED KALMAN FILTER
*Takashi Yamada, Yoshiharu Nishida, Akira Tsutsui (1. Kobe Steel, Ltd.)
4:18 PM - 4:34 PM

[2A17] ON THE NONDIMENSIONALIZATION OF NOMINAL HYDRAULIC CYLINDER DYNAMICS
*Satoru Sakai (1. Shinshu University)
4:34 PM - 4:50 PM
NEW HIGH SENSITIVITY MEMS SENSOR FOR INDIRECT PRESSURE MEASUREMENT

Massimiliano RUGGERI*, Giorgio MASSAROTTI*, Esteban CODINA **

* CNR-IMAMOTER, National Research Council of Italy, Institute for Agricultural and Earthmoving Machines, Via Canal Bianco 28, 44124 Ferrara, Italy
(E-mail: m.ruggeri@imamoter.cnr.it)

Abstract. Sensorization of modern electro-hydraulic systems is one of the key technologies for system observability and controllability. Increasing needs for closed loop controls, high precision, power control and energy monitoring, diagnosis and safety concerns, ask for both pressure and flow acquisition in industrial and mobile applications. Pressure sensors need specific coupling systems for mounting, and both pipes and components must be modified to install pressure sensors. Traditional pressure sensors are related to mini-mess and to oil flow modification in the sensor area. Direct pressure measurement is often made using thin film sensors whose measurement principle is related to a strain measurement. Modern Silicon based technologies offer new solutions for a less invasive pressure measurement. Micro Electro-Mechanical Systems (MEMS) Technology is suitable to design new sensors for indirect pressure measurement. This paper present a new MEMS resonant sensor, for low strain measurement that can be successfully used to indirectly measure oil pressure acquiring component’s strain measurement.

Keywords: Pressure Measurement, MEMS, DETF Strain Sensor, Gear Pump.

INTRODUCTION

The strain measurement technology in mechanical structures was developed through traditional strain gauges, whose optimal measurement range is around 1 mε/ε; traditional strain gauges signals are very complex to be measured near the zero-strain condition, due to very low signal/noise ratio, that obliged the electronic designers to realize complex analog stages in the electronic signal conditioning part of circuits (as shown in the example in FIGURE 1).

![Full-bridge strain gauge circuit](image)

**FIGURE 1.** Typical Full Bridge configuration for Strain Gauges polarization and signal acquisition and one application example in a cantilever.

Based on sensitivity and measurement range, that are typically respectively 1 µε/ε and 1 mε/ε, traditional strain gauges can be successfully applied in pipes and, in general, in components with thin mechanical structure, where strain is easily acquired.

The purpose of the research described in this paper is to use strain technology for pressure measurement in hydraulic components.
As it is known, thin film pressure sensors use the strain technology applied to a thin membrane inside a chamber connected with the oil flow in pipes and tubes. The same technology could then be applied to other mechanical parts, if these parts offer a similar level of strain.

Unfortunately, due to need for reliability, robustness and resistance to stress and for durability, hydraulic components offer Thick-walled mechanical structures that prevent the measurement of pressure through the component’s body strain. The Strain level is too low to be measured through traditional strain gauges with sufficient precision and sensitivity.

As a consequence, strain gauges can’t be used in pumps, valves, and, in general, components where the body offer higher stiffness in respect to strain gauge sensitivity.

The paper aims to demonstrate the benefits of emerging MEMS sensing technologies for strain and indirect pressure acquisition.

**MEMS-DETF TECHNOLOGY**

In the last two decades MEMS sensors were developed in many topologies and the MEMS technologies are now complex and very precise, offering previously unseen solutions for mechatronic systems.

A MEMS DETF (Double Ended Tuning Fork) sensor is a silicon made diapason inside a chip and the technology is explained in [1] to [4] reference.

The DETF is a device consisting of two straight and parallel segments anchored at their ends, to the substrate. On the sides of these segments are placed two fixed electrodes to which an electrostatic force to attract laterally the two parallel segments is applied. Applying a sinusoidal voltage to the fixed electrodes, a swing of the two parallel segments is induced, so that the oscillation of the mobile segments coincides with the resonance frequency imposed by the geometric characteristics of the device (F).

The resonance frequency of the MEMS sensor changes according to the stresses applied along the axial direction. Conceptually the working principle is equivalent to that of an elastic or a rope of guitar: increasing (or
decreasing) the mechanical effort along the axis, the mechanical resonance frequency increases (or decreases) accordingly (FIGURE 4).

FIGURE 4. DETF MEMS Resonance Frequency Shift Working Principle under material strain. Compression (red arrow) and extension (green arrow) are directional in frequency shift.

The acquisition of the resonance frequency allows to measure the strain of the component and of the mechanical structure on which the sensor is mounted in a uniform manner.

The MEMS DETF used in the paper was conceived, designed and produced by the IMM - Institute for Microelectronics and Microsystems - of the National research Council of Italy.

MEMS-DETF Characteristics and Performance

The signal to be acquired is the oscillation frequency of the sensor that, based on the technology and component tuning, can be centered on 250 kHz to 380 kHz without strain at ambient temperature 20° C depending on the model and tuning of the MEMS sensor.

Due to its high resonance oscillation frequency, the sensor strain condition is revealed using an input capture function of a microcontroller, whose is able to calculate the time of a oscillation wave This digital electronic function is a device able to capture level change of electric signal that is compared with tunable thresholds and that is correlated with values of a high speed and high resolution counter, normally running at a frequency higher than the microcontroller frequency. Practically speaking it means that the waveform of the sensor vibration frequency is acquired at a frequency higher than 1.000 times the resonance frequency, enabling a very high resolution strain acquisition in a very short acquisition time. Thus virtually a single wave could lead to the strain value in a time equal to 1/Fosc = 2,7×10^-6 sec, where Fosc = 350 kHz and it is the resonance frequency of the sensor.

In the real Due to the noise and to the sensor it is better to use a series of continuous waves, in order to acquire a mean value in a short period of time.

FIGURE 5. MEMS Sensor Linearity (left) and Sensitivity and precision (right) that is a function of acquisition time

The precision of the strain measurement is directly proportional to the number of samples acquired in stationary conditions, and the right number of waves shall be defined by the application.
In FIGURE 5 right, the relationship between sensor resolution and acquisition time is presented, where it can be noted that the sensor, using a 40 MHz microcontroller can reach a resolution of 150 pε/

It can be proved that because of sensor characteristics it is suitable to be used in hydraulic components.

**PRESSURE MEASUREMENT IN A GEAR PUMP**

In order to demonstrate the potential of the MEMS DETF technology, a smart component was prototyped. The idea is to measure the pressure of the hydraulic fluid inside a component through and indirect measurement. In that case the measurement is based on the strain of the component induced by the internal pressure.

A gear pump was used, because is one of the most used components in hydraulic plants and because in most traditional hydraulic circuits traditional pressure sensors are used or are considered too costly or there is no enough space to install a traditional sensor.

To evaluate the real strain of the pump under pressure a functional analysis was performed and a FEM analysis was done, following also the study found on [5].

The objective of the analysis was to have a reliable evaluation of the pump body strain and to find the position of the pump body where to apply the strain measurement, based on the strain value.

In FIGURE 6 (left) the result of the FEM analysis performed over a real gear pump model is presented with an inner pressure of 250 bar, considering the void cavity but applying the pressure gradient as in the real condition of the pump running. It can be noted that the body strain is not symmetric due to the pump main function, where the delivery port is on the right in the figure. It can also be noted that the top of the pump offers a too high strain for the sensor, that is higher than 1 mε, while moving from the top the strain is proportionally reduced.

In FIGURE 6 right it is shown that the sensor was mounted in the position chosen by the results of the FEM analysis, in order to apply the sensor in a region of the pump body where the calculated strain is optimal in respect to the sensor characteristics (no more than 300 µε/at 250 bar), in order to have a good sensitivity and sensor resolution, without risk to reach the maximum allowed strain for the sensor, where the measure can go in overflow.

![FIGURE 6. FEM of the Gear Pump Body and real Pump with Sensor Position Corresponding to the chosen area where strain is a good sensitivity sensor’s range](image)

Based on this study, the sensor was placed in an intermediate position between the top and the delivery port of the pump, in order to find a region with a maximum strain of 200 to 300 µε. Experimental results in static conditions were presented in [6] and confirm the FEM analysis.

Based on the FEM analysis the surface of the pump was prepared, in order to have a plain surface of 8 mm wide, to apply the sensor with a proper glue. The prototype sensor mounting is very complex both for the glue choice and for the bonding operation, that directly connect the diapason electrodes on the MEMS component.

This operation will be comparable with the one used in traditional strain gauges in the industrial version of the component.

The sensor was coupled with a flexible small electric board where the small filaments (bonds) that are directly connected to sensors are connected to normal BNC cables, to bring signals to the electronic control circuit at 1,5 m distance (FIGURE 6 right and Figure 3 right).

The sensor was bounded manually using a proper tool and was tested before and after mounting the pump ports. The small mass of the sensor and the bond was tested in similar application until 20 g acceleration.

The MEMS sensor shown in FIGURE 6 right (and FIGURE 3 right) is equipped with 8 DETF sensors at various angles in respect to the vertical direction but only one is connected and used.
THE TEST BENCH

For the tests two pumps were instrumented with the MEMS – DETF Sensor, a 3 cc and a 16 cc. In this paper the results obtained with the 16 cc gear pump are presented. This Pump was equipped also with a traditional strain gauge in equivalent position of the MEMS sensor, in order to demonstrate that in the same working conditions the traditional strain gauge doesn’t offer a sufficient resolution and sensitivity. The pump mounted in the bench and the particular of the sensors mounted in the pump body is presented in FIGURE 7.

FIGURE 7. Gear Pump mounted in the Test bench with sensors cables and on the right the particular of the MEMS sensor and traditional Strain Gauge sensor mounted in the pump body.

Finally the pump was connected to hydraulic pipes and to a hydraulic bench capable of generate constant and variable pressure, with oil flow and capable to generate constant speed of the pump shaft. The test bench schematic circuit is presented in FIGURE 8 left, while in the right it is shown the test bench.

FIGURE 8. Test bench schematic Circuit with the Device Under Test and the real test bench image

The test bench was used to maintain the pump at a defined rotational speed, performing the tests at different regulated pressures. Unfortunately the bench is not equipped with a system capable of maintain the oil at a constant temperature and this, as it will be shown in the test results, will introduce an uncontrolled variable in the strain measurement. Due to oil temperature changes, the linearity of the sensor, proved in [6] can be reproduced because of dynamic conditions.

TESTS DESCRIPTION AND TEST RESULTS

Preliminary tests were performed in order to understand the MEMS sensor and the connected electronic control system speed and resolution.
In the FIGURE 9 it is presented an image of the acquisition of the pump strain signal at low speed (200 rpm), where it is visible the strain ripple generated by the pump gears pressure ripple.

FIGURE 9. Strain (and Pressure) ripple of the pump’s gears at constant speed of around 200 rpm the X axis is the Time (s) and Y axis is the resonance Frequency of the sensor (Hz).

The number of samples can be changed and the strain ripple can be acquired at all the speed range of the pump. This Figure demonstrates the sensibility of the sensor and the opportunities that this technology can offer if applied on the components. It can be noted that the pressure ripple generated by the pump gears.

Also the dynamic characteristic of the sensor was preliminarily tested and in FIGURE 10 a transient from 5 to 20 bar is shown.

FIGURE 10. Transient from 5 to 20 bar at 600 rpm. the X axis is the Time (s) and Y axis is the resonance Frequency of the sensor (Hz).

The main test series was performed in order to characterize the strain sensor in respect to pressure, in order to evaluate the relationship between pump inner pressure and pump body strain. Test were carried out starting from different temperatures and applying constant speed and constant pressures to the pump, in order to measure and acquire the pump strain in stationary conditions, because of the objective of mapping the pump strain and the sensor characteristics.

This experiment offered very good results at low pressures, while at higher pressures the stationary pressure generated in the delivery port of the gear pump provoked the oil temperature increase during the stationary test, changing the temperature of the oil and of the pump in the area of the delivery port. The Oil Temperature and, as a consequence, the Pump Body Temperature, introduced a independent variable in the tests that made difficult to prove the pump body strain linearity as a function of pressure, but the sensor still demonstrated its high resolution.

Test were performed maintaining the pressure constant at delivery port of the pump and the revolution speed of the pump also constant, in order to acquire a big number of data from the sensor and to evaluate a mean value for the strain. This is need both for validate stationary conditions and for the characteristic of the sensor, whose output is an oscillating signal at the resonance frequency of the DETF structure.

In FIGURE 11 are shown the pump strain characterization at two different oil temperatures in test sessions with oil pressure rise – on the left -, and oil pressure drop – on the right.

The tests show a very good sensor sensibility and resolution in respect pressure in the entire pressure range of [0, 250] bar, while the linearity of the measure is affected by the strain addictive due temperature transient as a consequence of oil resistance due to the high pressure.
FIGURE 11. Stationary characterization of the Pump Strain at different pressures and oil temperatures at 1000 rpm.

It can be noted that the linearity is more affected by the temperature change effect in the left figure, where the test started from 39°C of oil temperature, while at the end of the test the measured oil temperature in the circuit at the delivery port was 51°C and in the second test started from 49°C of oil temperature, while at the end of the test the measured oil temperature in the circuit at the delivery port was 57°C.

During the tests carried out reducing the pressure by stationary steps of 10 to 15 seconds, the linearity of the strain as a function of the oil pressure is less affected by the change of oil temperature, due to the thermal inertia of the oil and of the metal of the pump. In fact the test started at 54°C and ended at 49°C with around half of the temperature transient in respect to the tests performed rising the pressure through stationary steps of 10 seconds each.

In TABLE 1 it is presented the result of the calculation performed by the data collected during the tests. The most remarkable data is that the sensor offers a resolution of more than 100 Hz/bar in all pressure condition. This lead to a virtual resolution of 0.01 bar through the strain acquisition, without need of pressure ports in the component or in the circuit.

TABLE 1. Resolution in Hz/bar for the strain sensor and linearity analysis in the pressure drop stationary tests at 39°C

<table>
<thead>
<tr>
<th>Pressure [bar]</th>
<th>20.4</th>
<th>60.4</th>
<th>120.2</th>
<th>180.2</th>
<th>250</th>
</tr>
</thead>
<tbody>
<tr>
<td>Resonance f [Hz]</td>
<td>360616.63</td>
<td>364838.15</td>
<td>372253.26</td>
<td>380108.94</td>
<td>391025.82</td>
</tr>
<tr>
<td>F(x)-f(x-1) [Hz]</td>
<td>-</td>
<td>4221.51</td>
<td>11636.63</td>
<td>19492.31</td>
<td>30409.18</td>
</tr>
<tr>
<td>Hz/bar resolution</td>
<td>-</td>
<td>105.54</td>
<td>116.60</td>
<td>121.98</td>
<td>132.44</td>
</tr>
</tbody>
</table>

Linearity is not a must, because using electronic control systems it possible to generate non-linear transfer function to manage sensor data, but the temperature of the pump body near the sensor area acquisition is a must, as it is demonstrated by the data, where strain and then pressure interpretation, is affected by the thermal strain of the pump body.

In order to understand the thermal strain another series of test were carried out, measuring the pump strain at different temperatures at the same pressure at pump port. The results are presented in TABLE 2 where it can be noted that at the same pressure at different Temperatures the pump body strain is different. And the sensitivity of the sensor for thermal strain is higher than the one for pressure strain (2000 Hz for 3°C, 686 Hz/°C).

TABLE 2. Thermal Strain effect at constant pressure and constant pump speed (1000 rpm).

<table>
<thead>
<tr>
<th>Resonant Frequency [Hz]</th>
<th>Pressure [bar]</th>
<th>Temperature [°C]</th>
</tr>
</thead>
<tbody>
<tr>
<td>356713.56</td>
<td>20.40</td>
<td>54.00</td>
</tr>
<tr>
<td>365072.57</td>
<td>119.50</td>
<td>54.00</td>
</tr>
<tr>
<td>381317.94</td>
<td>249.20</td>
<td>54.00</td>
</tr>
<tr>
<td>358775.30</td>
<td>20.70</td>
<td>51.00</td>
</tr>
<tr>
<td>367133.21</td>
<td>119.50</td>
<td>51.00</td>
</tr>
<tr>
<td>383339.70</td>
<td>248.00</td>
<td>51.00</td>
</tr>
</tbody>
</table>

The results are graphically presented in FIGURE 12. In the Figure it can be noted that the thermal strain represents an important part of the whole strain acquisition process and it can’t be neglected if the MEMS Strain sensor is used as a pressure sensor.
FIGURE 12. Thermal Pump Strain Characterization at different pressures and oil temperatures at 1000 rpm.

But the most important data to be noted is that the thermal strain seems to be constant at different pressures, then it can be mapped if the temperature is acquired contextually with the strain.

CONCLUSIONS

The paper presents the first experiences in a dynamic bench, performing the strain measurement through the new MEMS DETF sensor. The tests demonstrate that the sensor sensitivity and resolution are suitable to be applied to hydraulic components. The test also show that the thermal strain effect is very important and that this effect can’t be neglected if the sensor is used as a pressure sensor.

The good news is that temperature acquisition is not a problem and that in the next version of the sensor a floating resonator will be added, to measure the temperature in the component, and thus in the pump body in the sensor area.

The important result is that a new class of sensor was designed and realized, that could be disruptive for hydraulic components, allowing to measure, pressure, temperature, strain and also pressure ripple, from the external side of the components, without any need of pressure ports, then without altering the oil flow. This class of components could also be used to understand if the component strain is normal or excessive in the working condition, thus preventing breakages of components and being part of a predictive diagnosis system. Many activities are planned for the MEMS – DETF sensor and its application on different class of components will be presented soon to the international fluid power community.

ACKNOWLEDGMENTS

The research is carried out under the project "NANOMEMS-X, NANO-structured MEMS Sensors for Strain Applications", project code CUP B5216000120009, financed in the POR-FESR Regione Emilia Romagna 2016-2020 Program in Italy based on European Funding for Regional Development..

REFERENCES

2. FERRI M., BELSITO L., MANCARELLA N., RONCAGLIA A. et al Fabrication and testing of a high resolution extensometer based on resonant MEMS strain sensors, 16th International Conference Solid-State Sensors, Actuators and Microsystems 2011 (TRANSUDERS), DOI: 10.1109/TRANSUDERS.2011.5969171
DYNAMIC CHARACTERISTICS OF THE PRESSURE-DRIVEN DEVICE BY CONSIDERING THE PRESSURE FLUCTUATIONS INDUCED BY THE PROCESS OF DROPLET FORMATION

Wen Zeng, Hai Fu, Shuai Yuan and Songjing Li*

* Department of Fluid Control and Automation, Harbin Institute of Technology
Harbin, 150001, China
(E-mail: zengwen@hit.edu.cn)

Abstract. The pressure-driven device is designed to control the flow rates of the droplet microfluidic systems, which can eliminate the flow-rate fluctuations coming from the pump source. As monodisperse droplets are formed in the microchannel, periodic pressure fluctuations can be induced by the dynamic process of droplet formation, which can influence the stability and control precision of the pressure-driven flows. The effects of the pressure fluctuations induced by the droplet formation process on the dynamic characteristics of the open-loop and closed-loop control pressure-driven devices are comparatively studied. Particularly, a PI controller is integrated with the closed-loop system and by properly choosing the parameters of the PI controller, the amplitude of the pressure fluctuations of the pressure-driven device can be reduced drastically, which can increase the control accuracy of the driven pressure. Additionally, the effects of the container volume on the dynamic characteristics of the pressure-driven device are also discussed.

Keywords: pressure-driven device, droplet formation, closed-loop control, dynamic characteristics

INTRODUCTION

To precisely control the flow rates of the fluids is quite important for the stability and applications of the droplet microfluidic systems [1-10]. To date, the syringe pump is widely used to supply the flow rates of the fluids. However, for syringe-pump-driven microfluidic devices, there are flow-rate fluctuations coming from the pump source which are induced by the mechanical oscillations of the pump motor [11]. Consequently, because of the instability of the flow rates, periodic pressure fluctuations can be observed in the microchannels [12]. In order to eliminate the flow-rate fluctuations of the syringe pump, the pressure-driven device is designed to control the flow rates of the droplet microfluidic systems [13]. Both the stability and uniformity of droplet generation can be improved by the pressure-driven flows [14,15].

During the dynamic process of droplet formation, it has been observed that there are periodic pressure oscillations at the T-junction where monodisperse droplets are produced [16]. The frequency of the pressure fluctuations varies with the droplet production speed and the magnitude of the pressure fluctuations are associated with the geometrical parameters of the microchannel [17]. Additionally, for the pressure-driven microfluidic device, both the stability and control precision of the pressure-driven flows can be affected by the pressure fluctuations induced by the droplet generation process [18]. More importantly, due to the pinch-off process of droplets, periodic oscillations of the droplet velocity can also been observed along the microchannel [19]. Therefore, how to reduce the influences of the pressure fluctuations on the dynamic characteristics of the pressure-driven device is quite important for improving the monodispersity and uniformity of droplet production [20]. However, to the best of our knowledge, the stability of the pressure-driven device with the periodic pressure fluctuations has not been quantitatively studied during droplet production progress.

In this paper, both the open-loop and closed-loop control of pressure-driven devices are established. By varying the droplet production speeds and geometrical parameters of the microchannel, the pressure fluctuations of different magnitudes and frequencies can be obtained. Meanwhile, by considering the pressure fluctuations, the dynamic characteristics of the open-loop and closed-loop control of pressure-driven devices are comparatively studied. In particular, by integrating the PI controller with the closed-loop control pressure-driven device, the effects of the PI controller on the stability and control precision of the pressure-driven device are mainly discussed. Consequently, the fluctuation magnitude of the driven pressure can be decreased drastically by properly choosing the coefficients of the PI controller. By increasing the control accuracy of the flow rates, the stability of the droplet microfluidic systems will be improved simultaneously.
MATHEMATICAL MODEL

The working principle of the pressure-driven microdroplet generator is shown in Fig. 1. Here, the flow rates of the fluids is controlled by the pressure-driven device, which can improve the stability and accuracy of the flow-rate supply for the droplet microfluidic system. For the two immiscible fluids, the silicone oil is chosen as the continuous phase and the DI water is chosen as the dispersed phase. The viscosity of DI water is $\mu_d = 1$ cP and the viscosity of silicone oil is $\mu_c = 20$ cP. Additionally, the interfacial tension between the two phases is approximated as $\gamma = 40$ mN/m.

For the pressure-driven device, based on our formal research [21], the dynamic characteristics of the open-loop control pressure-driven device can be described by a first-order transfer function, as given by

$$G(s) = \frac{K_0}{\tau_0 s + 1}$$  \hspace{1cm} (1)

where $K_0$ is a gain coefficients of the transfer function, especially for the open-loop control of the pressure-driven device, it can be approximated $K_0 \approx 1.0$. Meanwhile, the time constant can be expressed as $\tau_0 = \frac{V_0}{k_1 p_0}$. Here, $V_0$ is the original volume of the container, $p_0$ is the original pressure of the container and $k_1$ is a gain coefficient.

In this paper, a PI controller is integrated with the closed-loop system to improve the dynamic characteristics of the pressure-driven device. The transfer function of the PI controller is given by

$$F(s) = k_p + k_i \frac{1}{s}$$  \hspace{1cm} (2)

where $k_p$ is the proportional gain coefficient and $k_i$ is the gain coefficient of the integrator for the PI controller.

In a T-junction microdroplet generator, during the droplet formation progress, there are periodic pressure fluctuations induced by the process of droplet formation. In particular, the pressure drop between the upstream and downstream of each droplet is a periodic function of time, and the amplitude of the periodic pressure fluctuations can be described by

$$\Delta p_{drop} = \gamma \left( \frac{2}{h} - \frac{1}{R_{\text{max}}} \right)$$  \hspace{1cm} (3)

where $R_{\text{max}} = \max(w_c, w_d)$. Here, $w_c$ is the channel width of the continuous phase, $w_d$ is the channel width of the disperse phase, $h$ is the channel height of the T-junction and $\gamma$ is the interfacial tension between the two immiscible fluids.

From the above equation, it can be observed the magnitude of the pressure fluctuations is mainly determined by the geometrical parameters of the T-junction and the interfacial tension between the two phases. More importantly, the frequency $f_p$ of the pressure fluctuations is consistent with the production rate $f_d$ of droplets. Therefore, by increasing the droplet production speed, the frequency of the pressure fluctuations can be verified.
RESULTS AND DISCUSSION

Dynamic characteristics of the Open-loop Control Pressure-driven Device

Based on the mathematical model of the pressure fluctuations induced by the process of droplet formation in the T-junction microchannel, it can be observed that the pressure at the T-junction is periodic function of time. Especially, for different geometries of the T-junctions, the time-varying pressure drop between the upstream and downstream of each droplet is associated with the droplet production speed, as shown in Fig. 2.

From the above results, it can be seen the frequency of the pressure fluctuations coincides with the droplet production speed. Especially, because of the droplet production progress, both the dynamic response speed and control accuracy of the open-loop control pressure-driven device will be affected by the pressure fluctuations in the T-junction microchannel. In this paper, the effects of the pressure fluctuations induced by the droplet formation process on the dynamic characteristics of the open-loop and closed-loop control pressure-driven devices are studied comparatively. Fig. 3 shows the schematics of the open-loop and closed-loop control pressure-driven devices. Especially, a PI controller is integrated with the closed-loop system to improve the stability and control precision of the pressure-driven device.

For the open-loop control pressure-driven device, the pressure fluctuations can be considered as a disturbance source of the device, and using the first-order transfer function of equation (1), the dynamic characteristics of the open-loop control pressure-driven device can be obtained for different volumes of the container and production rates of droplets, as shown in Fig. 4.
From Fig. 4, we note that for a given production rate of droplets, the driven pressure $p_i$ of the container is changing periodically with time as monodisperse droplets are formed in the microchannel, and the frequency of the pressure fluctuations is consistent with the droplet production speed. Especially, the fluctuation magnitude of the driven pressure $p_i$ is associated with the volume of the container. Particularly, by increasing the volume of the container, the magnitude of the pressure fluctuations of the pressure-driven device can be obviously reduced. In addition, to quantify the magnitude of the pressure fluctuations for the pressure-driven device, the fluctuation magnitude $\Delta p_i$ of the driven pressure is normalized by $\Delta p_{drop}$. For different volumes of the container, the normalized magnitude $\Delta p_i / \Delta p_{drop}$ of the pressure fluctuations is a function of the droplet production speed, as shown in Fig. 5.

**FIGURE 5.** The amplitude of the pressure fluctuations as a function of the production rates of droplets for the open-loop control pressure-driven device.
It can be observed that for the same volume of the container, the amplitude of the pressure fluctuations will be further decreased by increasing the production rate of droplets. Especially, for lower production rates of droplet, the amplitude of the pressure fluctuations is quite sensitive to a very small variation of the droplet production speed. As a result, by increasing the volume of the container and the droplet production speed, the control accuracy of the driven pressure can be increased simultaneously.

Dynamic Characteristics of the Closed-loop Control Pressure-driven Device

To reduce the fluctuation magnitude $\Delta p_j$ of the driven pressure and improve the stability and control precision of the pressure-driven device, the closed-loop control pressure-driven device is established. In particular, the PI controller is integrated with the closed-loop control system, and effects of the parameters of the PI controller on the control precision of the pressure-driven device are mainly studied. Fig. 6 shows the step-response characteristics of the open-loop and closed-loop control pressure-driven devices, while the droplet production speed is specified as $f_d = 1 \text{ s}^{-1}$.

By comparing the step-response characteristics between the open-loop and closed-loop control pressure-driven device, it can be observed that the fluctuation magnitude $\Delta p_j$ of the driven pressure will be drastically reduced for the closed-loop control system, and meanwhile the control precision of the driven pressure is increased significantly for the pressure-driven device.

For the closed-loop control pressure-driven device, the PI controller is used to improve the dynamic characteristics of the closed-loop control system. Here, by varying the parameters of the PI controller (the gain coefficient of the integrator is given $k_i = 1.0$), the normalized magnitude $\Delta p_j/Ap_{dop}$ of the pressure fluctuations can be calculated and meanwhile compared with the predicted value of the open-loop control pressure-driven device, as shown in Fig. 7.
FIGURE 7. Comparison between the amplitude of the pressure fluctuations induced by the process of droplet generation both for open-loop and closed-loop control of the pressure-driven device. For PI controller, the gain coefficient of integrator is already given $k_i = 1.0$, while the proportional gain coefficient spans from 0.1 to 1.0.

We note that by increasing the proportional gain coefficient $k_p$ of the PI controller, the amplitude of the pressure fluctuations of the pressure-driven device can be further reduced. As a result, by integrating the PI controller with the closed-loop control system, the influences of the pressure fluctuations induced by droplet generation process on the instability of the pressure-driven device can be greatly reduced, and high control precision of the driven pressure is obtained for the closed-loop control pressure-driven device. More importantly, by increasing the control accuracy of the driven pressure, precise control of the flow rates of the fluids can be achieved for the droplet microfluidic systems.

CONCLUSIONS

The flow rates of the droplet microfluidic systems are controlled by the pressure-driven device, which can eliminate the flow-rate fluctuations coming from the pump source. Meanwhile, during the dynamic process of droplet formation, periodic pressure fluctuations are observed in the microchannel, which greatly affect the stability and control precision of the pressure-driven flows. Compared with the open-loop control of pressure-driven device, the fluctuation magnitude $\Delta p_f$ of the driven pressure will be obviously reduced for the closed-loop control system. Particularly, by integrating a PI controller with the closed-loop control system and properly choosing the coefficients of the PI controller, the effects of the pressure fluctuations on the instability of the pressure-driven device are greatly reduced. As a result, the dynamic characteristics of the pressure-driven device will be improved and precise control of the flow rates of the droplet microfluidic systems can be achieved.

ACKNOWLEDGMENTS

We thank the National Natural Science Foundation of China (No: 51175101) and the Programme of Introducing Talents of Discipline to Universities (No: B07018) for financial support of this research.
REFERENCES


A STUDY ON INTUITIVE CONFIGURATION OF JOYSTICK FOR OPERATOR IN FLATTENING TASK OF EXCAVATOR

Quang Hoan LE*, Soon Yong YANG**

* Department of Mechanical Engineering, University of Ulsan
  93, Daehag-ro, Namgu, Ulsan, 680-749
  (E-mail: lequanghoan87@yahoo.com)

** Department of Mechanical Engineering, University of Ulsan
  93, Daehag-ro, Namgu, Ulsan, 680-749
  (E-mail: soonyy@ulsan.ac.kr)

Abstract. Hydraulic excavators are among the most versatile earthmoving equipment: these machines are used in civil engineering, hydraulic engineering, grading and landscaping, pipeline construction and mining. Especially, in the flattening task, the excavator operator actuates the machine controls (joysticks, pedals and switches) in an organized form to achieve the desired straight motion. The actuation of these controls is a complex and not intuitive task so it requires long and costly training periods. This paper presents an intuitive configuration of joystick for operator in flattening task of excavator. This method control lets each joystick perform independently horizontal and vertical motions of the end of bucket. The proposed method allows users to control the excavator intuitively and operate complex motions smoothly with no constraints. For the test of the proposed method, a virtual excavator scheme with a joystick control has been developed. Besides, the virtual reality interface is designed to allow the user be able to observe the motion of the attachment of this model.

Keywords: Control, Excavator, Operator, Flattening Task, Virtual Reality Interface, Inverse Kinematic

1. INTRODUCTION

Excavation is an importance work in mining, earth removal and general earthworks. An excavator generally consists four links: swing, boom, arm and bucket. To execute these operations, the excavator operator actuates the machine controls (joysticks, pedals and switches) in an organized form to achieve the desired machine motion the actuation of these controls is a complex and not intuitive task, and therefore it requires long and costly training periods [1-3]. Specially, flattening task with straight-line motion require effort and concentration to map the inverse kinematic between actuator and the bucket tip. This takes a long period to train an operator proficiently.

FIGURE 1 Excavator tasks: digging, loading, grading etc.

Thus, this paper presents an intuitive configuration for joystick to reduce the difficulty and the time consuming for worker while flattening ground. At first, the design of intuitive configuration of joystick is presented base on the vertical and horizontal motion of the bucket tip. Second, the excavator simulation is proposed to apply the new method of joystick control for analysis and verify the advantage when comparing with the conventional joystick control configuration. Besides, the Virtual Reality Interface is described to ensure the intuitive monitoring for the excavator operator.
2. KINEMATIC AND INVERSE KINEMATIC OF EXCAVATOR

2.1. Kinematic of excavator

The forward kinematic is used to describe the positions and orientations of the points on the excavator in the Cartesian coordinate for the given joint positions during the digging operation. The problem can be summarized as below:

For the given \( \theta = [\theta_1 \ \theta_2 \ \theta_3 \ \theta_4] \) find the coordinate \( P = [X \ Y \ Z]^T = [f_x(\theta) \ f_y(\theta) \ f_z(\theta)]^T \)

To determine the positions of the points on the excavator in the base Cartesian coordinate frame, the relations between the fixed coordinate system and other coordinate systems is necessary. Therefore, the transformation matrix relating two adjacent coordinate frames was studied by Koivo [4] as follows:

\[
A_{i,i} = \begin{bmatrix}
\cos \theta_i & -\cos \alpha_i \sin \theta_i & \sin \alpha_i \sin \theta_i & a_i \cos \theta_i \\
\sin \theta_i & \cos \alpha_i \cos \theta_i & -\sin \alpha_i \cos \theta_i & a_i \sin \theta_i \\
0 & \sin \alpha_i & \cos \alpha_i & d_i \\
0 & 0 & 0 & 1
\end{bmatrix}
\]

(1)

Where, \( \alpha_i \) is the twist angle of link \( I \), \( a_i \) is the length of link \( i \) and \( d_i \) is the offset distance in link \( i \), \( i = 1, 2, 3, 4 \).

By the given coordinates of the origin in each coordinate frame \( O_i \), the coordinates of points \( O_i \) in the base coordinate frame can be described as follows using the equation (1):

\[
P_0^{O_i} = A_{i-1}^i P_0^i
\]

(2)

Where \( i \) specifies point \( O_i \) in the \( i \)th coordinate frame. From equation (2) we can describe the origin of each coordinate frame \( O_i \) in the base coordinate frame as follows:

\[
P_0^{O_1} = A_{0}^{O_1} P_0^{O_1} = [a_1 c_1 \ a_2 s_1 \ 0 \ 1]^T
\]

(3)

\[
P_0^{O_2} = A_0^{O_2} P_0^{O_2} = \left[ a_2 c_2 + a_1 c_1 \ a_2 s_2 + a_1 s_1 \ 0 \ 1 \right]^T
\]

(4)

\[
P_0^{O_3} = A_0^{O_3} P_0^{O_3} = \left[ a_3 c_3 + a_2 c_2 + a_1 \ a_3 s_3 + a_2 s_2 \ a_3 c_3 + a_2 c_2 + a_1 \ a_3 s_3 + a_2 s_2 \ 0 \ 1 \right]^T
\]

(5)

\[
P_0^{O_4} = A_0^{O_4} P_0^{O_4} = \left[ a_4 c_4 + a_3 c_3 + a_2 c_2 + a_1 \ a_4 s_4 + a_3 s_3 + a_2 s_2 \ a_4 c_4 + a_3 c_3 + a_2 c_2 + a_1 \ a_4 s_4 + a_3 s_3 + a_2 s_2 \ a_4 c_4 + a_3 c_3 + a_2 c_2 + a_1 \ a_4 s_4 + a_3 s_3 + a_2 s_2 \ 0 \ 1 \right]^T
\]

(6)

Where, \( c_i = \cos \theta_i, \ s_i = \sin \theta_i, \ \theta_{ij} = \theta_i + \theta_j, \ \theta_{34} = \theta_3 + \theta_4 \)
2.2 Inverse kinematics

The inverse kinematic equations are used to determine the joint angles and the actuator lengths corresponding to a specific position and orientation of the bucket, given in the base coordinate system shown in figure 3.

It is assumed that the coordinate of point O4 $\equiv N$ are given in the coordinate system. According to Tafazoli [5], the inverse kinematic model of the excavator is given as follows:

$$\theta_1 = \theta_b$$

$$l_1 = \sqrt{(z_b - a_1)^2 + z_b^2}$$

$$\xi_1 = a \tan \left[ \frac{z_b}{z_b - a_1} \right]$$

$$l = \sqrt{1 + a_4^2 - 2a_4 \cos (\xi_1 - \alpha)}$$

$$\xi_2 = a \sin \left[ \frac{(z_b - a_4 \sin a_4)}{l} \right]$$

$$\theta_2 = \xi_2 + a \cos \left[ \frac{2}{a_2} + l^2 - a_3^2}{l} \right]$$
\begin{equation}
\theta_3 = -\pi + a \cos \left[ \left( \frac{a_2^2 + a_3^2 - l^2}{2a_2a_3} \right) \right]
\end{equation}

(13)

\begin{equation}
\theta_4 = \alpha - \theta_{23}
\end{equation}

(14)

3. INTUITIVE CONFIGURATION OF JOYSTICK FOR EXCAVATOR

The model of the new joystick control method for operator in flattening task is described in Fig. 4. The end of bucket is moved independently with horizontal motion and vertical motion. In conventional joystick control method, it is required a complex combination between the boom, arm and bucket to move these two directions. In this model, user only operates one axis of each joystick to move the bucket forward-backward or upward-downward. Therefore, the new method takes less time to complete a task than the conventional method.

Specially, in the flattening task, the trajectory of the bucket's tip usually moves straight line when it contacts with the ground. So, with the conventional configuration, operator must estimate the inverse kinematic of the boom, arm and bucket. Meanwhile, with the new model, user only control the forward-backward direction.

4. VIRTUAL EXCAVATOR SIMULATION USING INTUITIVE CONFIGURATION OF JOYSTICK

4.1 Simulation setup

A virtual excavator simulation is contributed to apply the new joystick control method. The schematic of the model is shown in Fig. 5. Besides, the motion of the dynamic model of excavator is describe in the Virtual Reality Interface (VRI). This VRI is described in Fig. 6 ensures the operator be able to observe the relation between the attachment and the environment where the excavator is operating.
It is assumed that the coordinate of point O4 \( \equiv N \) are given in the coordinate system. In this simulation, the user operates the twin joystick to set the velocity of the horizontal movement, base rotation, vertical movement and the dig angle.

The simulation for conventional configuration of excavator is also conducted to compare the performance between two methods.

### 4.2 Simulation result

#### 4.2.1 Path and trajectory result

The trajectory and path of the bucket's tip of the intuitive configuration method and conventional configuration method are shown in Fig. 6 and Fig. 7. The path describes two process of flattening task. First step, the tip begins moving from the inertial point to the start point. Second, it will contact the ground to flatten the soil following the horizontal direction.
FIGURE 8 Trajectory of horizontal and vertical movement of intuitive configuration method (left) and conventional configuration method (right).

From the Fig. 7 and Fig. 8, the path of the new method is linear than the conventional method in both horizontal movement and vertical movement. It means that the motion of the bucket’s tip in new method is smoother than the current one. Moreover, the time consuming of the intuitive configuration method is less than the conventional method. Therefore, the proposed method has advantage in time saving.

4.2.2 Operation effort

The input signals of the twin joystick for both methods are shown in Fig. 9. Following these result, to complete the flattening task, the new method requires five joystick manipulations. However, in conventional method, it took 14 manipulations to finish the work. It is clearly shown that the new method is more simple in operation than the current method.

Finally, based on the above result, the automatic method takes more advantages than conventional method in flattening task in:
- Simple Control
- Intuitive interface to determine linear motion
- Cost less time to complete the task
- Less difficulty for training the new worker

5. CONCLUSION

This paper presents the design of the intuitive joystick configuration in flattening task of excavator. The suggested interface scheme with two electric joysticks is composed to perform independent horizontal and vertical motions. A virtual excavator simulation controlled by two real joysticks is developed.

ACKNOWLEDGEMENT

This work was supported by the Industrial Strategic technology development, 10052968, Development of core components and systems for special purposed machine in disaster environment funded By the Ministry of Trade, Industry & Energy (MI, Korea)

REFERENCES

ONLINE PARAMETER ESTIMATION OF HYDRAULIC SYSTEM BASED ON UNSCENTED KALMAN FILTER

Takashi YAMADA*, Yoshiharu NISHIDA* and Akira TSUTSUI*

*Technical Development Group, Kobe Steel, Ltd.
1-5-5, Takatsuka-dai, Nishi-ku, Kobe, Hyogo, 651-2271 Japan
(E-mail: yamada.takashi-1@kobelco.com)

Abstract. An online parameter estimation of a hydraulic system based on the unscented Kalman filter was evaluated in simulation. It was confirmed that the proposed method can achieve a practical estimation even though not all of the states are measured and the measured signals contain noise. The bulk modulus and the discharge coefficient were chosen as estimation parameters in the simulation. These parameters were added to a state vector of a state space model and estimated by the unscented Kalman filter algorithm. The simulation result showed that the maximum estimation error of the bulk modulus was 0.5[%] and that of the discharge coefficient was 2.8[%].

Keywords: Parameter Estimation, Unscented Kalman Filter, Online Estimation

INTRODUCTION

Accurate and efficient control of a hydraulic system is still a challenging problem because of its high nonlinearity. To achieve this, model-based control theories such as model predictive control and adaptive control have been proposed [1][2][3]. Performances of these methods depend on the degree of precision of the model. Therefore, we need to make the physical parameters of hydraulic systems as precise as possible. However, there are a lot of hydraulic parameters which vary dynamically and cannot be measured directly. For example, it is known that the bulk modulus varies depending on pressure, temperature, and the content rate of air [4] though it has a great influence on a system. To identify such parameters, an online parameter estimation can be a practical solution.

The least square (LS) is one of the most widely used estimation methods for hydraulic systems. GHAZALI et al. [5] proposed the recursive LS estimation method with varying forgetting factor, BARANOWSKI and TUTAJ [6] proposed the moving horizon estimation method which is an extension of the LS, and MATSUMOTO and SAKAI [7] proposed the online estimation method based on the recursive LS. The LS has advantages in its low calculation and implementation cost. However, generally, all of the system states need to be measurable on the LS estimation, and the LS estimation accuracy for nonlinear systems is highly influenced by measurement errors which often exist in hydraulic systems.

On the other hand, estimation methods based on the Bayesian inference can compensate for those disadvantages because they treat state variables and noises as random values. The extended Kalman filter (EKF) [8] is a major nonlinear estimation method. However, the estimation error of the EKF is not negligible if nonlinearity of the system is large. This is because the EKF approximates a nonlinear system by the 1st order Taylor series. To improve the EKF, the unscented Kalman filter (UKF) [9] has been proposed. The UKF approximates the statistics of state variables of nonlinear systems with a mathematical transformation called the unscented transformation. It is known that the unscented transformation is accurate to the 3rd order for Gaussian inputs and to at least the 2nd order for any inputs [10]. Because of this, the UKF makes better estimation results than the EKF. The particle filter [11] is another established nonlinear estimation method. The particle filter has less estimation errors than the EKF and UKF. However, its calculation cost is much larger and this makes the particle filter unsuitable for an online estimation.

In this paper, we applied the UKF to an estimation of hydraulic parameters. The bulk modulus and the discharge coefficient were chosen as the estimation parameters in simulation. These parameters were added to a state vector of a state space model and estimated by the UKF algorithm. The simulation result showed that the maximum estimation error of the bulk modulus was 0.5[%] and that of the discharge coefficient was 2.8[%], even though not all of the states are measured and the measured signals contain noise.

UNSCENTED KALMAN FILTER

In this section, online estimation algorithm of the UKF [9] is briefly described. When we apply the Kalman filter to nonlinear systems, calculation of propagation of state variable’s statistics becomes a problem. The UKF
approximates the statistics with a mathematical transformation called the unscented transformation. Subsequently, it applies the approximated statistics to the Kalman filtering theory. Consider a discrete nonlinear dynamic system expressed as follows:

\[ x_{k+1} = f(x_k, u_k, v_k) \]  \hspace{1cm} (1)

\[ y_k = h(x_k, w_k) \]  \hspace{1cm} (2)

\[ E[v] = 0 \]  \hspace{1cm} (3)

\[ E[w] = 0 \]  \hspace{1cm} (4)

where \( x \) is the state vector, \( u \) is the control input vector, \( v \) is the process noise vector, \( y \) is the measurement signal vector, \( w \) is the measurement noise vector, and the subscripts indicate discrete time. Note that \( x, v, y, \) and \( w \) are random variables.

First, the UKF approximately calculates propagation of the mean and covariance of \( x_{k-1} \) through equation (1) and (2) using the unscented transformation. In the unscented transformation, sample points called sigma points are used. The sigma point matrix \( X_{k-1} \) which consists of sigma point vectors is calculated as

\[ X_{k-1} = \begin{bmatrix} \hat{x}_{k-1} & \hat{x}_{k-1} + \sqrt{L + \lambda \hat{P}_{k-1}} & \hat{x}_{k-1} - \sqrt{L + \lambda \hat{P}_{k-1}} \end{bmatrix} \]  \hspace{1cm} (5)

where \( \hat{x}_{k-1} \) and \( \hat{P}_{k-1} \) are the mean and covariance matrix of \( x_{k-1} \) estimated by the UKF at the previous time step, \( L \) is the dimension of \( x \), \( \sqrt{\hat{P}_{k-1}} \) is the lower triangular Cholesky factorization of \( \hat{P}_{k-1} \), and \( \lambda \) is a scaling parameter given by

\[ \lambda = \alpha^2 (L + \kappa) - L. \]  \hspace{1cm} (6)

\( \alpha \) and \( \kappa \) in equation (6) are tuning parameters. Using the sigma point matrix, the mean and covariance of \( x_k \) and \( y_k \) are calculated approximately as

\[ \begin{cases} \hat{x}_k^- = \sum_{i=0}^{2L} W_i^m X_{k,i}^- \ \\
\hat{P}_k^- = \sum_{i=0}^{2L} \left[ W_i^c \left[ X_{k,i}^- - \hat{x}_k^- \right] \left[ X_{k,i}^- - \hat{x}_k^- \right]^T \right] + Q \end{cases} \]  \hspace{1cm} (7)

\[ \begin{cases} \hat{y}_k^- = \sum_{i=0}^{2L} W_i^m Y_{k,i}^- \ \\
\hat{P}_{y_k | y_k}^- = \sum_{i=0}^{2L} \left[ W_i^c \left[ Y_{k,i}^- - \hat{y}_k^- \right] \left[ Y_{k,i}^- - \hat{y}_k^- \right]^T \right] + R \end{cases} \]  \hspace{1cm} (8)

where \( \hat{x}_k^- \) is the approximated mean of \( x_k \), \( \hat{P}_k^- \) is the approximated covariance matrix of \( x_k \), \( \hat{y}_k^- \) is the approximated mean of \( y_k \), \( \hat{P}_{y_k | y_k}^- \) is the approximated covariance matrix of \( y_k \), \( Q \) is the covariance matrix of \( v \), \( R \) is the covariance matrix of \( w \), \( X_{k,i}^- \) is \( i \)th column of \( X_k^- \), \( Y_{k,i}^- \) is \( i \)th column of \( Y_k^- \), and \( X_k^- \), \( Y_k^- \), \( W_i^m \), and \( W_i^c \) are given by

\[ X_{k,i}^- = f(X_{k-1,i}, u_{k-1}, 0) \]  \hspace{1cm} (9)

\[ Y_{k,i}^- = h(X_{k,i}^-) \]  \hspace{1cm} (10)

\[ W_0^m = \frac{\lambda}{L+\lambda} \]  \hspace{1cm} (11)

\[ W_0^c = \frac{\lambda}{L+\lambda} + 1 - \alpha^2 + \beta \]  \hspace{1cm} (12)

\[ W_i^m = W_i^c = \frac{1}{2(L+\lambda)} \]  \hspace{1cm} for \( i = 1, \ldots, 2L \).  \hspace{1cm} (13)
\( \beta \) in equation (12) is a tuning parameter. The calculation described above is called the unscented transformation. It is known that the approximation of the unscented transformation is accurate to the 3rd order for Gaussian inputs and to at least the 2nd order for any inputs [10]. Then, the UKF estimates the mean and covariance of \( x_k \) applying the approximated statistics to the Kalman filtering theory [8]. The estimation result of the UKF is calculated as follows:

\[
\begin{align*}
\hat{x}_k &= \hat{x}_k^- + K_k (y_k - \hat{y}_k^-) \\
\hat{p}_k &= \hat{p}_k^- - K_k \hat{y}_k^- K_k^T
\end{align*}
\]

(14)

where \( K_k \) is given by

\[
K_k = \sum_{i=0}^{2L} \left\{ W_i \left[ X_{k,i}^- - \hat{x}_k^- \right]\left[ Y_{k,i}^- - \hat{y}_k^- \right]^T \right\} \left( \hat{y}_{k,y_k}^- \right)^{-1}.
\]

(15)

MODELING OF HYDRAULIC SYSTEM

In this section, mathematical models of a considered hydraulic system are described. Figure 1 shows a schematic of the considered system. Dynamics of the system is expressed as

\[
m \ddot{x}_c = A_h P_h - A_r P_r
\]

(16)

\[
\dot{P}_p = \frac{K}{V_p} (Q_p - Q_{hin} - Q_{rin} - Q_R)
\]

(17)

\[
\dot{P}_h = \frac{K}{V_h + A_h x_c} (Q_{hin} - Q_{hout} - A_h \dot{x}_c)
\]

(18)

\[
\dot{P}_r = \frac{K}{V_r + A_r (l - x_c)} (Q_{rin} - Q_{rout} + A_r \dot{x}_c)
\]

(19)

\[P_t = 0\]

(20)

where \( P_p, P_h, P_r, \) and \( P_t \) are the pressures defined on figure 1, \( x_c \) is the position of the piston, \( K \) is the bulk modulus of the oil, \( V_p, V_h, \) and \( V_r \) are the capacities of conduits pressurized by \( P_p, P_h \) and \( P_r \) respectively, \( A_h \) and \( A_r \) are the cross-sectional areas of the piston, \( l \) is the length of the piston, \( m \) is the mass of the piston, and \( Q_p, Q_{hin}, Q_{rout}, Q_{rin}, Q_{hout}, \) and \( Q_R \) are the flow rates defined on figure 1. The flow rates are modeled by an orifice equation as

\[Q_p = \text{constant}\]

(21)

\[Q_{hin} = CA_{ch} \sqrt{\frac{2}{\rho}} \text{sign}(P_p - P_h) \sqrt{|P_p - P_h|}\]

(22)

\[Q_{rout} = CA_{ch} \sqrt{\frac{2}{\rho}} \text{sign}(P_r - P_t) \sqrt{|P_r - P_t|}\]

(23)

\[Q_{rin} = CA_{cr} \sqrt{\frac{2}{\rho}} \text{sign}(P_p - P_r) \sqrt{|P_p - P_r|}\]

(24)

\[Q_{hout} = CA_{cr} \sqrt{\frac{2}{\rho}} \text{sign}(P_h - P_t) \sqrt{|P_h - P_t|}\]

(25)

\[Q_R = CA_R \sqrt{\frac{2}{\rho}} \text{sign}(P_p - P_t) \sqrt{|P_p - P_t|}\]

(26)

where \( C \) is the discharge coefficient and \( \rho \) is the density of the oil. \( A_{ch}, A_{cr}, \) and \( A_R \) are the opening areas of the valves which are functions of the spool position \( x_v \) or the pressure \( P_p \):

\[A_{ch} = \begin{cases} \alpha x_v & \text{if } x_v > 0 \\ 0 & \text{if } x_v < 0 \end{cases}\]

(27)
In this section, an estimation of the bulk modulus and the discharge coefficient based on the UKF is evaluated in simulation. As mentioned before, the bulk modulus varies dynamically and cannot be measured directly though it has a great influence on a system. The discharge coefficient is also an unmeasurable parameter. The simulation result showed that the proposed method can achieve a practical estimation even though not all of the states are measured and the measured signals contain noise.

Formulation of State Space Equation

It is known that the secant bulk modulus $K$ and the tangent bulk modulus $\beta$ can be approximated as a function of the pressure [12]:

$$ K = K_0 + K_p P $$

$$ \beta = K_0 \left[ 1 + \frac{(K_p - 1)P}{K_0} \right] \left[ 1 + \frac{K_p P}{K_0} \right] $$

(30)

(31)

where $K_0$ is the secant bulk modulus at zero gauge pressure and $K_p$ is the coefficient of dependence on pressure. For simplicity, the secant bulk modulus $K$ is used as an estimation parameter in this section though the tangent bulk modulus $\beta$ can be estimated in the same manner. The discharge coefficient $C$ is assumed to be a constant. Here, our goal becomes to estimate $K_0$, $K_p$, and $C$. To formulate the state space equation, the state vector is defined as

$$ x = [P_p \quad P_h \quad P_r \quad x_c \quad \dot{x}_c \quad K_0 \quad K_p \quad C]^T. $$

(32)
Note that the estimation parameters $K_0$, $K_p$, and $C$ are added to the state vector. Then the state space equation is expressed nonlinearly as follows:

$$
\begin{align*}
\dot{\mathbf{x}} &= \begin{bmatrix} \dot{P}_p \\
\dot{P}_h \\
\dot{P}_r \\
\dot{x}_c \\
\dot{K}_0 \\
\dot{K}_p \\
C 
\end{bmatrix} = \begin{bmatrix} 
\frac{K_0+K_pP_p}{v_p} (Q_p - Q_{hin} - Q_{rin} - Q_R) \\
\frac{K_0+K_pP_h}{v_h} (Q_{hin} - Q_{hout} - A_h \dot{x}_c) \\
\frac{K_0+K_pP_r}{v_r+A_r(l-x_c)} (Q_{rin} - Q_{rout} + A_r \dot{x}_c) \\
0 \\
0 \\
0 \\
0 
\end{bmatrix} + \nu
\end{align*}
$$

(33)

where $\nu$ is the process noise vector. In the simulation, only $P_p$, $P_h$, $P_r$, and $x_c$ are measured, which is a typical experiment condition. Therefore, the measurement signal vector $\mathbf{y}$ is defined as follows:

$$\begin{align*}
\mathbf{y} &= \begin{bmatrix} P_p \\
P_h \\
P_r \\
x_c \\
K_0 \\
K_p \\
C 
\end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 1 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 1 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 
\end{bmatrix} \begin{bmatrix} P_p \\
P_h \\
P_r \\
x_c \\
K_0 \\
K_p \\
C 
\end{bmatrix} + \mathbf{w}
\end{align*}
$$

(34)

where $\mathbf{w}$ is the measurement noise vector. The equation (33) and (34) are discretized by the 4th order Runge-Kutta method. The UKF are applied to the discretized equations to estimate $K_0$, $K_p$, and $C$ as well as the other states.

### Simulation Conditions

The dynamic behavior of the system was simulated solving equation (33) without $\nu$ numerically. Table 1 shows the parameters used in the simulation. Table 2 shows the initial conditions of the simulation and the UKF estimation. We added errors to the initial estimation parameters to evaluate a response of the UKF. Figure 2 shows $x_p$, $A_{ch}$, $A_{cr}$ and $A_R$ used in the simulation. $\nu$ and $\mathbf{w}$ were assumed to be Gaussian noise. The mean and the covariance matrix of $\nu$ and $\mathbf{w}$ were defined as follows:

$$E[\nu] = 0$$

(35)

$$E[\mathbf{w}] = 0$$

(36)

$$Q = \text{diag}(10[\text{kPa}]^2, 10[\text{kPa}]^2, 10[\text{kPa}]^2, 5[\text{mm}]^2, 5[\text{mm/s}]^2, 1[\text{MPa}]^2, 0.1^2, 0.01^2)$$

(37)

$$R = \text{diag}(1[\text{MPa}]^2, 1[\text{kPa}]^2, 1[\text{kPa}]^2, 0.5[\text{mm}]^2)$$

(38)

where $Q$ is the covariance matrix of $\nu$ and $R$ is the covariance matrix of $\mathbf{w}$. Values of $R$ were determined by typical sensor specifications and values of $Q$ were determined by an assumption that the state space equation is 10 times less accurate than the measured values.
TABLE 1. Simulation parameters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Value</th>
<th>Symbol</th>
<th>Value</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>m</td>
<td>100[kg]</td>
<td>V_p</td>
<td>9.42 × 10^{-4}[m³]</td>
<td>K_0</td>
<td>1600[MPa]</td>
</tr>
<tr>
<td>A_h</td>
<td>2.00 × 10^{-3}[m²]</td>
<td>V_h</td>
<td>9.42 × 10^{-4}[m³]</td>
<td>K_p</td>
<td>5.0</td>
</tr>
<tr>
<td>A_r</td>
<td>2.00 × 10^{-3}[m²]</td>
<td>V_r</td>
<td>9.42 × 10^{-4}[m³]</td>
<td>C</td>
<td>0.6</td>
</tr>
<tr>
<td>α</td>
<td>1.96 × 10^{-3}[m]</td>
<td>l</td>
<td>1.0[m]</td>
<td>α</td>
<td>1.0</td>
</tr>
<tr>
<td>b</td>
<td>2.24 × 10^{-6}[m²/MPa]</td>
<td>ρ</td>
<td>8.69 × 10^{-2}[kg/m³]</td>
<td>β</td>
<td>2.0</td>
</tr>
<tr>
<td>P_R</td>
<td>3.0[MPa]</td>
<td>Q_R</td>
<td>8.0[l/min]</td>
<td>c</td>
<td>-5.0</td>
</tr>
</tbody>
</table>

TABLE 2. Initial conditions of simulation and UKF estimation.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Initial state of Simulation</th>
<th>Initial state of UKF estimation</th>
<th>Initial variance of UKF estimation</th>
</tr>
</thead>
<tbody>
<tr>
<td>p_p</td>
<td>0[MPa]</td>
<td>0[MPa]</td>
<td>0.1[MPa]^2</td>
</tr>
<tr>
<td>p_h</td>
<td>0[MPa]</td>
<td>0[MPa]</td>
<td>0.1[MPa]^2</td>
</tr>
<tr>
<td>p_r</td>
<td>0[MPa]</td>
<td>0[MPa]</td>
<td>0.1[MPa]^2</td>
</tr>
<tr>
<td>x_c</td>
<td>0.5[m]</td>
<td>0.5[m]</td>
<td>1.0[mm]^2</td>
</tr>
<tr>
<td>x_\dot{}c</td>
<td>0[m/s]</td>
<td>0[m/s]</td>
<td>1.0[mm]^2</td>
</tr>
<tr>
<td>K_0</td>
<td>1600[MPa]</td>
<td>1500[MPa]</td>
<td>100[MPa]^2</td>
</tr>
<tr>
<td>K_p</td>
<td>5.0</td>
<td>5.0</td>
<td>10^2</td>
</tr>
<tr>
<td>C</td>
<td>0.6</td>
<td>0.3</td>
<td>0.3^2</td>
</tr>
</tbody>
</table>

(a) Spool position of control valve: x_v  
(b) Opening areas of control valve: A_{ch} and A_{cr}  
(c) Opening area of relief valve: A_R

FIGURE 2. Functions of valves used in simulation

Estimation Results

Figure 3 shows P_p, P_h, P_r, and x_c without the measurement noise on the simulation. Figure 4 shows the estimated K_0, K_p, C, and the bulk modulus K_h at the cylinder head chamber. K_h was calculated by the following equation using the estimated parameters:

\[
K_h = K_0 + K_p P_h.  \tag{39}
\]

Figure 4 shows that the initial errors were eliminated within 0.05[s]. Table 3 shows the maximum estimation errors after t =0.05[s].
FIGURE 3. Simulation results

FIGURE 4. Estimation results

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Maximum error</th>
<th>Ratio to true value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K_0$</td>
<td>7.38[MPa]</td>
<td>0.5[%]</td>
</tr>
<tr>
<td>$K_p$</td>
<td>0.78</td>
<td>15.6[%]</td>
</tr>
<tr>
<td>$C$</td>
<td>$1.68 \times 10^{-2}$</td>
<td>2.8[%]</td>
</tr>
<tr>
<td>$K_h$</td>
<td>7.67[MPa]</td>
<td>0.5[%]</td>
</tr>
</tbody>
</table>
Discussion

The result showed that the proposed method could compensate for the initial errors within 0.05[s]. The remaining errors were due to the measurement noise, the discretization error of the Runge-Kutta method, and the approximation error of the unscented transformation. As a result, the maximum estimation error of the bulk modulus was 0.5[%] and that of the discharge coefficient was 2.8[%] after the initial errors were compensated. Although needed estimation accuracy depends on system requirements, generally, a parameter estimation whose error is less than 3[%] can be used as a practical tool, especially if we do not have backgrounds of the parameters.

On the domains that \( x_v \) was close to zero, the oil was highly pressurized and direction of the flow was reversed, i.e., the states were in great transition. On the other hand, the states were almost steady on the other domains. From the standpoint of control engineering, a system in transition is generally more observable than a system in a steady state. This is the reason why the standard deviation of \( K_0 \) decreased in the transitional domains and increased in the other domains.

Contrary to \( K_0 \), the standard deviation of \( C \) increased on the transitional domain. This is simply because the equations (22)-(26) which have information about \( C \) became almost zero on the domain.

In this section, the bulk modulus was modeled as a function of the pressure, and the discharge coefficient was modeled as a constant. Strictly speaking, the bulk modulus depends on pressure, temperature, and the content rate of air [4], and the discharge coefficient depends on Reynolds number [13]. However, even if these un-modeled relations exist, tuning of \( Q, \alpha, \beta \), and \( \kappa \) in the unscented transformation can compensate for them to a certain extent. This is because the UKF estimates states statistically based on the covariance matrices. Of course, if mathematical models of the relations are known, they can be implemented in the state space equation in the same manner mentioned in this section.

CONCLUSION

An online parameter estimation of hydraulic system based on the UKF was evaluated in the simulation. The simulation result showed that the proposed method can achieve a practical estimation even though not all of the states are measured and the measured signals contain noise.

REFERENCES

ON THE NONDIMENSIONALIZATION OF NOMINAL HYDRAULIC CYLINDER DYNAMICS

Satoru SAKAI

*Department of Mechanical Engineering, Faculty of Engineering, Shinshu University
Wakasato 4-17-1, Nagano, 380-0928 Japan
(E-mail: satorus@shinshu-u.ac.jp)

Abstract. First, the special nondimensionalization of nominal hydraulic cylinder dynamics is reviewed and the nonlinear effects by the piston asymmetry are introduced. Second, the nonlinear effects by the piston asymmetry and that by the pipeline asymmetry are numerically compared with each other via the special nondimensionalization. In our numerical study, the nonlinear effects are less sensitive to the pipeline asymmetry.

Keywords: Modeling, Simulation, Control

INTRODUCTION

In comparison with electric systems, not only nonlinearity but also many physical parameters increase the difficulty in design, analysis, and control of hydraulic systems. Of course, it is well known that nondimensional representations exactly simplify the original representation with respect to the parameter space. For example, the well-known mass-damper-spring dynamics with 3 parameters (the mass, the damping, and the spring) in the original representation is exactly simplified into a new mass-damper-spring dynamics with only 1 parameter (the nondimensional damping). It is very important to develop such nondimensional representations of the nominal hydraulic cylinder dynamics as well.

In our previous works, a special nondimensional representation is developed. Unlike the usual nondimensional representation, the developed nondimensional representation preserves the physical structure, that is, the developed nondimensional representation with 10 parameters in rotation case can be equal to a special case of the original representation with only 5 parameters. However, many numerical properties of the developed nondimensionalization are not studied yet.

In this paper, asymmetry of the nominal hydraulic cylinder dynamics is numerically studied partially. First, the developed nondimensional representation is reviewed briefly. Second, with respect to the nonlinear effects in translation case, the piston area asymmetry and the pipeline length asymmetry are compared comprehensively due to our special nondimensional representation.

THE NONDIMENSIONALIZATION

Let us consider the nominal hydraulic cylinder dynamics in the original representation [1]:

FIGURE 1. Nominal hydraulic cylinder
Due to the special nondimensional representation \([2][3]\), without loss of generality, we can assume that is, the 8 dimensional parameter space is exactly simplified into the 3 dimensional parameter space in translation case. (The rotation case is skipped in this paper.)

\[ \Sigma_0 \begin{cases} M \frac{d^2 s}{dt^2} = -D \frac{ds}{dt} + A_+ p_+ - A_- p_- \\ \frac{dp_+}{dt} = \frac{b}{A_+ (L/2 + s(t))} \left[ -A_+ \frac{ds}{dt} + Q_+(p_+, u) \right] \\ \frac{dp_-}{dt} = \frac{b}{A_- (L/2 - s(t))} \left[ +A_- \frac{ds}{dt} - Q_-(p_-, u) \right] \end{cases} \]

and

\[ Q_+ = B(p_+, +u)u, \quad Q_- = B(p_-, -u)u \]

with

\[ B(r, u) = \begin{cases} C\sqrt{-r + F} & (u > 0) \\ 0 & (u = 0) \\ C\sqrt{+r - 0} & (u < 0) \end{cases} \]

whose parameters are defined in Figure 1. Due to the special nondimensional representation [2][3], without loss of generality, we can assume

\[ (M, A_+, L, b, C) = (1, 1, 1, 1, 1) \]

that is, the 8 dimensional parameter space is exactly simplified into the 3 dimensional parameter space in translation case. (The rotation case is skipped in this paper.)

![Graphs showing nonlinear effect by the piston asymmetry](image.png)

**FIGURE 2.** A nonlinear effect by the piston asymmetry (CASE(a), CASE(c))

**Piston-Asymmetry and Pipeline-Asymmetry**

Figure 2 shows that the piston asymmetry (the piston area asymmetry) generates the nonlinear behaviors by which the control performance can be lost. One may jump to think that this nonlinear behavior is caused by the nonlinear friction effects. However, this conjecture is not true. In fact, the nominal hydraulic cylinder dynamics does not have such nonlinear friction terms. If the linear approximation is applied, it is needless to say that the nonlinear piston behaviors disappear even in the presence of the piston asymmetry.

On the other hand, in many practical situations, since we can have the pipeline asymmetry (the pipeline length asymmetry), there is a possibility to design new pipelines so as to cancel or reduce the nonlinear piston behaviors coming from the piston asymmetry. In a word, it is not clarified whether a certain pipeline-asymmetry cancels or reduces the piston asymmetry or not.

In the following, let us consider the modified nominal hydraulic cylinder dynamics in the original representation:


\[
\begin{align*}
M \frac{d^2 s}{dt^2} &= -D \frac{ds}{dt} + A_+ p_+ - A_- p_-
\frac{dp_+}{dt} &= \frac{b}{A_+(L_+/2 + s(t))} \left[ -A_+ \frac{ds}{dt} + Q_+(p_+, u) \right]
\frac{dp_-}{dt} &= \frac{b}{A_-(-L_-/2 - s(t))} \left[ +A_- \frac{ds}{dt} - Q_-(p_-, u) \right]
\end{align*}
\]

which has the pipeline asymmetry:

\[
V_+(s(t)) := A_+(L_+/2 + s(t)) = A_+(L/2 + s(t)) + \tilde{V}_+
\]

\[
A_+(L/2 + s(t)) + \tilde{V}_+
\]

\[
V_-(s(t)) := A_-(L_-/2 - s(t)) = A_-(L/2 - s(t)) + \tilde{V}_-
\]

In this paper, with respect to the nonlinear effects, the piston area asymmetry and the pipeline length asymmetry are compared numerically. The modified backward differential formula with the variable step is applied (20-sim, Ver. 4.1, 64-bit CPU 2.60 GHz, Memory 8.0 GB). See [3] for more details.

Figure 3 shows the linearization error in the nondimensional parameter space. The color bar depicts the magnitude of the linearization error which is the output signal difference between the nondimensional transfer function (the linearized model) and the nondimensional nominal hydraulic cylinder dynamics (the nonlinear model). The vertical axis corresponds to the piston asymmetry and the two horizontal axis correspond to the nondimensional pipeline length. The same sinusoidal inputs are applied to the both models.

At any nondimensional frequency, unexpectedly, the linearization error is very sensitive to the piston asymmetry but not sensitive to the pipeline asymmetry. These results justify our previous results, that is, the nondimensional nominal hydraulic cylinder dynamics without pipeline asymmetry.

**FIGURE 3.** Linearization error (LEFT: Low nondimensional frequency, RIGHT: Very high nondimensional frequency).

### CONCLUSION

This paper numerically studies the nonlinear effects by the piston asymmetry and that by the pipeline asymmetry. Since our nondimensional representation preserves the parametric structure of the original representation, we can study very comprehensively. In every nondimensional frequency, the nonlinear effects by the piston asymmetry is much stronger than that by the pipeline asymmetry in this paper. These results justify the nondimensional nominal hydraulic cylinder dynamics without pipeline asymmetry, but at the same time, imply that we need not any pipeline design but some control to overcome the nonlinear behavior in Figure 2. In our next work, these results are investigated under the nondimensional damping and the nondimensional source pressure.

### REFERENCES

[2A18-22] H10 (Control & Measurements 3)
Chair: Kazuhisa Ito (Shibaura Institute of Technology), Wataru Kobayashi (Okayama University of Science)
Thu. Oct 26, 2017 5:00 PM - 6:20 PM Room A (ACROS Fukuoka)

[2A18] RESEARCH ON THE CHARACTERISTICS OF CONSTANT-SPEED STRETCH OF A HIGH-SPEED TENSILE MACHINE CONTROLLED BY THE ELECTRO-HYDRAULIC SERVO SYSTEM
*Enze Zhu¹, Guanglin Shi¹ (1. Shanghai Jiao Tong University)
5:00 PM - 5:16 PM

[2A19] DEVELOPMENT OF FLEXIBLE ELECTRO-HYDRAULIC CYLINDER FOR FLEXIBLE SPHERICAL ACTUATOR
*Hiroaki Tamaki¹, Shujiro Dohta¹, Tetsuya Akagi¹, Wataru Kobayashi¹, Yasuko Matsui¹ (1. Okayama University of Science)
5:16 PM - 5:32 PM

[2A20] HYDRAULIC RESONANCE CHARACTERISTICS OF THE HIGH-FREQUENCY EXCITATION SYSTEM CONTROLLED BY A 2D ROTARY VALVE
*Yan REN¹, Hesheng TANG¹, Jian RUAN² (1. Department of Mechanical and Electrical Engineering, Wenzhou University, 2. Department of Mechanical Engineering, Zhejiang University of Technology)
5:32 PM - 5:48 PM

[2A21] PERCEIVED STIMULI IN HYDRAULIC OPERATION LEVER OF CONSTRUCTION MACHINERY
*Hironao Yamada¹, Fumichika Okada², Katsutoshi Otsubo¹, Takuya Kawamura¹ (1. Dept. of Mechanical Engineering, Gifu Univ., 2. Toyota Motor Corporation)
5:48 PM - 6:04 PM

[2A22] A NOVEL INTEGRATED LOAD-SENSING ELECTRO-HYDRAULIC ACTUATOR FOR AIRCRAFT STRUCTURAL TESTS
*Yaoxing Shang¹, Xiaochao Liu¹, Zongxia Jiao¹, Jiaokang Wu¹, Liang Yan¹ (1. Beihang University)
6:04 PM - 6:20 PM
Research on the Characteristics of Constant-speed Stretch of a High-speed Tensile Machine Controlled by the Electro-hydraulic Servo System

Enze Zhu*, Guanglin Shi**

School of Mechanical Engineering, Shanghai Jiao Tong University
800 Dongchuan Road, Minhang District, Shanghai, 200240, China
(E-mail: *ezzhu@foxmail.com, **glshi@263.net)

Abstract. Close-loop control strategy is widely used in system control and it does work in most situations. But when it comes high-speed tensile machines, it may fail because of inevitable delay of control components, such as valves. To control this kind of high-speed systems and realize a constant-speed stretch, suitable parameters are crucial, but they are achieved only from data of repeated experiments. Luckily, neural network is good at this kind of work. It can make the most proper parameters after several rounds of experiments by self-learning. This control strategy will finally do better than an open-loop-only one and even as well as a close-loop one after plural rounds.

Keywords: High-speed Tensile Machine, Constant-speed Stretch, Electro-hydraulic Servo, Neural Network, Self-learning

INTRODUCTION

Tensile machines are widely used in material testing. Classified by axials, there are biaxial tensile machines for sheet materials and uniaxial machines for conventional materials [1]. Classified by categories of materials, there are tensile tests for steel, fiberglass, plastics and so on [2-3]. Classified by strain rates, there are quasi-static and dynamic material tests. [4] Classified by testing strategies, there are conventional direct tensile test for materials, such as steels, and indirect tensile tests, such as Brazilian test, ring test and bending test for brittle solids like rocks [5]. With the fast development of the automotive industry, people pay more attention to the characteristics, especially dynamic properties, of materials used on automobiles. In some terrible traffic accidents, one automobile may crash into another at a speed of at least 20m/s, causing structures of themselves deformed or broken by tensile or pressure forces at a rather high speed. To simulate this kind of situation, researchers need to stretch specimens with a high speed and keep the speed constant. It is difficult for conventional tensile machines to do so, because close-loop control systems can hardly respond to such a high speed of stretch (up to 20 m/s) in such a short time (several milliseconds), if take the responding time of system elements into consideration. A high-speed tensile machine with a novel control strategy is urgently needed and has a bright future. High-rate Instron VHS 8800 hydraulic testing machine is a typical high-speed tensile machine. It has a load capacity of 100kN and use a VHS software to keep velocities up to 25m/s constant [6]. Another new kind of high-speed tensile machine is driven by a linear motor, which can provide a strain rate from 1s⁻¹ to 100s⁻¹ [7].

This research is aimed to develop a high-speed tensile machine with a load capacity of 50kN and being able to achieve a constant velocity of stretch up to 20m/s. The high-speed tensile machine is used to stretch different kinds of specimens, then observe the fracture. Some of the specimens are likely of high strength. It means that the system should provide a large tensile force (up to 50kN) as well as a high speed of stretch (up to 20m/s) at the same time. An electro-hydraulic servo system is a good choice to meet both the requirements. A real-time close-loop control system is ineffective to control the high-speed system. Besides, the high-speed system is a nonlinear system and a simple open-loop system is of little use to it. However, although the relationship between control signals and speed of stretch is nonlinear and unknown, it can be studied by a neural network. After tens of rounds of self-study and self-adjustment, the neural network will have the capacity to control the system well.

SYSTEM DESIGN

FIGURE 1 is about the hydraulic scheme of the stretcher. Only stretch-related functional areas are shown in the figure.
FIGURE 1. Hydraulic Scheme

As is shown in FIGURE 1, oil is pumped from fuel tank into the hydraulic system. At the beginning, Valve 22 and Valve 23 are closed, oil are pumped directly into Accumulator 24. Then, Valve 22 and Valve 23’s cores move to right, which means oil from accumulators and pump will flow into the lower chamber of hydraulic cylinder and the piston is going to be lifted. The piston has some free space to accelerate to the set target, then tension begins. During the stretch, the speed is hoped to be kept constant. This is realized by control of the opening size of Valve 22 and 23. When speed is larger than 20m/s, the opening size decreases, and when speed is smaller than 20m/s, the opening size increases. After fracture, force sensors will give a feedback to stop the piston and one tensile test is completed.

A vertical tensile machine model is shown in FIGURE 2. The speed is so fast that a clearance hydraulic cylinder, rather than an ordinary hydraulic cylinder, is used to reduce the friction during stretch of specimens [8]. The clearance hydraulic cylinder and several accumulators are set on the upper platform. The upper platform is able to be adjusted in height by two small hydraulic cylinders. A specimen will be fully clamped in the lower static grips on the base and upper motive grips on the above. The upper grips are linked to the piston rod with a structure, which allows the piston to have space for speeding up, then stretch the specimen.

FIGURE 2. Tensile Machine Model
The main parameters of the system are shown in TABLE 1.

<table>
<thead>
<tr>
<th>Components</th>
<th>Parameters</th>
<th>Values</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pump (6)</td>
<td>Flow Rate</td>
<td>30</td>
<td>L/min</td>
</tr>
<tr>
<td>Over Flow Valve (14)</td>
<td>Pressure Relief</td>
<td>280</td>
<td>bar</td>
</tr>
<tr>
<td>Accumulator (24)</td>
<td>Nominal Volume</td>
<td>6</td>
<td>L</td>
</tr>
<tr>
<td>Accumulator (25)</td>
<td>Permitted Pressure</td>
<td>330</td>
<td>bar</td>
</tr>
<tr>
<td>Hydraulic Cylinder (26)</td>
<td>Piston Diameter</td>
<td>80</td>
<td>mm</td>
</tr>
<tr>
<td></td>
<td>Rod Diameter</td>
<td>56</td>
<td>mm</td>
</tr>
<tr>
<td></td>
<td>Length of Stroke</td>
<td>0.3</td>
<td>m</td>
</tr>
</tbody>
</table>

**MODELING**

**Modeling in AMESim**

Amesim is a commercial simulation software for the modeling and analysis of multi-domain systems. It is part of systems engineering domain and falls into the mechatronic engineering field. The software package is a suite of tools used to model, analyze and predict the performance of mechatronics systems. Models are described using nonlinear time-dependent analytical equations that represent the system's hydraulic, pneumatic, thermal, electric or mechanical behavior [9].

In this research, the simulation systems in AMESim are created depending on the scheme and control strategy. There are two systems, one of which is 'Initial Simulation System' (INSS), another of which is 'Iteration Simulation System' (ITSS). As can be seen from FIGURE 3 and FIGURE 4, hydraulic (blue) and mechanical (green) parts of INSS and ITSS are the same, and differences lie on the red parts, which mean the signal and control parts. The differences of them are listed in TABLE 2.

<table>
<thead>
<tr>
<th>Operation Times</th>
<th>INSS</th>
<th>ITSS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Control Signals</td>
<td>Input manually</td>
<td>'newvoltage.txt'</td>
</tr>
<tr>
<td>Data imported from files</td>
<td>/</td>
<td>'newvoltage.txt'</td>
</tr>
<tr>
<td>Data exported to files</td>
<td>'initialvelocity.data', 'initialvoltage.data', 'velocity.data' and 'voltage.data'</td>
<td>'velocity.data' and 'voltage.data'</td>
</tr>
</tbody>
</table>

INSS is shown in FIGURE 3. The blue and green parts are about hydraulic and mechanical components and have been explained before. The red parts represent control signals. Component 9 is a function about tensile property of specimens, whose input is deformation displacement and output is the tensile force. Component 10 is able to write data to files, so it's used to save data. INSS exports four groups of data: 'initialvelocity.data', 'initialvoltage.data', 'velocity.data' and 'voltage.data'. These data will later be imported to MATLAB. Component 11 is the initial control signal of the simulation system, which is set depending on calculations and experience.

ITSS is shown in FIGURE 4. Component 1 is able to achieve data from a certain file, which comes from MATLAB as a processing result and its name is 'newvoltage.txt'. Component 2 and 3 are used to save data to certain files, whose names are 'velocity.data' and 'voltage.data', waiting for further processing in MATLAB.

**FIGURE 3.** Initial Simulation System (INSS) in AMESim

1. Import from ‘newvoltage.txt’  2. Write to ‘voltage.data’  3. Write to ‘velocity.data’

**FIGURE 4.** Iteration Simulation System (ITSS) in AMESim
Neural Network in MATLAB

Use MATLAB to create the neural network. The neural network will be trained using two groups of data. Velocity is the input parameter and voltage is the target parameter. This is an input-output time-series problem, so it is suitable to use a time delay neural network [10].

What matters are the velocity and voltage data recorded from the beginning of simulation to the fracture time of specimen. Those data after fracture time are useless and may add difficulty to the training of neural network. Here is the GUI of creating a neural network in FIGURE 5. Open this GUI by inputting ‘ntstool’ in Command Window in MATLAB. It’s easy to get a neural network by just several steps of settings in ‘ntstool’.

![FIGURE 5. Neural Time Series (ntstool)](image)

Step 1: Choose ‘Nonlinear Input-Output’ as the problem type, as is shown in FIGURE 5. Click Next.

Step 2: Choose velocity as input parameter and voltage as target parameter. Set time step as ‘Matrix row’. Click Next.

Step 3: Default. Click Next.

Step 4: Set ‘Number of Hidden Neurons’ as 20. Click Next.

Step 5: Use Levenberg-Marquardt as the training algorithm. Click Train. After training, a neural network will be available.

At the last page of this GUI, a simple script is available. This script can create a neural network the same as what we create through ‘ntstool’.

Python

The loop computation needs to operate AMESim and MATLAB repeatedly, and if operate them manually, it won’t be a pleasant experience. Python is suitable for this kind of work [11]. All operations can be done automatically by python.

CONTROL STRATEGY

The control strategy is based on the offline learning of the neural network. By input real parameters into simulation systems and joint simulation of AMESim and MATLAB, neural network will have quantities of effective data for offline self-learning. After offline self-learning is finished, the neural network will be able to return an ideal control signal.

The first step of experiment is to test the tensile property of specimens, using displacement and force sensors. Stretch the specimen at a low rate, get the data of tensile property and import them to Component 9 in INSS and the same one in ITSS. What needs attention is that the mechanical properties of materials at high rates of strain are probably different from those at low rates [12], which may lead to some deviations to the results.
Next is to set the initial control signals. It depends on calculations and experience to set a relatively appropriate one. No need to be precise because neural network will revise it later.

Start simulation in INSS, and it will return results in no time. The velocity data will be saved to ‘initialvelocity.data’ and ‘velocity.data’. The voltage data will be saved to ‘initialvoltage.data’ and ‘voltage.data’.

Start data processing in MATLAB. Get data from ‘.data’ files. Use velocity as input parameter and voltage as output parameter to train the network. After training, a neural network function \( f(x) \) will be achieved. Set an ideal velocity array as the independent variable of the function, and get the new voltage array. The new one will be saved to ‘newvoltage.txt’ and waiting for imported to ITSS in AMESim.

\[
newvoltage = f(velocity)
\]  

Start simulation in ITSS. The control signal comes from file ‘newvoltage.txt’. After a new round of simulation is completed, new data of velocity and voltage will be exported into ‘velocity.data’ and ‘voltage.data’.

Then, the loop computation cycles between ITSS and MATLAB. FIGURE 6 is the flow chart of simulation. It can help understand the control strategy.

The final results need to be tested in a real tensile experiment to confirm its effectiveness. If the system simulates the real one well, the deviations would be little.

**RESULTS AND ANALYSIS**

The initial control signal will affect the iteration times and results of the experiment. I suggest a monotone increasing function, and the simplest type of it is a linear function.

As is shown in FIGURE 7, the specimen breaks at 17.9ms, after when the signal is set to zero to stop stretch. The data between 0 and 17.9ms are effective according to interpretations before.

Use MATLAB and ITSS to loop compute the control signals, and the requirement is that error should be less than 3%.

**FIGURE 7.** Case 1: A Possible Initial Control Signal and Corresponding Velocity Curve
FIGURE 8 shows the change of speed, and we can see that after 60 rounds of loop computation, stretch speed meets the requirement that error is smaller than 2%. The blue vertical line marks fracture time of a specimen. The speed reaches 20m/s at 13.9ms and the specimen breaks at 15.6ms. The speed ranges from 19.61m/s to 20.31m/s between 13.9ms and 15.6ms, which means the error is about 1.9%

It takes 40 rounds to reduce the error from 6.2% to 1.9% because there are several rounds where the errors increase (from 3% to 8%) instead of decreasing. The neural network will have a mutation to revise this mistake. This phenomenon also demonstrates that velocity comes near to 20m/s because of the self-adjustment of neural network functions rather than consistently increasing of the entirety of velocity curve after every round of simulation. In other words, the neural network works.

As is shown in FIGURE 9, the final signal has changed a lot compared with the initial one.

As is shown in FIGURE 10, use another initial control signal to test the effectivity of this method.
FIGURE 10. Case 2: Another Initial Control Signal and Corresponding Velocity Curve

As is shown in FIGURE 11, the speed reaches 20 m/s at 14.3 ms and the specimen breaks at 16.1 ms. Speed ranges from 19.65 m/s to 20.23 m/s between 14.3 ms and 16.1 ms, which means the error is 1.4%. It takes 32 rounds to get a suitable velocity curve.

FIGURE 11. Case 2: Final Control Signal and Corresponding Velocity Curve (Round 32)

When target velocity is set to 15 m/s, use the initial data in FIGURE 12 to train the neural network.

FIGURE 12. Case 3: Initial Control Signal and Corresponding Velocity Curve

The results are shown in FIGURE 13. The speed reaches 15 m/s at 16.1 ms and the specimen breaks at 18.5 ms. Speed ranges from 14.63 m/s to 15.26 m/s between 16.1 ms and 18.5 ms, which means the error is 2.5%. These three different simulations prove that when certain conditions are satisfied, this simulation system is effective to get an ideal control signal.
CONCLUSIONS

When target velocity is 20m/s, the initial control signal is a monotone increasing linear function and initial velocity is smaller than target velocity, just like Case 1 and Case 2, an ideal result with error smaller than 3% can be achieved. And this method also works when target velocity is 15m/s. In conclusion, it works in different situations. In real cases, parameters of real cylinders, valves and so on need to be recorded for simulation. What also need to be tested are the tensile properties of real specimens by using force and displacement sensors. The data of tensile properties should be imported to Component 9 of INSS in FIGURE 3 and the same component of ITSS in FIGURE 4. After the simulation is finished, it will return a group of voltage data, which will lead to an ideal real velocity curve of the stretch, up to 20m/s and approximately constant.

ACKNOWLEDGMENTS

Thank my university tutor, Mr. Shi, for giving me this research project and providing a general solving idea. I appreciate Engineer Hu for his favors in 3D modeling and other works. Finally, I need to thank the authors of books related to neural network, and they help me a lot in having a better understanding of neural network.

REFERENCES

DEVELOPMENT OF FLEXIBLE ELECTRO-HYDRAULIC CYLINDER FOR FLEXIBLE SPHERICAL ACTUATOR

Hiroaki TAMAKI*, Shujiro DOHTA*, Tetsuya AKAGI*, Wataru KOBAYASHI*, Yasuko MATSUI*

* Department of Intelligent Mechanical Engineering, Graduate School of Engineering, Okayama University of Science
1-1 Ridai-cho, Kita-ku, Okayama, 700-0005 Japan
(E-mail: t16rm03th@ous.jp, {dohta, akagi, kobayashi}@are.ous.ac.jp, t16sd03my@ous.jp)

Abstract. Inexpensive rehabilitation devices that can be used at home are required because of a lack of PT and welfare workers. In the previous study, the low-cost portable rehabilitation device using a flexible spherical actuator that consists of two flexible pneumatic cylinders was proposed and tested. However, compact and high power compressors have not been developed yet. In this study, a flexible electro-hydraulic cylinder driven by an electric motor and a hydraulic gear pump is proposed and tested. The construction and the operating principle of the proposed cylinder are described. The empirical equation of the suitable pump rotation for the desired displacement can be obtained. The position control of the tested cylinder is also carried out. As a result, by using the on/off control scheme based on the empirical equation, the multi position control within the tracking error of 6 mm can be realized.

Keywords: Flexible electro-hydraulic cylinder, Spherical actuator, Gear pump with encoder, Sequential control

INTRODUCTION

In an aging society, it is required to develop a system to aid in nursing care [1] and to support activities of daily life for the elderly and the disabled [2,3]. Rehabilitation devices help the elderly who is injured temporarily to recover their physical ability for keeping Quality of Life (QOL). Actuators used in such a system need to be flexible so as not to injure the human body [4]. The purpose of this study is to develop a portable rehabilitation device that can be safe enough to use it while users are handling it with human hands. In our previous study, a flexible pneumatic cylinder that can be used even if it is deformed by external forces has been proposed and tested [5]. We also developed a spherical actuator using the flexible pneumatic cylinders, which can be used on a table as a rehabilitation device for human wrist and arm [6-8]. A portable rehabilitation device using the flexible spherical actuator that consists of two flexible pneumatic cylinders was proposed and tested. The flexible spherical actuator can create large bending motion along the spherical surface. However, these pneumatic drive rehabilitation devices require an air compressor for continuous driving. In general, the compressor is heavy, bulky and noisy. Therefore, these pneumatic drive devices are unsuitable for using at home. In this paper, a flexible electro-hydraulic cylinder (we call it “FEHC” for short) driven by electric motors and a hydraulic gear pump is proposed and tested. The construction and the operating principle of the proposed actuator are described. The position control of the tested actuator is also carried out.

FLEXIBLE SPHERICAL ACTUATOR

Figure 1 shows the construction of a flexible pneumatic cylinder developed in our previous study [5]. The cylinder consists of a flexible tube as a cylinder and gasket, one steel ball as a cylinder head and a slide stage that can move along the outside of the cylinder tube. The steel ball in the tube is pinched by two pairs of brass rollers from both sides of the ball. The operating principle of the cylinder is as follows. When the supply pressure is applied to one side of the cylinder, the inner steel ball is pushed. At the same time, the steel ball pushes the brass rollers and then the slide stage moves toward opposite side of the pressurized while deforming the tube. The minimum driving pressure of the cylinder is about 120 kPa. The frictional force of the cylinder is about 10 N. The frictional force is larger than the case using the typical pneumatic cylinder that is less than 100 kPa. It is necessary to develop a novel flexible pneumatic cylinder with lower frictional force.
Figure 2 (a) shows an appearance of the previous spherical actuator using the cylinders mentioned above. The actuator was developed as a rehabilitation device for upper limbs. We imagine that patients hold both handling stages which are top and bottom stages in Fig. 2 (a) by their both hands while rehabilitation. Two slide stages of each flexible cylinder are not connected with one side base, that is, each slide stage of the flexible cylinder is fixed on each handling stage as shown in Fig. 2 (b). The size of the actuator is 260 mm in width and 270 mm in height. The total mass of the actuator is only 310 g.

Figure 3 shows the transient view of the movement of the actuator. In the experiment, a sequential on/off operation of the control valve every 0.8 seconds was done. The supply pressure is 450 kPa. From Fig. 3, it can be seen that both handling stages can change the position while the cylinders are bending and twisting. It can be confirmed that the actuator can create the different attitudes easily.

In order to drive the flexible pneumatic cylinders in the device, the compressor that can generate pressure of more than 400 kPa is required. In addition, if a small-sized compressor is used, the device might have a limitation of driving time. It is because the compressor generates heat while compressing the air. The heat prevents the continuous driving of the compressor. A flexible actuator with simpler driving system is required.
Therefore, a flexible cylinder with compact fluidic pressure source is proposed and tested. Figure 4 shows the construction of the flexible electro-hydraulic cylinder (FEHC). The FEHC consists of a gear pump (ASSIST Co. Ltd., PE1024N) with an electric motor and an encoder, and a flexible hydraulic cylinder as shown in Fig.4. The construction of the cylinder is almost same as the conventional flexible pneumatic cylinder. It consists of a flexible tube as a cylinder and gasket, one steel ball as a cylinder head, and a slide stage that can move along the outside of the cylinder tube. The slide stage has 12 steel balls which are set on the inner bore of the stage to press and deform the tube. The steel ball in the tube is held by two slide stages from both side of the ball. By using the improved slide stage, the minimum driving air pressure of the novel cylinder was reduced from 120 kPa to 94 kPa.

The operating principle of the cylinder is as follows. When the gear pump is driven toward clockwise, oil in A-side chamber is moved into B-side chamber through the pump in Fig.4. At the same time, the pressure difference of both side of the inner steel ball is generated in Fig.5. As a result, the ball is pushed and then it pushes the slide stage.

FIGURE 4. Flexible electro-hydraulic cylinder.

FIGURE 5. Construction of flexible electro-hydraulic cylinder.

POSITION CONTROL SYSTEM OF FEHC

In ideal, the flow rate of the pump is proportional to the displacement of the slide stage. Figure 6 shows the relationship between the rotational angle and discharging oil volume of the gear pump under the no-load condition. From Fig. 6, it can be seen that one rotational movement of the pump generates the volume change of 0.3 cm³. The volume change is equivalent to the displacement of about 6 mm of the slide stage. Therefore, by measuring of the rotational angle of the pump, the position of the slide stage can be controlled without displacement sensors.
In order to measure the rotational angle of the pump, an optical encoder is installed between the pump and the motor as shown in Fig. 7. The encoder consists of a slit disc with two slits and two transmission type photoelectric sensors (OMRON Co. Ltd., EE-SX493). Figure 8 shows the shape of the slit disc and each position of two photoelectric sensors in the encoder. By using the slit disc, A-phase and B-phase signals with 90 degrees phase difference are generated every 180 degrees rotation. Therefore, the resolution of the encoder using the disc is 45 degrees that is corresponding to the displacement of 0.75 mm.

![encoder_in_gear_pump](image1)

**FIGURE 7.** Encoder in the gear pump.

![slit_disk_encoder](image2)

**FIGURE 8.** Slit disk in encoder.

Figures 9 (a) and (b) show the schematic diagram and photograph of the position control system using the tested FEHC, respectively. The position control is done as follows. First, the microcomputer (Renesas Electronics Co. Ltd., SH7125) gets the A-phase and B-phase signals from the encoder through the I/O ports. In the microcomputer, binary data from these signals replace the value such as “+1” or “-1” in accordance with the algorithm as shown in Fig. 10. By counting the replaced value, the controller can measure the rotational angel of the pump. The deviation from the desired counting value, that is position (angle), based on the sequential data is
calculated. The motor is driven by the microcomputer through the motor driver based on the following simple on/off control scheme with dead zone of +3 or -3 counts as shown in Eq. (1) and Table 1.

\[ u_c = r_e - y_e, \]  

(1)

where \( u_c, r_e, \) and \( y_e \) mean the controlled input for motor, desired counting value corresponding to desired position and the measured counting value from the encoder, respectively. The counting value of 3 is corresponding to the displacement of about 2 mm under the no-load condition. In addition, the desired counting value for each displacement change is calculated to compensate the frictional force of the cylinder mentioned later. By this method, the displacement of the cylinder can be controlled while the encoder is measuring the rotational angle of the gear pump.

![Schematic diagram of the control system](a) Schematic diagram

![Photograph of the tested cylinder](b) Photograph

FIGURE 9. Control system of the tested cylinder.

![Algorithm of Up/Down counter in the controller](FIGURE 10. Algorithm of Up/Down counter in the controller.

<table>
<thead>
<tr>
<th>Pattern</th>
<th>Phase</th>
<th>Binary data</th>
<th>Step1</th>
<th>Step2</th>
<th>Step3</th>
<th>Difference from previous pitch</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>H</td>
<td>L</td>
<td>2</td>
<td>Replace</td>
<td>3</td>
<td>-3 Replace +1</td>
</tr>
<tr>
<td>II</td>
<td>H</td>
<td>H</td>
<td>3</td>
<td>Replace</td>
<td>2</td>
<td>+1</td>
</tr>
<tr>
<td>III</td>
<td>L</td>
<td>H</td>
<td>1</td>
<td>1</td>
<td>+1</td>
<td>-1</td>
</tr>
<tr>
<td>IV</td>
<td>L</td>
<td>L</td>
<td>0</td>
<td>0</td>
<td>+3 Replace -1</td>
<td></td>
</tr>
</tbody>
</table>

FIGURE 10. Algorithm of Up/Down counter in the controller.

TABLE 1. On/off control scheme with dead zone.

<table>
<thead>
<tr>
<th>Controlled input</th>
<th>Motor rotation</th>
</tr>
</thead>
<tbody>
<tr>
<td>( u_c &gt; 3 )</td>
<td>Clockwise</td>
</tr>
<tr>
<td>( u_c &lt; -3 )</td>
<td>Counter-clockwise</td>
</tr>
<tr>
<td>( -3 \leq u_c \leq 3 )</td>
<td>Stop</td>
</tr>
</tbody>
</table>

RESULT OF POSITION CONTROL OF FEHC

In order to get the desired counting value for each displacement of the slide stage in the tested cylinder, the relation between the rotational angle of the gear pump and the displacement of the slide stage as shown in Fig. 11 was investigated. In the experiment, we measured the displacement of the slide stage when the counting values of + 30, 50, 100, 200, and 300 were given as a desired counting value. In Fig. 11, symbols and the solid line show the experimental results and calculated displacement based on the discharging oil volume of the pump with no load, respectively. Figure 11 shows that the relation between the rotational angle and measured displacement of the cylinder is almost linear. On the other hand, the difference between the experimental and the calculated results can be observed. It is caused by the leakage at the gear pump and the frictional force of the cylinder.
slide stage. To compensate the leakage and frictional force, the following empirical equations for desired counting value \( r_c \) are applied.

\[
r_c = \begin{cases} 
1.63l - 0.07 & (l \leq 0) \\
1.65l + 2.10 & (l > 0) 
\end{cases}
\]

where \( l \) (mm) means the desired displacement. The equation when \( l \) is less than or equal to 0 mm in Eq. (2) was obtained by the result when the desired counting value was negative, that is the negative movement in Fig. 11. Similarly, the equation when \( l \) is greater than 0 mm in Eq. (2) was obtained by the result of the positive movement in Fig. 11. To compensate the difference of characteristics depending on the rotate direction of the pump, Eq. (2) is applied when the desired displacement \( l \) (mm) is negative and positive, respectively. These compensated displacements are shown by broken line in Fig. 11.

![Graph showing relationship between rotational angle and displacement](image)

**FIGURE 11.** Relationship between rotational angle of gear pump and displacement of slide stage.

Figures 12 (a) and (b) show the transient response of cylinder displacement for negative and positive movement, respectively. In Fig. 12, the broken and solid lines show the desired and controlled displacement of the cylinder, respectively. In the experiment, the displacement of the slide stage was measured by a wire type linear potentiometer [9]. The desired displacement change of 50, 100, 30 and 70 mm toward the negative and positive direction were given every 5 s. In the case of positive movement, the start point of the slide stage was set at about 10 mm from the left side cylinder end as shown in Fig. 9 (b). In opposite case, the slide stage was set at about 10 mm from the right side end. From Fig. 12, it can be seen that the position of the slide stage can trace the desired position. We confirmed that the proposed control method compensating the leakage and friction is useful to apply the sequential position control of FEHC.

![Transient response graphs](image)

**FIGURE 12.** Transient response of displacement of tested cylinder.

Figure 13 shows the view of the flexible spherical actuator using tested FEHCs. The total mass of the actuator with oil is 1.3 kg. This value is extremely light weight compared with the actuator using the whole pneumatic driving system that includes a compressor, regulator, valves and so on. In addition, the compliance of the actuator can be realized by the flexibility of the spherical actuator even if the stiffness for the longitudinal...
direction of the cylinder becomes large by using the hydraulic cylinder. As future work, we are going to carry out the attitude control of the tested spherical actuator using FEHCs.

![View of flexible spherical actuator using FEHC with controller.](image)

**FIGURE 13.** View of flexible spherical actuator using FEHC with controller.

**CONCLUSIONS**

This study aiming to develop a home rehabilitation device that it does not need a heavy and noisy compressor can be summarized as follows. The flexible electro-hydraulic cylinder that consists of the gear pump and the flexible hydraulic cylinders was proposed and tested. The relation between the rotational angle of the gear pump and the cylinder displacement was investigated. The empirical equation of the suitable pump rotation for the desired displacement can be obtained. The improved electro-hydraulic cylinder that the encoder installed into the pump was also proposed and tested. The position control of the slide stage was carried out. In the position control, in order to compensate the leakage of the pump and the frictional force of the cylinder, the simple on/off control scheme based on the empirical equation was applied. As a result, by using the proposed control scheme, the multi position control of the tested cylinder within the tracking error of 6 mm can be realized. As future work, we are going to construct the spherical actuator using the tested cylinder. The attitude control of the spherical actuator and the estimation of the actuator as a rehabilitation device will be carried out.

**ACKNOWLEDGMENTS**

This study was supported by the Education Ministry of Japan through a Financial Assistance Program for QOL Innovative Research (2012-2016) and a Grant-in-Aid for Scientific Research (C) (Subject No. 24560315 & 16K06202)

**REFERENCES**

HYDRAULIC RESONANCE CHARACTERISTICS OF THE HIGH-FREQUENCY EXCITATION SYSTEM CONTROLLED BY A 2D ROTARY VALVE

Yan REN *, Hesheng TANG*, Jian RUAN**

* Department of Mechanical and Electrical Engineering, Faculty of Engineering, Wenzhou University
Wenzhou University Town, Wenzhou, Zhejiang, 325035 China
(E-mail: rentingting211@163.com)

** Department of Mechanical Engineering, Faculty of Engineering, Zhejiang University of Technology
18 Chaowang Road, Hangzhou, Zhejiang, 310014 China

Abstract. A 2D rotary valve has been as an enabling technology used in electro-hydraulic excitation systems to greatly improve the excitation frequency. However, as the excitation frequency is increased to the hydraulic resonant frequency, pressure transient occurs in excitation systems. Therefore, the objective of this paper is to propose an analysis of hydraulic resonance characteristics for predicting the pressure surge and controlling this resonant peak. A number of simplified mathematical models are established and approximate analytical expressions about describing hydraulic resonance characteristics are solved. Finally, the experimental system is build to valid theoretical results. This research provides an access to exploit resonance energy at the special frequency point for realizing the decoupling of large output force and high excitation frequency.

Keywords: Hydraulic Resonance, Pressure Transient, High-frequency Excitation

INTRODUCTION

Pressure transients are commonly referred to pressure surges. These pressure peaks may become substantially higher than steady-state and cause damage to system. In general, pressure peaks are difficult to predict even in simple circuits. However, some common physical situations give predictable results. The first situation is usually called water hammer, of which the pressure peak is generated and controlled by keeping fluid velocities low. The second situation arises when a hydraulic actuator is suddenly stopped. There have been some researches about water hammer effect [1-2] and flow-induced pressure surge when a piston or motor is suddenly stopped [3-4], of which results have been applied in electro-hydraulic driven excitation system. In few papers only resonance effect in the hydraulic system is analyzed [5-6], and hardly ever, the nature of the hydraulic resonance. A high-frequency electro-hydraulic excitation system controlled by a 2D rotary valve is designed [7]. In experimental studies, some special phenomenon is found, where amplitude is increased significantly and the phase of output waveforms is varied suddenly in resonant range. It is necessary to explore the nature of hydraulic resonance phenomenon. Consequently this study is an attempt to address this issue.

ROTARY VALVE CONTROLLED HIGH-FREQUENCY EXCITATION

A high-frequency electro-hydraulic excitation system is completely analogous to the conventional combination of a three-way valve controlled the single rod actuator, as shown in Fig. 1. However, the characteristics of this system is that the natural frequency can be regulated as loads (it is also named the variable resonant system), and an improving 2D rotary valve is used in order to the excitation frequency be matched with the natural frequency of this system. The system can output large excitation force at high excitation frequencies, so it is especially applied to high-cycle fatigue testing of elastomeric materials, or crack growth testing, even high temperature condition.
FIGURE 1. Three-way 2D valve-piston combination

RESONANCE CHARACTERISTICS

Working Principle

Reference to Fig. 1, when the shoulder I of the spool is active, the hydraulic fluid flows into the head chamber. However, because acceleration of the piston is conspicuously big in resonance, the driving force is consumed mainly by mass load. Consequently the piston continues to move and compresses the fluid in the head chamber, as shown in Fig. 2 (a). As the pressure increases rapidly above the system pressure, the kinetic energy of the moving mass of fluid is instantly converted and momentarily stored as elastic potential energy until the piston comes to rest and moves in the reversed direction. At this time, the fluid travels back to the valve until the pressure is decreased below the system pressure, as shown in Fig. 2 (b). When the shoulder II of the spool is active, the pressure in the return chamber is decreased rapidly below the steady-state level and then causes the cavitate. The piston continues to travel back and forth with the associated interchanges of kinetic and potential energies. This oscillatory behavior will continue as the rotary motion of 2D valve spool even though the friction or leakage losses dissipate the energy involved.

FIGURE 2. Working principle in resonance assuming that the load mainly consists of inertia: (a) Valve ports on the shoulder I are active, the piston moves to the head chamber, the kinetic energy of the fluid is stored as the potential energy. (b) The piston continues to move to the head chamber, the pressure increased above the system pressure, which causes "special phenomenon", the fluid traveling back to the valve until the piston comes to a stop.
Over-all Natural Frequency

The pressure-flow equations could be expressed as a Taylor’s series about a particular operating point. However, nonlinear algebraic equations are not applicable to the high-frequency excitation system with fast-varying pressures. The transfer function for the linear region is

\[
y_p = \frac{E_i A_i}{s[\frac{V_o K_i + E_i A_i}{V_o m} + \frac{s}{s^2 + \frac{2\delta s}{\omega_0} + 1}] + 1} = \frac{K_i}{s^2 + \frac{2\delta s}{\omega_0} + 1}
\]

Where \(Y_p\) is the displacement of piston, \(Q_L\) is the flow through the load, \(E_h\) is effective bulk modulus of system, \(A_h\) is head side area of piston, \(V_0\) is initial head chamber volume, \(m\) is total mass of piston and load referred to piston, \(B_p\) is a viscous damping coefficient, \(K_L\) is load spring gradient, \(K_i\) is the flow gain, \(\omega_0\) is over-all natural frequency, \(\delta_0\) is damping ratio, dimensionless.

Referring to Eq. (1), the expression of the over-all natural frequency is

\[
\omega_0 = \sqrt{\frac{V_o K_i + E_i A_i}{V_o m}} = \sqrt{\omega_h^2 + \omega_m^2}
\]

Where \(\omega_h\) is the hydraulic undamped natural frequency, \(\omega_m\) is mechanical frequency.

Resonant Peak

Because the piston continues to travel back and forth with the associated interchanges of kinetic and potential energies, the equation is obtained as

\[
\frac{1}{2}mv_0^2 = \frac{1}{2}K_h A_{r_{\text{max}}}^2 + \frac{1}{2}K_L A_{r_{\text{max}}}^2
\]

Where \(v_0\) is initial velocity of piston, \(K_h = \frac{E_h A_i^2}{V_i}\) is defined as hydraulic fluid spring gradient of chamber, \(A_{r_{\text{max}}}\) is the over-travel of piston (the resonant peak). The excitation waveform in resonance is assumed as

\[
y_p(\theta) = A_{r_{\text{max}}} \sin(z\theta - \pi)
\]

Where \(y_p\) is the resonant waveform, \(\theta\) is the angular position of 2D valve spool, \(0<\theta<4\alpha\) in one cycle. Assuming zero damping, the force equation is rewritten as

\[
A_p p_L = m\omega_0^2 \frac{d^2 y_p}{d\theta^2} + K_L y_p
\]

Substituting Eq. (2) and (4) into Eq. (5), the resonant peak is obtained as

\[
A_{r_{\text{max}}} = \frac{A_h p_{L_{\text{max}}}}{K_h}
\]

Where \(p_{L_{\text{max}}}\) is load pressure peak value. Combining and solving for \(v_0\) gives

\[
v_0 = \sqrt{\frac{K_h + K_L A_p p_{L_{\text{max}}}}{m K_h}}
\]

The pressure drop across the load at resonant frequency is
\begin{equation}
p_z(\theta) = p_{z_{\text{max}}} \sin(z\theta)
\end{equation}

So the pressure in the control chamber \(p_{rc}\) is
\begin{equation}
p_{rc}(\theta) = p_{z_{\text{max}}} \sin(z\theta) + \frac{P}{2}
\end{equation}

Assuming all of hydraulic horsepower is used as piston work, the expression is
\begin{equation}
\int_0^{\theta_u} Q_L p_z d\theta = 0
\end{equation}

Where the expression of the load flow about orifice areas of rotary valve are given as [7]

Consequently
\begin{equation}
p_{z_{\text{max}}} = wp_z
\end{equation}

Where \(w\) is a pressure ratio.

### Simulations

The main parameters of the excitation structure and the hydraulic system are selected to be the same as its physical counterpart used in the experimental test. The numerical values of the parameters are listed in Table 1.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>(A_0) (m²)</td>
<td>(1.54 \times 10^{-2})</td>
</tr>
<tr>
<td>(C_d)</td>
<td>(0.62)</td>
</tr>
<tr>
<td>(D) (m)</td>
<td>(1.6 \times 10^{-2})</td>
</tr>
<tr>
<td>(E_h) (Pa)</td>
<td>(7.0 \times 10^8)</td>
</tr>
<tr>
<td>(K_L) (N/m)</td>
<td>(6 \times 10^7)</td>
</tr>
<tr>
<td>(L) (m)</td>
<td>(0.2)</td>
</tr>
<tr>
<td>(m) (kg)</td>
<td>(6)</td>
</tr>
<tr>
<td>(p_z) (Pa)</td>
<td>(0.7 \times 10^7)</td>
</tr>
<tr>
<td>(x_v) (m)</td>
<td>(4 \times 10^{-3})</td>
</tr>
<tr>
<td>(z)</td>
<td>16</td>
</tr>
<tr>
<td>(\rho) (kg/m³)</td>
<td>900</td>
</tr>
</tbody>
</table>

The Rung-Kutta method is used to solve Equations (8), (10) and (11). The resonant peak as the over-travel of piston can be solved by the displacement of the fluid spring. These general equations are used to derive the pressure-flow curve and orifice area of rotary valve-displacement of piston curve as shown in Fig. 3.
Remarks

1. The resonant peak depends on electro-hydraulic exciter system itself and is irrelevant with other parameters including the orifice areas or shapes of valve ports.
2. In resonance there is a region of the traveling back, which is the single important reason for the decoupling of large output force and high excitation frequency and it has no damage to the excitation system.

EXPERIMENTS AND RESULTS

Experimental Hardware

The prototype of high-frequency electro-hydraulic exciter is shown in Fig. 4.
Improving 2D Valve

The prototype of improving 2D rotary valve is shown in Fig. 5.

![Valve spool](image1.png) ![Valve sleeve](image2.png)

**Figure 5.** Prototype rotary valve hardware

The working frequency in this electro-hydraulic exciting type depends on the rotary speed of spool. However, if the speed exceeds a limited value the rotary motion of spool will be quite instability caused by spool whirl. So it is necessary to design hydrodynamic pressure lubrication for increasing critical speed of spool whirl as illustrated in Fig. 6. And the matching between groove on the spool and window on the sleeve is designed as shown in Fig. 7.

![Lubricating structure](image3.png) ![Matching between groove and window](image4.png)

**Figure 6.** Lubricating structure  **Figure 7.** Matching between groove and window

Experimental Results

The amplitude of vibration increases considerably and flow is minimum at resonant frequency. The behavior is graphically displayed in Fig. 8 (a), which is a plot of the amplitude response of the electro-hydraulic exciter versus 2D rotary valve rotaries at different speeds. As shown in Fig. 8 (b), variations in the excitation frequency especially in resonant frequency occur and cause considerable shifting in the flow response with different operating points.

![Characteristics of output force and flow](image5.png)

**Figure 8.** Characteristics of output force and flow

CONCLUSIONS

As the excitation frequency is extended to the resonant frequency, the excitation process varies widely. Being different from the mechanical resonance, the pressure transient peak of the hydraulic resonance or the over-travel of the piston approximates to a constant only depending on the system itself, and does not increase incessantly to destroy this system. In addition, there is 90° phase angle difference between orifice area of rotary...
valve and the displacement of piston. The exciter achieved large hydrostatic force output (10 kN) at high excitation frequency (900 Hz) for the initial length of the head chamber (10mm).

ACKNOWLEDGEMENTS

The authors are grateful to the support from the National Science Foundation of China (No. 51505338), the Zhejiang Provincial Natural Science Foundation of China (Nos. LQ16E050004, LQ17E050003).

REFERENCES

PERCEIVED STIMULI IN HYDRAULIC OPERATION LEVER OF CONSTRUCTION MACHINERY

Hironao YAMADA*, Fumichika OKADA**, Katsutoshi OTSUBO*, Takuya KAWAMURA*

* Department of Mechanical Engineering, Faculty of Engineering
Gifu University
1-1 Yanagido, Gifu 501-1193, Japan
(E-mail: yamada@gifu-u.ac.jp)
** Toyota Motor Corporation
1 Toyota-Cho, Toyota City, Aichi Prefecture 471-8571, Japan

Abstract. In construction machinery controlled by an operation lever with a hydraulic pilot valve, it is said that operators manipulate the machine based on subtle operational reaction forces in the operating lever. In this study, we attempt to calculate the reaction force in a hydraulic operating lever caused by machine operation, and to clarify the characteristics of the feeling experienced by the operator. We found that a minute reaction force was generated on the hydraulic operating lever, which the operator could feel. In addition, it is thought that at the actual work site, the operator uses and integrates information in addition to haptic information. The cross-modal effect of auditory and force-sense stimuli was verified through experiments. It was confirmed that the haptic stimulus was amplified by the auditory stimulus, the operating reaction force sensed by the operator was increased, and the response time to the force sense presentation was shortened.

Keywords: Construction machine, Hydraulic Control Lever, Simulation, cross-modal effect

INTRODUCTION

Professional operators of construction machines are said to control the machines by sensing the subtle reaction force against the hydraulic pilot valve-type operation lever. In other words, information regarding speed of the machine and change of driving force is assessed from the reaction force in the operation lever, and used for machine operation. Here, the reaction force of the hydraulic operating lever is the force generated in the operating lever due to dynamic states inside the machine, such as piston displacement, acceleration, and cylinder pressure of the actuators in the boom, arm, and swing. It has not yet been clarified, however, whether the operator can sense the generation principle and the generated force.

In this study, we sought to calculate the reaction force to the hydraulic operating lever caused by machine operation, and to clarify the characteristics of the feeling experienced by the operator against the operation reaction force. We used an experimental apparatus (test bench) to simulate the actual machine, and analyzed operation reaction force based on the dynamic model at the moment the port of the pilot valve opens and the hydraulic fluid begins to flow.

It can be inferred that the feeling obtained when operating a construction machine is perceived not only from the force-sense information transmitted through the operation lever, but also from various forms of information in the work environment, such as auditory and visual stimuli. It is known that human perception is integrated by the interaction of information obtained through the five senses, represented by vision and hearing. The phenomenon of integrating two or more types of sensory stimuli is referred to as the "cross-modal phenomenon" or "cross-modal effect". For example, with the McGurk effect [1] a visual stimulus affects an auditory stimulus. Cross-modal phenomena have been confirmed among various types of perceptual stimuli. Narumi et al. [2] demonstrated cross-modal effects in visual and taste stimuli. They developed a taste display that "changed" taste in the user’s perception by superimposing colors on the beverage with an LED light source. In another study by Yokoyama [3], a cross-modal effect between visual and tactile stimuli was confirmed by integrating haptic feedback into a touch panel. They developed a system for improving flick input performance. If the cross-modal phenomenon affects information processing based on human perception, we think that it corresponds to the operation of construction machines.

In this study, we focused on auditory stimulation and conducted an experiment to investigate the interaction with the force sense of the operation lever. That is, in order to quantify the sense of the operation obtained by
the operator handling the lever of an excavator, a sensory evaluation was performed for the cross-modal effect of the force-sense and auditory stimulus assumed to occur in an actual working environment.

**NOMENCLATURE**

- $A$: Opening area of the valve port [m$^2$]
- $c_d$: Damping coefficient [N $\cdot$ s/m]
- $c_p$: Velocity of pressure wave [m/s]
- $F$: Reaction force acting on the pusher [N]
- $g$: Gravitational acceleration [m/s$^2$]
- $k_1$: Spring constant of return spring [N/m]
- $k_2$: Spring constant of balance spring [N/m]
- $K$: Bulk modulus [Pa]
- $l_{s1}$: Natural length of return spring [m]
- $l_{s2}$: Natural length of balance spring [m]
- $l_{s1}$: Set length of return spring [m]
- $l_{s2}$: Balance spring set length [m]
- $l$: Distance from the fulcrum of the lever to the action point of the pusher [m]
- $L$: Length of lever [m]
- $m$: Mass of pusher [kg]
- $M_S$: Spool mass [kg]
- $M_L$: Mass of lever [kg]
- $p$: Surge pressure [Pa]
- $P$: Output port pressure [Pa]
- $Q$: Volume flow rate of hydraulic fluid [m$^3$/s]
- $S$: Pressure receiving area of output port [m$^2$]
- $T$: Reaction force applied to the point of application of operating force [N]
- $v$: Initial speed of hydraulic fluid [m/s]
- $v_e$: Terminal speed of hydraulic fluid [m/s]
- $x_1$: Pusher stroke [m]
- $x_2$: Spool displacement [m]
- $x$: Distance from the fulcrum of the lever to the application point of the operating force [m]
- $\rho$: Density of hydraulic oil [kg/m$^3$]
- $\theta$: Operation angle of the lever [$^\circ$]
- $\phi$: The angle between cam and pusher [$^\circ$]

**EXPERIMENTAL SETUP**

Fig. 1 contains a schematic diagram and a photograph of the test bench. The elements that make up the test bench include a pump, relief valve, lever unit (photo on the right), throttle valve, and pressure sensor.
The internal structure of the one-pilot valve constituting the lever unit is shown in Fig. 2. The cam is tilted by operation of the lever and rotates accordingly, and the pusher converts it into a vertical motion. The stroke of the pusher is proportional to the operation of the lever. The pilot port is opened or closed by the displacement of the spool accordingly. The balance spring adjusts the degree of opening of the port and determines the output pressure.

In the actual test bench, the items shown in Fig. 2 were arranged at the front, back, left and right in the lever operation direction. By pushing the lever connected to each cam in one of the four directions, the corresponding pusher was depressed.

**MATHEMATICAL MODEL OF HYDRAULIC CONTROL LEVER**

In order to perform a dynamic analysis of the reaction force, the operation of the pilot valve was simplified by the spring mass damper model. The spring mass damper model used is shown in Fig. 3. The figure shows a model created based on the internal structure of Fig. 2. Equation (1) defines the equation of motion of the pilot valve in the neutral state.
\[ m\ddot{x}_1 = F + mg - k_1(l_{n1} - l_{s1} + x_1) - k_2(l_{n2} - l_{s2}) \]  
Equations (2) and (3) show the equations of motion of the pilot valve in the reduced-pressure state.

\[ m\ddot{x}_1 = F + mg - k_1(l_{n1} - l_{s1} + x_1) - k_2(l_{n2} - l_{s2} + x_1 - x_2) \]  
\[ M_s\ddot{x}_2 = M_s g + k_2(l_{n2} - l_{s2} + x_1 - x_2) - PS - cx_2 \]

At the moment the state changes from neutral to the reduced-pressure state, the port opens in the pilot valve, and the high-pressure hydraulic fluid discharged from the pump flows into the spool. The surge pressure \( p \) at this time can be derived by equation (4).

\[ p = \rho c_p (v - v_e) \]  

Here, \( \rho \) is the density of the hydraulic fluid, \( v \) is the initial speed of the working fluid \((v = 0 \text{ [m/s]})\), \( v_e \) is the terminal speed of the hydraulic fluid, and \( c_p \) is the speed of the pressure wave.

\[ c_p = \sqrt{K/\rho} \]  
The terminal speed \( v_e \) of the hydraulic fluid is expressed by equation (6).

\[ v_e = \frac{0}{A} \]  

From equations (4), (5) and (6), the surge pressure \( p \) is given by equation (7).

\[ p = -\frac{0}{A} \sqrt{\rho K} \]  

Fig. 4 shows the state when a reaction force acting on the pusher acts on the lever.

The reaction force \( T \) applied to the point of application of the operating force is given by Equation (8) from the moment balance.

\[ T = \frac{IF \cos(\varphi - \theta) - \frac{1}{2} LM_1 g \sin \theta}{x \cos \theta} \]
EVALUATION OF OPERATION REACTION FORCE

In this experiment, the reaction force at the moment when the valve port opened was measured with a six-axis force sensor. The pressure of the hydraulic pump was set to 4.0 MPa, and the operating speed of the lever was set to two patterns of 10 °/s and 100 °/s. Measurements were made ten times for each pattern. A schematic diagram of the experimental setup, including the operation lever, is shown in Fig. 5.

In order to keep the lever at a constant operating speed (10 °/s, 100 °/s), it was run while monitoring the measured angular velocity in real time with a motion recorder.

Fig. 6 shows the results for an operating speed of 10 °/s, and Fig. 7 for an operating speed of 100 °/s. The vertical axis represents the operation reaction force [N], and the horizontal axis represents the pusher stroke [mm]. The sampling period of data acquisition of the force sensor was 50 Hz, and the average value for 10 measurements was calculated. In both figures, when the displacement of the pusher stroke reached 0.9 mm, the valve port opened, the hydraulic fluid flowed into the pressure chamber, and the operating reaction force tended to increase. However, when operating the lever at 100 °/s, it was confirmed that the operating reaction force increased by about 56% compared to the operation at 10 °/s. This was thought to be because when operating speed is high, pressure-rising speed in the pressure chamber increases, resulting in a large operating reactive force. In addition, it can be seen that when the operating speed was 100 °/s, the operating-reaction force increased or decreased in a shorter period than at 10 °/s. This was considered to be due to the relationship between the balance spring and the pressure of the hydraulic oil applied to the spool from the operation of the pilot valve.

These results confirmed that the reaction force tends to increase according to how much the operation lever is pushed, which agrees with the trend of the analysis results. However, both values are larger than the analysis result. This was thought to be due to external forces other than the reaction force created when an operator manipulated the pilot valve itself (that is, a human hand applied operation force to the lever). Therefore, review of the experimental method is expected to produce improvements.

Next, sensory evaluation experiments were conducted to quantify the sensation experienced by the operator when working the lever. The magnitude-estimation method [4] (hereafter, ME method) is a one-dimensional scale construction method in which subjects estimate intensities by comparing a stimulus to a reference stimulus in the form of numerical values. Ten subjects evaluated the operation reaction force when the valve port opened in the two patterns with an operating lever speed of 10 °/s and 100 °/s.
FIGURE 8. Results of sensory-evaluation experiments

In Fig. 8, the vertical axis represents the perceived force of the subject, and the horizontal axis represents the relative value of the physical quantity, both of which are dimensionless quantities. The perceived force is the subjective rating provided by subjects. The physical quantity is an analysis value of the operation reaction force corresponding to the movement of the lever. The value of the reaction force at the minimum hydraulic pressure among the six specified, set pressures was set as a reference value (10). Subjects were asked to numerically rate the reaction force against the reference value. Each measured value (perceived force) was obtained by calculating the average value and aligning the standards. The physical quantity was obtained by calculating the ratio to the theoretical value at each measurement point. Here, the operating reaction force at the point where the pusher stroke is the minimum was used as the reference for the theoretical value. All the power approximation was calculated, and it can be seen that the approximation function is linear in the graph of the two logarithms. As can be seen in the figure, the faster the operation speed, the larger the inclination, and the larger the operation reaction force felt. This is because pressure rise per unit of time is larger with faster operation speeds.

The results of this experiment confirmed that a large reaction force is felt when the operation speed is high, and when operation speed is slow, there is a tendency for minute operation reaction forces to be easily distinguishable. We also confirmed that the operator could operate the excavator according to feelings of the reaction force corresponding to the size and speed of the operation.

CROSS-MODAL EFFECT OF HAPTIC AND AUDITORY STIMULI

In the field, the operator is considered to use information comprehensively, not haptic information alone. We therefore investigated the cross-modal effect of auditory and force-sense stimuli through an operation-lever experiment. In this experiment, the joystick shown in Fig. 9 was used to present an operation reaction force of an arbitrary size.
In the experiment, subjects operated the joystick while blindfolded and wearing headphones. Subjects were asked to press the trigger button of the joystick at the moment they felt a force on the lever, and then to evaluate the force numerically. When the operation angle of the joystick reached the threshold value (4.0°), the force sense presentation simulating the pressure rise at the port opening was initiated. The magnitude of the force presented to the subject could be set at -10000 to 10000 (dimensionless quantity) by lever-control software. In this experiment, five haptic sensations of 1000, 2000, 3000, 5000 and 10000 were presented in the direction opposite to the operation.

During the experiment, the sound of an excavator driving was presented to the subject at about 50 dB via the headphones to simulate the environmental sounds of an actual work site. The auditory stimulus was set so that driving noise became higher (frequency increased) simultaneously with force-sense presentation. Subjects were instructed to press the trigger button of the lever (Fig. 9 right) the moment they felt the haptic stimulus. As a result, time elapsed from the time of force presentation was acquired, and reaction time to force sense presentation was measured. In all 11 patterns combined with the case without force sense presentation, subjects were asked to rate the haptic value.

In Fig. 10, the vertical axis represents the perceived force of the subject and the horizontal axis is the relative value of the physical quantity, both of which are dimensionless quantities. The results show that both the inclination and the intercept were larger with sound changes. This suggests that greater manipulation reaction force is felt by presenting the sound change while driving the excavator.

In Fig. 11, the vertical axis represents the reaction time[s], and the horizontal axis represents the magnitude of the dimensionless quantity of the reaction force presented; it shows the reaction time with respect to the magnitude of the presented force depending on the presence / absence of the sound change. Values are the
average and standard deviation for the presence / absence of each sound change presentation and magnitude of presented force. The figure shows that reaction time was shorter overall with sound change than without. In addition, when sound change was presented, there was not much difference in reaction time. On the other hand, when sound change was not presented, response time became shorter as the presenting force increased. This demonstrates that subjects could respond more quickly to minute changes in force when changes in sound were also presented. We found that haptic stimulation was amplified by auditory stimulation. A t test was conducted on the results, and a significant difference was confirmed at a 5% risk ratio depending on whether or not a sound change was presented when the magnitude of the presented force was set to 2000. This was because, when a sound change was presented in the 1000-magnitude-of-force condition, it is thought that the force was minute, producing a large variation in reaction time regardless of presence or absence of the sound stimulus. In contrast, when the presentation force was 3000 or more, it is considered that no significant difference was observed because reaction time was short even when no sound change was presented.

CONCLUSIONS

In this study, we analyzed the reaction force generated in a hydraulic control lever by the operation of a piece of heavy construction machinery, and quantitatively evaluated the feeling of the reaction force. First, the operational reaction force based on the limited dynamics model was analyzed using a test bench. We found that a minute reaction force was generated on the hydraulic operating lever, which operators were able to feel. Moreover, since it is considered that at actual work sites, operators use information comprehensively, not haptic information alone, the cross-modal effect of auditory and force-sense stimuli was verified by experiments. Results confirmed that the haptic stimulus was amplified by the auditory stimulus, rated value of the operating reaction force increased, and reaction time to the force sense presentation was shortened.

Professional operators of construction machinery insist that they perform precise operations based on sensing the subtle reaction force from the operation lever; in reality, however, they react not to the reaction force from the control lever alone, but to the cross-modal effect, which suggests that a pseudo reaction force may be obtained. It was suggested that a pseudo operation reaction force could be simulated by presenting appropriate sound, vibration information, etc., to the operator according to the motion of the machinery. In other words, we demonstrated the possibility of developing a new control lever interface to improve operational sensing.

In this study, we confirmed the cross-modal effect of auditory and force-sense stimuli at lever operation of a piece of construction machinery. However, actual work sites contain various other factors such as visual and vibration stimuli, which were not considered in this research. In future research, it will be necessary to verify how these factors affect haptic information during lever operation.

ACKNOWLEDGMENTS

This study was conducted as cooperative research with Hitachi Construction Machinery. The authors would like to acknowledge the considerable assistance and helpful advice of Mr. Hiroshi OGURA.

REFERENCES

A NOVEL INTEGRATED LOAD-SENSING ELECTRO-HYDRAULIC ACTUATOR FOR AIRCRAFT STRUCTURAL TESTS

Yaoxing Shang, Xiaochao Liu, Zongxia Jiao, Jiakang Wu, Liang Yan

School of Automation Science and Electrical Engineering, Beihang University, Beijing 100191, China
(E-mail: liuxiaochaoustb@163.com)

Abstract. The traditional loading system, for the full-sized aircraft structure test, requires centralized hydraulic power supply and distributed valve-controlled cylinder, which leads to complex and large pipeline systems. Moreover, the loading system efficiency is quiet low because of both the huge overflow loss and the huge throttling loss. In order to conduct the structure test more efficiently, this paper proposes a novel integrated Load Sensing Valve-Controlled Actuator (LSVCA) with high efficiency and low energy consumption, which can reduce overflow loss by the intermittent operation of motor and reduce throttling loss by the variation of supply pressure. It also simplifies the test platform reconfiguration due to its high level integration and Power-By-Wire feature. In order to verify the feasibility of the LSVCA, the mathematical model is established and the simulation is conducted. The results indicate that the LSVCA is adequate to satisfy the criteria for full-sized aircraft structure test.

Keywords: Full-sized aircraft structure test; Load sensing valve-controlled actuator; Low energy consumption;

INTRODUCTION

Full-sized aircraft structure tests [1], including static load tests and fatigue tests are indispensable in the process of aircraft development and certification. The intention of the static load tests is to test the structure strength, stiffness, stability and other characteristics under the different static loads. The fatigue tests [2] are conducted to guarantee the safety and the efficient operation during the service life. The aircraft structure must be inspected for some potential cracks [3] before it can be qualified.

As we know, the traditional loading system for the aircraft structure test is the typical force control system, which features the low loading speed and the small loading flow. Currently, the multi-channel loading system [4] powered by centralized hydraulic power is widely used in aircraft structure tests. However, it is not only inefficient but also very complicated in construction and installation due to the lack of flexibility. With the increase in aircraft size and wing loads, the structure test is facing much more serious questions and demands, especially in the two following aspects.

The first one is the need to reduce workload and improve work efficiency. The increasing aircraft size and the more complicated loading test system bring much burden to energy sources, actuators and the measurement of the control system [5]. According to data from the Boeing Company, the Boeing 777 aircraft wingspan 60.93m, its maximum test load nearly 1000 tons and its maximum wing tip deformation up to 6m. By controlling the 96 hydraulic loading channels to work, the aircraft body is pasted with 4300 strain gauges. Reference [6] mentions the static test needs to simulate different flying states, so the loading test bench layout is often changed, which usually takes dozens of people and nearly two weeks. Given this, it is a very important requirement for loading test systems to have enough flexibility. Therefore, the distributed integration of the loading system becomes the mainstream of development.

The second one is the need to improve energy efficiency [7]. According to data from Airbus Company, the loading test of a large civil aircraft, whose installed power of hydraulic system is 350kW. Its fatigue tests take about 30,000 hours in almost six years and the entire loading test cycle consumes 10 million degrees. If one third energy, around 3 million degrees, is saved, it is equivalent to the annual power of a small hydropower station. The energy saving potential is tremendous.

Due to the fact that the hydraulic power is transferred by the complex and heavy hydraulic pipelines, the limitations of traditional loading test systems are not helpful to improve the work efficiency and the energy efficiency [8]. In order to overcome the drawbacks of using hydraulics, a technology named Power-By-Wire (PBW) was proposed, in which electric wires are used to transfer energy and some specific PBW actuators take the place of traditional hydraulic actuators [9]. The concept of PBW develops from flight control system, which avoids complex pipelines. The system reliability and maintainability are greatly improved. Reference [10] introduces the commonly actuators by used PBW, which are the Integrated Actuator Package (IAP) and the Electro-Hydrostatic Actuator (EHA) adapt the volume control method to avoid the throttling loss of valve control system, so that the efficiency of the PBW actuator is greatly improved. Also, it is easy for energy
management and energy saving effect is obvious with multiple channels working off-peak. However, it still has same problems as the traditional volume control, such as large quality of the adjusting mechanism, complexity of the speed control and low response of the actuating system.

Reference [1] proposes the motor-pump-valve coordinated control actuator, which gives consideration to both the system response and system throttling loss efficiency. However, we can learn that in the system there are three adjustable parameters, which are the motor speed, the pump swashplate and the valve opening. Moreover, the multiplication between the motor speed and the pump displacement is a typical nonlinearity. It means this solution needs to control motor, pump and valve at the same time, which leads to complex control algorithm and huge actuator volume.

Therefore, this paper puts forward a novel integrated Load Sensing Valve-Controlled Actuator with constant speed motor and constant displacement pump. It improves the system efficiency by reducing the overflow loss and the throttling loss. At the same time, due to its high level integration and PBW feature, the workload of changing test configuration is greatly reduced.

WORKING PRINCIPLE OF THE LSVCA

The key point of the load sensing system is the load sensitive device, which gives the feedback of the load force changing to the adjusting mechanism of the system. Then, the output power of the system will be adjusted to match the load power, so as to reduce the system power loss and achieve great energy saving [12]. This paper puts forward a novel integrated Load Sensing Valve-Controlled Actuator with constant speed motor and constant displacement pump, so the load sensing control is hard to be achieved neither by controlling motor speed nor by changing pump displacement. Fig. 1 is the hydraulic schematic diagram of the new proposed LSVCA. In this paper, the load sensitive devices of the LSVCA are the force sensor 10 and the pressure sensor 8. The force sensor 10 is used to sense the variation of the load force and the pressure sensor 8 is applied to help the controller monitor the changing of the supply pressure.

FIGURE 1. Hydraulic schematic diagram of the LSVCA

For a constant pressure system, it is quite inefficient with a large variation in load pressure, especially when the load pressure is much lower than the supply pressure. In contrast, for a load sensing system, the supply pressure can be adjusted to match the load pressure, so as to make sure the supply pressure is not excessive but enough to meet the workload requirements. This is the reason why a load sensing system can reduce the throttling loss and achieve better energy-saving effect. In this paper, the adjusting mechanisms of the LSVCA are the proportional relief valve 5 and the motor 1. The supply pressure is automatically regulated with the workload requirements by the proportional relief valve 5, so as to reduce the system throttling loss. The accumulator 6 works as an auxiliary energy source to provide system flow for the load. The motor 1 is working in an intermittent operation mode. It means if the supply pressure is lower than the pre-set value, the motor will start up to fill the accumulator and the motor will stop when the accumulator is full. Thus the overflow loss is effectively reduced because of the intermittent operation mode of the motor.

This LSVCA consists of three independent control loops, which are the motor control loop, the relief valve control loop and the actuator control loop, as shown in Fig. 2. The motor control loop controls the working condition of the motor by detecting the pressure of the accumulator, so as to avoid huge overflow loss. The relief valve control loop regulates the supply pressure by detecting the load force, so as to avoid huge throttling loss. Finally, the actuator control loop completes the static force loading test by the feedback of the load force.
Compared with traditional actuator, the proposed LSVCA makes the labor of the test platform reconfiguration easier because of its high level integration and Power-By-Wire feature. It also can reduce the overflow loss by the intermittent operation of the motor and reduce the throttling loss by the variation of the supply pressure. Compared with the EHA, the proposed LSVCA is a typical valve control system, so the reciprocating motion of the hydraulic cylinder is controlled by the servo-valve not the bidirectional pump or the bidirectional motor. System response and system bandwidth are improved because the large inertia moment of the bidirectional pump or the bidirectional motor is avoided. Compared with the motor-pump-valve coordinated control actuator, the proposed LSVCA is a more low-cost solution by using the constant speed motor and constant displacement pump. More importantly, there is no need to control motor, pump and valve at the same time, which greatly simplifies the control law. Thus the proposed LSVCA is simple, low-cost, high-efficiency, highly-integrated and pretty appropriate for the full-sized aircraft structure loading test system.

THE EFFICIENCY ANALYSIS OF THE LSVCA

Compared to the volume control system, the valve control system is quick in dynamic response and precise in control. However, the main limitation is the lower efficiency caused by throttling loss and overflow loss. The load sensing technology makes the system flow and supply pressure can change with the load variation so as to enhance the system efficiency. In order to improve the efficiency of valve control system, the relationship among flow, pressure and efficiency is thoroughly investigated in this section.

Efficiency Analysis of the Valve Control System

Two kinds of extra energy consumption are responsible for the inefficiency of hydraulic system, including the flow loss $\Delta Q$ through the outlet relief valve of pump and the pressure drop $\Delta P$ through the servo-valve. The incompatibility between the output power and the load power is the cause of the extra energy consumption. In most of the cases, the hydraulic system load power is constantly changing. If the output power cannot change with the load power constantly, the power loss will happen. The power loss of the hydraulic system is

$$\Delta N = N_S - N_L$$

Where, $\Delta N$ is the power loss, $N_S$ is the output power, $N_L$ is the load power.

Substitute $N_S = P_S Q_S$ and $N_L = (P_S - \Delta P)(Q_S - \Delta Q)$ into Eq. (1), then

$$\Delta N = P_S \Delta Q + Q_S \Delta P - \Delta P \Delta Q$$

According to Eq. (2), when there is only pressure drop, which means $\Delta Q = 0$, the total power loss of the hydraulic system is

$$\Delta N = Q_S \Delta P$$  \hspace{1cm} (3)$$

In the other case, if there is only flow loss, which means $\Delta P = 0$, the total power loss of the hydraulic system is

$$\Delta N = P_S \Delta Q$$  \hspace{1cm} (4)$$

In order to minimize the total power loss, either the reduction of pressure drop or flow loss can be helpful. Hydraulic system flow is related to the existence of pressure drop and fluid resistance. When the pressure drop decreases to zero, the flow will reduce to zero as well. This means the pressure drop is inevitable if there is an output flow of the hydraulic system. Therefore, it is impossible to completely eliminate the energy loss in hydraulic system. However, by analyzing the relationship between the servo-valve efficiency and the pressure drop, proper control means can be applied to reduce the energy loss.
The ideal four-way spool valve with zero laps is shown in Fig. 3. The load flow and load pressure must be satisfied with the classic orifice equation.

\[ Q_L = C_0 \omega x_v \sqrt{\frac{1}{\rho} (P_S - P_L)} \]  

Where, \( C \) is the flow coefficient, \( \omega \) is the area grads, \( x_v \) is the displacement, \( \rho \) is the density of the hydraulic oil, \( P_L \) is the load pressure, \( Q_L \) is the flow of the load. Thus, the load power of four-way spool valve is given by

\[ N_L = P_L Q_L = P_L C_0 \omega x_v \sqrt{\frac{1}{\rho} (P_S - P_L)} \]  

Take the derivative of Eq. (6), which means \( dN_L/dP_L = 0 \), the maximum load power point can be get when \( P_L = 2/3 P_S \).

If a constant pressure variable displacement pump is used, the system output flow will be automatically adjusted to meet the load flow, which means \( Q_S = Q_L \). Thus, the system efficiency is given by

\[ \eta = \frac{P_L Q_L}{P_S Q_S} = \frac{P_L}{P_S} \]  

This means the system efficiency only depends on the ratio of load pressure to supply pressure. The maximum efficiency of spool valve in the constant pressure variable displacement pump system is \( 2/3 \approx 0.667 \) under the condition of \( P_L = 2/3 P_S \).

If a constant displacement pump is used, the system output flow should be equal or greater than the maximum no-load flow of the spool valve, which means \( Q_S \geq C_0 \omega x_{v_{\text{max}}} \sqrt{\frac{1}{\rho} P_S} \). Thus, under the condition of \( P_L = 2/3 P_S \), the system maximum efficiency is given by

\[ \eta_{\text{max}} = \frac{P_L Q_L}{P_S Q_S} = \frac{P_L C_0 \omega x_{v_{\text{max}}} \sqrt{\frac{1}{\rho} (P_S - P_L)}}{P_S C_0 \omega x_{v_{\text{max}}} \sqrt{\frac{1}{\rho} P_S}} = \frac{2\sqrt{3}}{9} \approx 38.5\% \]  

Where, \( x_{v_{\text{max}}} \) is the maximum opening of the spool valve. And a set of curves depicting the relationship between the system efficiency and the spool displacement are shown in Fig. 4. It shows the system efficiency is zero when \( P_L = 0 \) or \( P_L = P_S \). The system efficiency reaches its maximum value 38.5% under the maximum opening of the spool valve when \( P_L = 2/3 P_S \).
It is true for both variable displacement pump system and the constant displacement pump system that system efficiency is highest when \( P_L = \frac{2}{3}P_S \). By regulating either the load pressure or the supply pressure, the efficiency of hydraulic system can be enhanced. In terms of the aircraft structure loading system, the load pressure is predetermined by the load spectrum, which cannot be changed casually. The only way to improve system efficiency is to change the supply pressure with the load variation in real time based on the load sensing technology.

**Pressure Regulation Method of the LSVCA**

For the traditional valve-controlled loading test system, one of the reasons leading to low efficiency is the constant supply pressure, which could cause huge throttling loss. In this paper, one innovative idea is that the supply pressure can be adjusted with the load pressure in real time, so as to reduce the throttling loss. It is certainly the most efficient pressure regulation method is keeping the relationship of \( P_L = \frac{2}{3}P_S \) during the whole working process, as shown in Fig. 5, which is called real-time sensitive regulation method in this paper. This method could reduce the throttling loss and enhance the system efficiency to the greatest extent.

However, this real-time sensitive regulation method requires extremely high accuracy for the control of the supply pressure, which is difficult to be achieved for both the constant pressure variable displacement pump system and the constant displacement pump system. In addition, another disadvantage of this method is the pulsation of valve inlet pressure, which may render the poor performance of valve control system. More importantly, this real-time sensitive regulation method is not applicable for the LSVCA in this paper, because it not only needs the motor to work continuously, but also needs the relief valve to regulate the supply pressure continuously, which will result in a huge overflow loss.

**MODELLING AND SIMULATION OF THE LSVCA**

In last section, the efficiency of the valve control system is analyzed and the fact which the efficiency of hydraulic system can be enhanced by adjusting the load pressure \( P_L = \frac{2}{3}P_S \) is indicated. In addition, the step regulation method is used to design the LSVCA in this paper. In this section, some models about the LSVCA are built and some simulation results are achieved.

To illustrate the effectiveness of the LSVCA, a virtual simulation model has already been established in the MATLAB/Simulink condition. The LSVCA model is a typical hydraulic system, which includes the constant speed motor model, pump model, the accumulator model, the relief valve model, the valve-controlled hydraulic cylinder model and other accessories. In addition, to get a more accurate nonlinear model for virtual validation, some typical nonlinear effects have been taken into account for the virtual model. The friction and leakage in the LSVCA has also been taken into account. The diagram of the LSVCA model is illustrated in Fig. 6. The simulation parameters of LSVCA are shown as Table 1.
### FIGURE 6. Block diagram of the LSVCA model

### TABLE 1. Simulation parameters of LSVCA model

<table>
<thead>
<tr>
<th>Component</th>
<th>Parameter</th>
<th>Symbol</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Servo-valve</td>
<td>time constant</td>
<td>$\tau_v$</td>
<td>0.001</td>
<td>$-$</td>
</tr>
<tr>
<td></td>
<td>gain</td>
<td>$k_v$</td>
<td>3.04×10^{-3}</td>
<td>m/A</td>
</tr>
<tr>
<td></td>
<td>flow/opening gain</td>
<td>$k_q$</td>
<td>2.7</td>
<td>m$^2$/s</td>
</tr>
<tr>
<td></td>
<td>flow/pressure gain</td>
<td>$k_p$</td>
<td>1.75×10^{-11}</td>
<td>m$^3$/s/Pa</td>
</tr>
<tr>
<td></td>
<td>leakage flow</td>
<td>$Q_c$</td>
<td>0.026</td>
<td>L/min</td>
</tr>
<tr>
<td>Motor</td>
<td>gain of the voltage</td>
<td>$k_m$</td>
<td>12.5</td>
<td>r/min/V</td>
</tr>
<tr>
<td>Pump</td>
<td>displacement</td>
<td>$D_m$</td>
<td>0.6</td>
<td>mL/r</td>
</tr>
<tr>
<td></td>
<td>volume efficiency</td>
<td>$\eta_p$</td>
<td>0.85</td>
<td>$-$</td>
</tr>
<tr>
<td>Accumulator</td>
<td>gas exponent</td>
<td>$k$</td>
<td>1.4</td>
<td>$-$</td>
</tr>
<tr>
<td></td>
<td>initial value of pressure</td>
<td>$P_0$</td>
<td>5</td>
<td>MPa</td>
</tr>
<tr>
<td></td>
<td>initial value of volume</td>
<td>$V_0$</td>
<td>0.32</td>
<td>L</td>
</tr>
<tr>
<td>Relief valve</td>
<td>gain of the flow</td>
<td>$k_f$</td>
<td>50</td>
<td>L/min/MPa</td>
</tr>
<tr>
<td>Hydraulic cylinder</td>
<td>piston area</td>
<td>$A_l$</td>
<td>2.92×10^{-3}</td>
<td>m$^2$</td>
</tr>
<tr>
<td></td>
<td>volume</td>
<td>$V_l$</td>
<td>1.47×10^{-3}</td>
<td>m$^3$</td>
</tr>
<tr>
<td></td>
<td>bulk modulus</td>
<td>$\beta_e$</td>
<td>800</td>
<td>MPa</td>
</tr>
<tr>
<td></td>
<td>leakage coefficient</td>
<td>$C_l$</td>
<td>1.0×10^{-11}</td>
<td>m$^3$/s/Pa</td>
</tr>
<tr>
<td></td>
<td>piston mass</td>
<td>$m_l$</td>
<td>55</td>
<td>kg</td>
</tr>
<tr>
<td></td>
<td>damping coefficient</td>
<td>$B_l$</td>
<td>1.2×10^{4}</td>
<td>N/m/s</td>
</tr>
</tbody>
</table>

### Step simulation of the LSVCA

First, a step simulation of force tracking is conducted in the MATLAB/Simulink condition. A step command signal of $F = 3kN$ is given by the controller at 0.1s and the system response of the LSVCA is shown in Fig. 7. Simulation result indicates that the actuator responds well to the force command signal with the rising time of about 50ms, which could meet the requirements of loading system in aircraft structure test.

Then, the frequency response of the LSVCA is analyzed with frequency sweeping method, as shown in Fig. 8. The bandwidth frequency of this actuator is about 25Hz, at which the output amplitude is attenuated by a factor of 0.707 times the amplitude of input command signal, larger than the loading system frequency of 5 Hz in the aircraft static structure test.
Overflow loss simulation of the LSVCA

The reasons that the LSVCA avoids huge overflow loss are due to both the intermittent operation mode of the motor and the energy storage characteristic of the accumulator. In order to study the normal working positions of the motor and the accumulator, simulations on the flow of each component under a constant load level are conducted in this paper. The load command is a sinusoidal signal triggering from the 6s with amplitude of 2kN±600N and frequency of 2Hz. The command signal starting from the 6s is because the motor needs to fill the accumulator before the 5s. Fig. 9 shows the flow variation of different components, including the pump, the relief valve, the accumulator and the servo-valve.

At the beginning of the accumulation process, the motor reaches to the maximum speed of 2760r/min and starts to fill the accumulator. Then the motor instantly stops and cuts off the flow supply to the accumulator at about the 5s, when the accumulator has been fully filled up. The accumulator works as a hydraulic power source for the servo-valve since the 6s and the flow of the servo-valve presents reversed sinusoidal variations with the flow of the accumulator. From the 5s, the motor is not in a working state, until the system supply pressure is lower than the threshold value at about the 10s. This means the flow of the accumulator is consumed completely at about the 10s. Then the accumulator is being refilled till the 11.5s and the above-mentioned process repeats iteratively.

In addition, there will be only small amounts of overflow loss at the point when the motor stops working, as shown in the 5s and 11.5s of Fig. 9, which is due to the lagging features of the mechanical system and the hydraulic system. This kind of overflow loss is small and inevitable. Generally, the motor keeps working at an intermittent operation mode and the relief valve only works at the point when the accumulator is filled up. The simulation results indicate that the LSVCA can reduce huge amounts of overflow loss due to both the intermittent operation mode of the motor and the energy storage characteristic of the accumulator.
Throttling loss simulation of the LSVCA

The LSVCA can also avoid large amounts of throttling loss and this is due to the supply pressure can be adjusted step by step according to the workload requirements. In order to verify the load sensing feature of the LSVCA, simulations on the flow of each component under three different kinds of load levels are conducted in this paper. The input command signal is prescribed into three levels, as shown in Fig. 10. In the first 5s and the last 5s, the input command signal is a sinusoidal signal with amplitude of 600N and offset of 500N. In the 5s to 10s and the 15s to 20s, the offset is 1700N. In the 10s to 15s, the offset is 2600N. The simulation results show that there is no big difference on the force tracking between the LSVCA and the traditional actuator. It means the proposed LSVCA has the same good tracking performance as the traditional actuator.

FIGURE 10. Force tracking simulation results

The maximum pre-set value of the system supply pressure is also prescribed into three levels, which are 9MPa, 15MPa and 21MPa, as shown in Fig. 11. Other parameters under these three load levels are recorded in Table 1. Fig. 12 also shows the flow variation of different components, including the pump and the relief valve.

At the beginning 0s, 5s and 10s, flow variations of the hydraulic pump and the relief valve are similar with the ones in the section of overflow loss simulation. In addition, at the 15s and the 21s, the step decreasing of the command signal offset causes the oil release from the accumulator and the extra energy loss. This loss is result from the switching process from high-level load to low-level load. Actually, the load level is not switched frequently in the normal aircraft structure loading system, so the loss caused by switching can be ignored.

FIGURE 11. Pump flow and relief valve flow simulation results

In the LSVCA, the accumulator works as the auxiliary hydraulic source, so its pressure is the system supply pressure. By observing its pressure variation, as shown in Fig.12, the load sensing feature of the LSVCA is explained. Fig.12 indicates that the pressure of accumulator is changing with the different load levels at the 5s, 10s, 15s and 20s. Under each constant load level, the accumulator pressure presents a slowly decreasing trend, which is because the LSVCA is consuming the hydraulic oil. Obviously, the LSCVA can automatically adjust the supply pressure according to the real-time load level and then the throttling loss is reduced.

FIGURE 12. Simulation results of the accumulator pressure
Throttling loss simulation of the LSVCA

The traditional loading system requires a centralized hydraulic power supply and a valve-controlled cylinder. No matter whether the load level changes or not, its supply pressure always keeps at 21MPa, which will lead to low system efficiency and large energy consumption. In this paper, the LSVCA can automatically adjust the supply pressure according to the real-time load level, so the system efficiency is highly improved.

Assuming that the LSVCA and the traditional actuator having no overflow loss, then their throttling efficiency is compared, as shown in Fig.1. For the traditional actuator, its efficiency is very low when the load pressure is in a low level. In contact, the LSVCA, due to its load sensing feature, always keeps a high efficiency, no matter how the load pressure changes. Under a high-level load pressure, their efficiency has no much difference, as shown in the 10-15s. However, under a low-level load pressure, as shown in the 20-25s, the efficiency of the traditional actuator is 28% and the efficiency of the LSVCA is 49%. It also indicates that the throttling efficiency of the LSVCA loading system is 1.75 times of the throttling efficiency of the traditional loading system.

FIGURE 13. Simulation results of efficiency in traditional scheme and pressure regulation scheme

ACKNOWLEDGMENTS

This work is supported in part by Aeronautical Science Foundation of China (No. 20110951009), National Nature Science Foundation for Distinguished Young Scholars of China for the financial support (No. 50825502). The authors would like to acknowledge the support from the Science and Technology on Aircraft Control Laboratory and Aviation Key Laboratory of Science and Technology on Aero Electromechanical System Integration.

REFERENCES

Oral Presentation | Pneumatics

[2B07-11] P3 (Soft Actuator)
Chair:Taro Nakamura(Chuo Univ.), Kotaro Tadano(Tokyo Institute of Technology)
Thu. Oct 26, 2017 1:40 PM - 3:00 PM Room B (ACROS Fukuoka)

[2B07] SOFT ACTUATOR TRANSFORMED INTO HELICAL SHAPE AIMED FOR IN-PIPE INSPECTION ROBOT
  *Ginjiro Kawano¹, Hideyuki Tsukagoshi¹ (1. Tokyo Institute of Technology)
  1:40 PM - 1:56 PM

[2B08] SOFT SHAPING GRIPPER INSPIRED BY MARINE ANIMALS
  *Zhonghua Guo¹, Xiaoning Li¹, Zhongsheng Sun¹, Haopeng Lin¹, Miaoxin Xu¹ (1. Nanjing University of Science and Technology)
  1:56 PM - 2:12 PM

[2B09] DEVELOPMENT OF FLEXIBLE SPHERICAL ACTUATOR WITH 3D COORDINATE MEASURING DEVICE USING LOW-COST WIRE TYPE LINEAR POTENTIOMETERS
  *Yasuko Matsui¹, Tetsuya Akagi¹, Shujiro Dohta¹, Wataru Kobayashi¹, Hiroaki Tamaki¹ (1. Okayama University of Science)
  2:12 PM - 2:28 PM

[2B10] A LOW COST MOTION SERVO CONTROL SYSTEM WITH PNEUMATIC MUSCLE ACTUATORS BASED ON PRESSURE OBSERVER AND HIGH SPEED ON/OFF VALVE
  *Hao Liu¹, Xuping YAO¹, Jun TAO¹, Xinwei ZHOU¹, Pan LYU¹, Kun LIU¹ (1. State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University)
  2:28 PM - 2:44 PM

[2B11] DEVELOPMENT OF PORTABLE REHABILITATION DEVICE USING FLEXIBLE EXTENSION TYPE SOFT ACTUATOR WITH BUILT-IN SMALL-SIZED QUASI-SERVO VALVE AND DISPLACEMENT SENSOR
  *So Shimooka¹, Shujiro Dohta¹, Tetsuya Akagi¹, Wataru Kobayashi¹, Masataka Yoneda¹ (1. Okayama University of Science)
  2:44 PM - 3:00 PM
SOFT ACTUATOR TRANSFORMED INTO HELICAL SHAPE
AIMED FOR IN-PIPE INSPECTION ROBOT

Ginjiro KAWANO*, Hideyuki TSUKAGOSHI*

*Department of Mechanical and Control Engineering,
Graduate School of Science and Engineering,
Tokyo Institute of Technology
2-12-1-S5-19 Okayama, Meguro, Tokyo, 152-8552 Japan
(E-mail: g.kawano@cm.ctrl.titech.ac.jp)

Abstract. This study proposes a novel actuator which transforms into a helix pressurized by fluid power. This actuator is constituted of a rubber tube and two cloths, which can extend to one direction and doesn’t stretch to the others. Since both torsional deformation and bending one are necessary for helical deformation, we considered the two deformations from the viewpoint of material mechanics. It can generate the grip force in pipes of various diameters. So by adapting it to the inflated segments of the inch-worm robot, it is possible to adapt it to wider pipe diameter than before. We conducted the experiment to measure the grip force for various pipe diameters. Furthermore, we demonstrated that the helical deformation tube could be applicable for a linear contraction actuator. The relationship between the contraction force and the contraction rate of the proposed actuator is examined, and the validity of the actuator is experimentally verified.

Keywords: in-pipe, pneumatic, inch-worm, helix, contraction

INTRODUCTION

Pipes are necessary components of factories, power plant, and similar constructs. Water and gas pipes are also installed in standard homes. Block and leakage often occur in the pipelines, and they may disturb the daily life of the inhabitants, increasing their risk of losing life and losing money. Therefore, inspection and maintenance are strongly demanded. The pipes are typically inspected by ultrasound or echoes from their outer surfaces. However, exterior inspection cannot reveal the internal pipe conditions completely. To inspect the internal condition, there is a method of using an endoscope equipped with a small camera. When using this method, As the propulsive distance increases, the force necessary for pushing increases. So it is not suitable for long distance inspecting.

So far, various types of in-pipe mobile robots were proposed to move inside pipes. They can be classified into wheel-based[1][2], screw-like[3]-[5], inch-worm like[6]-[8], surface wave-like[9], legged robots[10]. Among them, the inch-worm locomotion, is one of the promising methods, since it has three advantages, such as the simple design, the softness not to damage the pipes and adaptability to various environments such like elbow pipes[11]. In addition, since many robots of the type driven by pneumatic pressure without electric power are being developed, it can be suited for working where explosion risks exist.

The inflated segments of them tended to block the flow in the pipe and they were hard to generate grip forces in pipes of slightly different diameters. In order to solve the problem of blocking the flow in the pipe, a cylindrical robot body has been developed[12]. However, many robots that have been developed can not accommodate a wide pipe diameter yet. Moreover, in the conventional robot, because there is only one that generates thrust, the resistance increases according to the search distance, and it is difficult to propel for a long distance. Therefore, we aim to develop a robot, “Long Mover”, effective for the long distance inspecting by realizing the operation principle, named "Segment-Decoupled Drive", which drives as shown in FIGURE.1. This robot is propelled by deforming the extending actuator in the section fixed by the grip actuators. That is, the parts that generate thrust are distributed and arranged. As thrust is generated in multiple parts, not like the conventional robot, it is effective for the long distance search. However, like the inch-worm robot, the grip actuator of this robot still has the disadvantage that it can not deal with various diameter.

In this paper, we focus on the grip actuator, for “Long Mover” or inch-worm robots, to grip pipes shown in FIGURE.2 and propose a new actuator to overcome the previous difficulties. Namely, using the helical deformation tube for gripping, which transforms into a helix by fluid pressure, would eliminate the prevention of flow paths and can accommodate a wide pipe diameter. We conducted the experiment to measure how much the grip force generated by this actuator for various pipe diameters.
FIGURE 1. The movement of “Long Mover”

The helical deformation tube is constituted of rubber tube and two cloths, one, which is called “one-way stretch fabric”, can extend in one direction and the other doesn’t stretch. A specific configuration is shown in FIGURE 3. The two cloths are stitched along the rubber tube.

FIGURE 2. Grip way with the helical deformation tube

HELICAL DEFORMATION TUBE

Basic Configuration

The helical deformation tube is constituted of rubber tube and two cloths, one, which is called "one-way stretch fabric", can extend in one direction and the other doesn't stretch. A specific configuration is shown in FIGURE 3. The two cloths are stitched along the rubber tube.

FIGURE 3. Constitution of helical deformation tube
For helical deformation, torsional deformation and bending deformation are necessary. By pressurization, two forces, a force extending in the axial direction and a force expanding in the radial direction, are generated in the rubber tube. As these forces propagate to the clothes, the one-way stretch fabric deforms and cause the above two deformations at the same time. The mechanisms by which these two deformations are caused are explained below.

**Torsional Deformation**

The One-way stretch fabric has a property that it is easy to stretch in one direction and is hard to stretch in a direction orthogonal. As shown in FIGURE.4, by weaving fibers in one direction in a stretchable cloth, it is difficult to stretch in the fiber direction, and it tends to stretch in the orthogonal direction. In other words, Young’s modulus in the fiber direction $E_f$ has a large value with respect to Young’s modulus in the fiber orthogonal direction $E_T$. In this way, a material whose physical properties vary depending on directions is called an anisotropic material. Since the specific property of anisotropic materials is involved as a mechanism for causing torsional deformation, it is explained briefly below. Details are described in reference[13].

![FIGURE.4. Constitution of helical-actuator](image)

As shown in FIGURE.5, when the $x−y$ coordinate system of the structural principal axis and the $L−T$ coordinate system of the material principal axis are rotated by $\theta$, the stress-strain relation is shown as Eq. (1).

$$
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix} =
\begin{bmatrix}
S_{11} & S_{12} & S_{16} \\
S_{12} & S_{22} & S_{26} \\
S_{16} & S_{26} & S_{66}
\end{bmatrix}
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau
\end{bmatrix}
$$

(1)

The coefficient of compliance $S_{ij}$ of the compliance matrix in Eq.(1) is determined by Young’s modulus in each of two directions $E_f$ and $E_T$, Poisson’s ratio for each direction $\nu_{LT}$, $\nu_{TL}$, and fiber angle $\theta$. From Eq. (1), the shear strain $\gamma_{xy}$ is expressed by Eq. (2).

$$
\gamma_{xy} = S_{16} \sigma_x + S_{26} \sigma_y + S_{66} \tau
$$

(2)

From Eq. (2), it can be seen that by applying stress in two directions $\sigma_x$, $\sigma_y$, without giving shear stress $\tau$, the anisotropic material causes shear deformation, shown in FIGURE.6.

![FIGURE.5. The relationship of x−y coordinate system and L−T coordinate system](image)  ![FIGURE.6. The image how shear deformation occur](image)

As shown in FIGURE.7(a), by adjusting the fiber angle of one-way stretch fabric to be the angle $\theta$ with respect to the axial direction, the above shearing deformation is caused and the helix angle $\gamma$. We developed various
prototypes, whose fiber angle $\theta$ is $0^\circ$, $15^\circ$, $30^\circ$, $45^\circ$, $60^\circ$, $75^\circ$, $90^\circ$, and measured the helix angle $\gamma$ of the helical deformation tube. The results are shown in FIGURE 7(b). It is theoretically clarified that anisotropic materials don’t cause shear deformation due to stress when the fiber angles $\theta$ is $0^\circ$ or $90^\circ$. As shown in FIGURE 7(c), the helical angle $\gamma$ of the helical deformation tubes whose fiber angle $\theta$ is $0^\circ$ and $90^\circ$ aren’t $0^\circ$, because of the interference of the actuator itself occurs. However, since they are infinitely close to $0^\circ$, the validity of the above theory could be confirmed. Therefore, it is possible to adjust the helix angle $\gamma$ by adjusting the fiber angle $\theta$ with respect to the axial direction.

Bending Deformation

By pressurizing, the axial force is generated in helical deformation tube. As shown in FIGURE 8, the one-way stretch fabric stretches against this force, but other cloth does not stretch, so bending deformation is caused in the helical deformation tube.

The radius of curvature $R$ can be calculated by using Young's modulus for the axial direction of the three materials, rubber tube, one-way stretch fabric and other cloth. Note that, a rubber tube, one-way stretch fabric and a cloth not stretching in all directions are placed as materials 1, 2, and 3, respectively. Set the distance from the neutral axis to the center of gravity of each material as $y_i$ and the cross-sectional area of each material as $A_i$. Furthermore, set the cross-sectional area of the flow path as $S$. The equation of the equilibrium of the axial force is Eq.(3).
\[ \frac{\sum_{i=1}^{3} E_i y_i A_i}{R} - pS = 0 \]  

(3)

Assuming that the neutral axis is the bottom of the cloth not stretching as in FIGURE 9, \( y_i \) is known. From Eq.(3), the radius of curvature \( R \) is given by Eq.(4).

\[ R = \frac{\sum_{i=1}^{3} E_i y_i A_i}{pS} \]  

(4)

FIGURE.9. Sectional view of helical deformation tube

Prototypes with fiber angle \( \theta \) are 0\(^\circ\) and 90\(^\circ\) are prepared so that the helix angle is close to 0\(^\circ\), and we compare the experimental value with the theoretical value. The results are shown in TABLE 1, and FIGURE.10 shows the state of the pressurized actuator. The pressure is 0.2 MPa, and the inner diameter and the outer diameter of the tube are 6 mm and 8.4 mm, respectively. From the result, the validity of modeling by Eq.(3) is confirmed.

<table>
<thead>
<tr>
<th>( \theta )</th>
<th>Theoretical value [mm]</th>
<th>Measured value [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0(^\circ)</td>
<td>160.4</td>
<td>154.0</td>
</tr>
<tr>
<td>90(^\circ)</td>
<td>89.4</td>
<td>81.4</td>
</tr>
</tbody>
</table>

FIGURE.10. The pressurized figure  
(left: \( \theta = 90^\circ \), right: \( \theta = 0^\circ \))

THE GRIP FORCES WITH VARIOUS DIAMETER

The larger the helix diameter is, the stronger the pressing force on the pipe wall is. So, in order to increase the grip force, it is preferable that the helix diameter is large. From the theoretical formula and experiments, we confirm that the helix diameter is maximized when \( \theta \) is zero. We conducted the experiment to measure the grip force for various pipe diameters. The experimental method is shown in FIGURE.11. This helical deformation tube was constructed from the rubber tube of inner and outer diameters 8 and 11.4 mm. The pressure was 0.2 MPa. The result is shown in TABLE 2. According to the result, it was confirmed that helical deformation can produce grip force for various pipe diameters.
APPLICATION OF HELICAL DEFORMATION TUBE

We proved that the helical deformation tube can be used as a linear contraction actuator like McKibben artificial muscle[14]. In order to create a displacement in the contraction direction due to helical deformation, it is desirable that the helical angle becomes as small as possible. It is confirmed that the contraction ratio becomes the minimum when the fiber angle is 90°. As shown in FIGURE.12, it is possible to contract with a large contraction ratio against the weight of 200 g.

We investigated the difference between the contraction of the McKibben artificial muscle and the helical deformation tube by experiment. The experimental method is shown in FIGURE.13. In order to eliminate twisting due to helical deformation, actuators and force gauge are connected by a string. FIGURE.14. shows the relationship between the contraction rate and the contraction force of them under the same conditions, the length of actuators is 150 mm, the pressure is 0.3 MPa, and the inner diameter and the outer diameter of the tube are 4 mm and 5.8 mm, respectively. From these results, it is confirmed that the contraction force of the helical deformation tube is lower than that of McKibben artificial muscle, but its contraction rate is much larger. It is considered that it becomes possible to create a large range of motion if it is used for the robot arm instead of the McKibben artificial muscle.
In this paper, we proposed the helical deformation tube which transforms into a helix by fluid pressure. It can generate the grip force in pipes of various diameter. We conducted the experiment to measure the grip force for various pipe diameters, from 50 mm to 150 mm. Furthermore, we proved that the helical deformation tube can be used as a linear contraction actuator like McKibben artificial muscle. The contraction rate of this actuator is 60-70%. So, it is considered that it becomes possible to create a large range of motion if it is used for the robot arm instead of the McKibben artificial muscle.

**REFERENCES**

13. KOBAYASHI, H., TODOROKI, A., Kotai no dansoseirikigaku [Elastoplastic dynamics of solid], (Suri-kogaku sha) pp118-122
Soft shaping gripper inspired by marine animals

Zhonghua GUO, Xiaoning LI, Zhongsheng SUN, Haopeng LIN, Miaoxin XU

School of Mechanical Engineering, Nanjing University of Science and Technology
Xiaolingwei-200, Nanjing, 210094, China
(gzh@njust.edu.cn)

Abstract. It is still a challenge to develop a single gripper that can handle objects with multiple shapes. To solve this problem, soft material like elastomeric polymer is used to provide compliance for better grasping performance. In this study, a new type of gripper is designed with an artificial actuator and several vacuum shaping tentacles. The soft actuator stretches and contracts in the radial direction and the work space is enlarged. Meantime, a new structure composed of high-density elastic soft body and vacuum shaping bags is proposed. The shaping bag is filled with granular material to conform to the object shape. Therefore a reliable constraint is realized between the gripper and the object under vacuum conditions. By using the fabricated gripper, grasping experiments are carried out with objects of various shapes and sizes. The gripper can grasp several kinds of objects like rectangular box, cylindrical roll and glass ball.

Keywords: pneumatic gripper, soft actuator, vacuum shaping method

INTRODUCTION

The Chinese population is rapidly aging, and that creates a series of problems. The use of domestic service robots could finish tasks for the aged who are physical challenged. Objects at home are different in shapes and it is not convenient to change the gripper frequently for a service robot. The robot intends to use a single gripper to complete the service task, like passing a cup or washing an apple. However, it is still a challenge to develop a single gripper that can handle objects with multiple shapes[1-4]. For example, for fruits and vegetables, which are very diverse with a wide range in size, susceptibility to damage, and shape, it is difficult or impossible to grip then by traditional mechanical or vacuum grippers[5-6]. Moreover, soft and tender gripping operation is very important for elderly people. Hard and high speed industrial gripper is not suitable for domestic service applications. Instead, the soft universal gripper is to be developed and the gripping operation should be gentle and smoothing.

To solve this problem, soft material like elastomeric polymer is used to provide compliance for better grasping performance. Noritsugu developed a rotary type soft actuator made of silicone rubber[7]. This actuator is driven by compressed air and realizes rotary motion by using fiber reinforcement material. Suzumori developed flexible micro actuator (FMA) and made a soft gripper on basis of FMA actuator[8-11]. Each actuator has 3 degrees of freedom. Due to the motion of these actuators, object can be carried around and gripped up. Scholars in Zhejiang designed another kind of soft actuator called FPA, which can twist along with bend and used it in soft hand design[12]. Moreover, new types of grippers with bionic structures are designed instead of conventional mechanical structure. Laschi developed a soft manipulator modelled on the characteristic muscle structure of the octopus. In such manipulator, a deformable plastic fiber braid is designed like the octopus arm. Shape memory alloy actuators are arranged transversely and longitudinally to produce deformations, so the octopus-like robot arm wraps around the object in water[13]. Joshua Lessing and Matthew Driscoll demonstrated octopus-inspired robotic gripper to handle delicate objects[14]. The gripper tentacles curl like tentacles around a piece of broccoli, ginger or quince, and carry it over to another place. The tentacles flex in midair, and then move the object one spot over. Whitesides group developed soft structures made of elastomeric materials. They form the starfish-like gripper[15].

In this paper, a new type of gripper is designed with an artificial actuator and several vacuum shaping tentacles. The soft actuator stretches and contracts in the radial direction and the work space is enlarged. Meantime, a new structure composed of high-density elastic soft body and vacuum shaping bags is proposed. The shaping bag is filled with granular material to conform to the object shape. Therefore a reliable constraint is realized between the gripper and the object under vacuum conditions.
ENVELOPING GRIPPER STRUCTURAL DESIGN

For different environments, mollusk animals change the shapes and realize the required function (FIGURE 1). According to multi-function manipulation requirements, multi-tentacles soft gripper is designed on basis of mollusk animal structural characteristics, as shown in FIGURE 2. The enveloping gripper consists of soft actuator and several tentacles. Soft actuator completes the expansion and contraction movements. The gripper wraps the tentacles around the object and the tentacles bend according to the shape of the object. The Soft actuator made of silicone rubber locates at the center of the gripper. As the compressed air flows into the actuator, it expands and folds the tentacles connected to it. The gripper wraps the tentacles around the object and the tentacles bend according to the shape of the object. In this design, the soft actuator and tentacles are both driven by compressed air. To realize the structure of FIGURE 2, the enveloping gripper is to be designed and the tentacle must conform to the contour of the grasped object.

FIGURE 1. Marine mollusk animals

![Marine mollusk animals](image1.png)

FIGURE 2. Soft actuator and tentacle deformation

Principle of enveloping gripper

The enveloping gripper can be realized by a soft actuator and several tentacles. One proposal of the actuator is pictured in FIGURE 3. It consists of a compliant silicone bladder, surrounding a stiff pole with shell covers. There are seals at upper shell cover and downside shell cover. Threaded fasteners are used to fix the silicone bladder and seals. The tentacle units are connected to the actuator. To show the structure clearly, only one unit is shown in this figure.

FIGURE 3. Composition of proposed gripper

![Composition of proposed gripper](image2.png)

When the supply valve opens, the air source provides compressed air to the circuit and the compressed air flows into the actuator. The silicone bladder expands and the tentacle unit rotates around the hinge at the connector. As
multiple tentacle units are installed around the soft actuator, the gripper folds the tentacles and the object is grasped. When the gripper releases the object, the supply valve is cut off, and the vacuum valve opens. The air is entrained from the silicone bladder to the vacuum source and the pressure inside the soft actuator decreases. As a result, the tentacles are spread and the object is released.

When the tentacles are folded up, the end sides of tentacles touch the object. If the tentacles can be bended, the gripper and the grasped object would be a reasonably good fit. To realize such function, an air balloon is designed for each tentacle.

In this study, two component silicone gel is chosen to fabricate the body of the gripper. The two-component silicone gel is easy to mix and cure at room temperature. The silicone bladder of Soft actuator is fabricated by casting silicone gel into a custom designed mold. The mold and core are made by three-dimensional (3-D) printer with PLA. Two parts of mold and the core are then assembled and the silicone gel is poured into the chamber inside the mold. The molding device is then placed into a vacuum chamber and air bubbles are eliminated. After solidification at room temperature for about 12 hours, the core is removed out from the mold. The obtained silicone bladder is then installed in the soft actuator, and the fabricated actuator is the shape of a truncated cone.

With the same method, the tentacle balloon is made of silicone gel. The fabricated gripper is shown in FIGURE 4. The Soft actuator and tentacle balloon are controlled by two pneumatic circuits independently. When the grasping process starts, the first air source provides compressed air into the actuator circuit and the compressed air flows into the actuator. The silicone bladder expands and the tentacle units rotate. As the tentacle balloon touches the object, the second air source provides compressed air into the tentacle circuit and the compressed air flows into the tentacle balloon. The tentacles are bended to the inside of the gripper. The charged air balloon helps the gripper to grasp the object tightly.

![Actuator and Tentacle](image)

**FIGURE 4.** Photo of fabricated gripper

### Improvement for the gripper structure

The enveloping gripper is verified to be effective by designing a soft actuator and several tentacles. However, the fabricated gripper shown in FIGURE 4 is to be strengthened for the real applications. Therefore, the structure is modified and the final structure is shown in FIGURE 5. The length of the soft actuator increases when inflated with compressed air at low pressure. The operation is more stable with the mechanical components outside the rubber bladder. Besides, with the vacuum shaping bag, the gripper is much easier to conform to the contour of the grasped objects.
VACUUM SHAPING TECHNOLOGY

Vacuum shaping method

To increase the gripping force and gripping stability, the vacuum shaping method was proposed to bring tight contact between the gripper and objects. As shown in FIGURE 6, the shaping bag is stuffed with jamming granular material. At the atmospheric pressure, the granular material are free in the module. However, they will be restricted to the specific shape when the air is pumped outside the bag. Brown developed a shaping gripper of a single bag stuffed with coffee bean grains[16]. The gripping effect is verified through experiments. However, it is limited by using single bag for objects of different sizes and shapes. In this paper, multi shaping bags are developed and combined with the soft tentacles. Gripping performances with different kinds of granular material are also compared and analyzed.

Important parameters for vacuum shaping

The gripping effect of vacuum shaping bags is influenced by two parameters. They are vacuum degree and the depth of the object inside the shaping bag. Experimental apparatus are shown in FIGURE 7. The object of a wood ball is connected with the cylinder rod. The wood ball is then inserted into the shaping bag with the thrust force of a cylinder rod. After that, the air is entrained by a vacuum ejector from the shaping bag to the atmosphere. Vacuum condition is established in the shaping bag and the vacuum degree is measured by a pressure sensor. A laser sensor is used to measure the insert depth of the wood ball into the shaping particle bag. Change the lifting force by adjusting the cylinder inner chamber pressure until the wood ball gets out of the shaping bag. Record the air pressure at that second. The lifting force can be calculated by using the air pressure and the effective area of the cylinder.
Experiments are carried out where the vacuum degree varies from 5 kPa to 85 kPa and the insert depth varies from 3 mm to 30 mm. Shaping bags stuffed with two kinds of granular material (Coffee beans and quartz sands) are tested in the experiments. The result is shown in FIGURE 8. Below the vacuum degree of 60 kPa, the maximum lifting air pressure increases when the vacuum degree is higher. However, over 60 kPa, the increasing trend is not significant. Also, the relationship between the lifting force and the insert depth is influenced by the material type. For insert depth below 26 mm, the maximum lifting air pressure is higher for coffee bean grains. That means larger lifting force is needed to pull the wood ball outside of the shaping bag and coffee bean grains provide tighter contact for the gripping operation. However, as the insert depth increases and the depth is above 26 mm, the quartz sand provides better gripping performance because of higher rigidity.

**FIGURE 8.** Experimental apparatus for vacuum shaping parameters

**ENVELOPING GRIPPER WITH TENTACLES AND VACUUM SHAPING BAGS**

**Overall structure**

The gripper is developed where soft tentacles and vacuum shaping bags are designed, as shown in FIGURE 9. The actuator is bought from FESTO. The length of the actuator decreases when air is entrained out of the actuator bladder and increases when inflated with compressed air. As a result, the enveloping gripping is accomplished through the levers and the tentacles. Besides, the vacuum shaping bags are filled with granular material to conform to the object shape. Therefore a reliable constraint is realized between the gripper and the object under vacuum conditions.
The soft tentacles are used to realize the enveloping gripping and they contact with the objects completely. To obtain effective bending action, the non-symmetry folded air bag is designed where the cutting plane is located at one side of former axis on basis of folded air bag. As shown in FIGURE 10, when inflated or entrained with air, the length changes of the left side and right side are different, and therefore the bending action is achieved. The soft tentacle consists of the main body, shaping membrane and strengthened membrane. They are located at two sides of the soft stators. The mold for the tentacles is shown in the right side of FIGURE 10, where the buckling-closing structure is proposed for larger contact area of different parts of the mold.

The shaping membrane and strengthened membrane is made as box structure stuffed with shaping granular material. There is also some gap between the inner layer of the shaping membrane and strengthened membrane to ensure that the granular material flow freely inside the shaping membrane and strengthened membrane. The charging and discharging hose is embedded in the soft stators and there are sieves at the inlet port of the hose to prevent the shaping granular material enter the pneumatic circuit. To make the fabricated tentacle soft, elastic, and environment-friendly, the food grade silicone gel is used to make the soft elements. To realize the shapes of marine soft animals, the complicated custom designed molds are made by three-dimensional (3-D) printer with PLA.

FIGURE 9. Gripper with soft tentacles and vacuum shaping bags

FIGURE 10. Structure and fabrication mold for tentacles
FIGURE 1. Soft tentacles combined with vacuum shaping bag

**Gripping experiment**

In the experiment, as shown in FIGURE 12-13, objects of different shapes can be manipulated easily and stably by the enveloping soft gripper with soft tentacles and vacuum shaping bags. The magic tentacles conform to the contour of the objects well.

FIGURE 12. Gripping operation conforming to contour of the object

FIGURE 13. Objects handled by proposed soft gripper
CONCLUSION

The soft gripper is designed with a soft pneumatic actuator and several tentacles. It is flexible with shape adaptation for objects of different shapes and sizes. With the power of compressed air, the soft actuator expands and folds the tentacles connected to it. Based on such principle, the gripping operation is accomplished. The silicone bladder of soft actuator and the tentacle balloon are fabricated by casting silicone gel into custom designed molds. Soft actuator made of silicone rubber is located at the center of the gripper. As the compressed air flows into the actuator, it expands and folds the tentacles connected to it. Therefore, the enveloping gripping is realized inspired from marine animals.

On basis of such structure, the gripper is strengthened by improving the actuating components of the soft actuator and adding the vacuum shaping bags into the gripper tentacles. The fabricated gripper tentacles are composed of high-density elastic soft body and vacuum shaping bags. The shaping bag is filled with granular material to conform to the object shape. Therefore a reliable constraint is realized between the gripper and the object under vacuum conditions. By using the fabricated gripper, grasping experiments are carried out with objects of various shapes and sizes.

ACKNOWLEDGEMENTS

Financial supports of following projects are gratefully acknowledged:

- Project supported by National Natural Science Foundation of China (Grants No. 51305202)
- Project supported by the Natural Science Foundation of Jiangsu Province (Grants No. BK20130764)

REFERENCES

DEVELOPMENT OF FLEXIBLE SPHERICAL ACTUATOR WITH 3D COORDINATE MEASURING DEVICE USING LOW-COST WIRE TYPE LINEAR POTENTIOMETERS

Yasuko MATSUI*, Tetsuya AKAGI*, Shujiro DOHTA*, Wataru KOBAYASHI*, Hiroaki TAMAKI*

* Graduate School of Engineering, Okayama University of Science
1-1 Ridai-cho, Kita-ku, Okayama, 700-0005 Japan
(E-mail: t16sd03my@ous.jp, {akagi, dohta, kobayashi}@are.ous.ac.jp, t16rm03@ous.jp)

Abstract. Rehabilitation devices help to recover physical ability of patients. This study aims to develop a portable rehabilitation device which can be safe to use while patients are holding it by hands. In the previous study, to realize a home rehabilitation device, the flexible spherical actuator that can give motions to patients was developed. In this study, to measure the relative position between both handling stages, a 3D coordinate measuring device using three wire type linear potentiometers and an embedded controller was proposed and tested. In this paper, the spherical actuator with built-in in the 3D coordinate measuring device is described. The measuring method and experimental results using the device are also described. The tracking position control of the actuator using the measuring device was carried out. As a result, the position of the handling stages can be successfully controlled by using the feedback signal from the tested device.

Keywords: Flexible spherical actuator, Flexible pneumatic cylinder, 3D coordinate measuring device, Wire type linear potentiometer.

INTRODUCTION

Recently, a system to aid in nursing care [1] and to support activities of daily life for the elderly and the disabled is required in an aging society [2, 3]. Rehabilitation devices help to recover physical ability of patients for keeping Quality of Life. The actuators used in such a system need to be flexible so as not to injure the human body [4]. This study aims to develop a portable rehabilitation device which can be safe to use while patients are holding it by human hands. In particular, from a view point that patients use it at home by themselves, the device must be able to be used without special knowledge and be easily available at a low cost. In our previous study, a flexible pneumatic cylinder that can be used even if the cylinder is deformed by external forces has been proposed and tested [5]. We also developed a spherical actuator using the flexible pneumatic cylinders, which can be used on a table as a rehabilitation device for human wrist and arm [6-8]. A portable rehabilitation device using the flexible spherical actuator that consists of two flexible pneumatic cylinders was proposed and tested. The portable rehabilitation device can be produced at low cost. In addition, to realize an inexpensive home rehabilitation device, the flexible spherical actuator that can give motions to patients with sequential control scheme. However, in order to keep safety, the device needs to have a monitoring function of relative position between both handling stages to prevent to crash each other. Moreover, the relative position is used as a feedback signal for attitude control of the device. In this study, to measure the relative position between both handling stages for keeping safety and attitude control of the device, a 3D coordinate measuring device using three wire type linear potentiometers and an embedded controller was proposed and tested [9]. In this paper, the spherical actuator with built-in 3D coordinate measuring device is described. The measuring method and experimental results using the device are also described.

FLEXIBLE SPHERICAL ACTUATOR

Figure 1 shows the construction of a low-friction type flexible pneumatic cylinder developed in the previous study [10]. The cylinder consists of a flexible tube as a cylinder and gasket, one steel ball as a cylinder head, and a slide stage with 12 steel balls which are set on the inner bore of the stage to press and deform the tube. The operating principle is as follows. When the supply pressure is applied to one side of the cylinder, the inner steel ball is pushed. At the same time, the steel ball pushes the slide stage moves toward opposite side of the pressurized side while deforming the tube. The frictional force of the cylinder is relatively large compared with...
a typical rigid pneumatic cylinder. The minimum driving pressure of the cylinder is 94 kPa. This value is smaller than the case using the previous flexible pneumatic cylinder, that is 120 kPa [5].

![Diagram of a typical rigid pneumatic cylinder](image1)

**FIGURE 1.** Flexible pneumatic cylinder.

Figure 2 shows the appearance of the tested spherical actuator using two ring-shaped flexible pneumatic cylinders. In this case, the previous type flexible pneumatic cylinders were used. Two cylinders are intersected at right angle and each slide stage is fixed on each handling stage. The actuator can give a passive exercise for user’s shoulders and arms while they hold both handling stages with hands. The actuator can give movement of the upper limb. However, to realize the desired motion, it is necessary to measure the relative coordinate between both stages. Figure 3 shows the transient view of the spherical actuator. The supply pressure is 450 kPa. From Fig.3, it can be seen that the actuator can create the different attitudes easily. In addition, from the view of the movement of both arms, we found that it gave the motion for not only wrist but also arms. Generally, the passive exercise such as proposed motion is useful to recover the moving area of joints and the function of nerves and muscles. However, as patients must use it alone, it is better to observe a relative position between both handling stages to prevent to occur crash accident of patient’s hands.

![Diagram of the flexible spherical actuator](image2)

**FIGURE 2.** Appearance of the flexible spherical actuator.

![Transient view of the movement of the device](image3)

**FIGURE 3.** Transient view of the movement of the device.

**LOW-COST WIRE TYPE LINEAR POTENTIOMETER**

In order to measure the displacement of the flexible pneumatic cylinder directly, a low-cost wire type linear potentiometer was proposed and developed in our previous study [11]. A wire type liner encoder on the market is well known as a displacement sensor with a long stroke measurement. However, the cost of the encoder is very expensive, that is about 500 US dollars. The measuring resolution of the encoder is too high to apply a position control of flexible pneumatic cylinder that required resolution of less than 1 mm. In addition, in order to obtain the displacement from the encoder output, a high-spec built-in embedded controller with faster processing speed is required for processing high-frequency pulse signals from the encoder. Therefore, the low-
cost wire-type linear potentiometer whose output analogue signal is easily processed by a tiny embedded controller through A/D converter was developed. Figure 4 shows the construction of a low-cost wire type linear potentiometer. The tested potentiometer consists of a helical potentiometer (BOURNS Co. Ltd., 3590S-A26-104L) that can measure 10 times rotational angle, a clockwork wire spool with diameter of 22 mm and a flexible stainless steel wire with diameter of 0.4 mm. Both shafts of the potentiometer and the wire spool are connected each other. From a rotational angle of the helical potentiometer and the diameter of the wire spool, the maximum length for measurement of about 0.7 m can be expected. The resolution of the potentiometer using 10 bit A/D converter is about 0.74 mm. The cost of the linear potentiometer except for the stainless steel wire is inexpensive, that is about 8 US dollars. In the previous study, the flexible pneumatic cylinder with built-in wire type linear potentiometer that the cylinder head is connected with the wire of the tested potentiometer through special sealing mechanism as shown in Fig. 5 was developed [11]. Figure 6 shows the transient response of displacement of the tested cylinder. In Fig. 6, the blue broken line and red solid line show the desired and controlled displacement of the tested cylinder, respectively. The sampling period of position control is 1 ms. From Fig.6, it can be confirmed that the position of the cylinder can be controlled by using the tested sensor. However, the relative coordinate between both stages cannot be measured by using the displacement sensor of the flexible pneumatic cylinder because of lower stiffness of the spherical actuator.

![Helical potentiometer and Clockwork wire spool](image1)

**FIGURE 4.** Low-cost wire type linear potentiometer.

![Flexible pneumatic cylinder with special sealing mechanism](image2)

**FIGURE 5.** Flexible pneumatic cylinder with wire type linear potentiometer and special sealing mechanism.

![Transient response of displacement of the cylinder](image3)

**FIGURE 6.** Transient response of displacement of the cylinder for multi-position control.

As mentioned above, the displacement of the cylinder was successfully measured. However, in order to apply the spherical actuator to rehabilitation devices, it is required to measure not only displacement of the flexible pneumatic cylinder but also the relative coordinate between both handling stages because of its flexibility of the spherical actuator. Therefore, a low-cost three dimensional coordinate measuring device that can apply rehabilitation devices was proposed and tested [9].
3D COORDINATE MEASURING DEVICE

Figure 7 shows a 3D coordinate measuring device using three improved wire type linear potentiometers described below. In the device, the end of wire of each potentiometer are connected each other. Each wire outlet from the potentiometer are arranged so that each distance from the measuring origin is kept at a certain distance \( d \). Figure 8 shows the measuring model of the device. From a geometric relationship, following equations related to the distance \( d \) and coordinate of measuring point \((x, y, z)\) can be obtained.

\[
\begin{align*}
D_1^2 &= (x - d)^2 + y^2 + z^2, \\
D_2^2 &= x^2 + (y - d)^2 + z^2, \\
D_3^2 &= (x + d)^2 + y^2 + z^2. 
\end{align*}
\]

(1)

(2)

(3)

Equations (4) to (6) are derived from Eqs. (1) to (3).

\[
\begin{align*}
x &= \frac{1}{4d} (D_3^2 - D_1^2), \\
y &= \frac{1}{4d} (D_3^2 - 2D_2^2 + D_1^2), \\
z &= \sqrt{D_3^2 - (x + d)^2 - y^2}. 
\end{align*}
\]

(4)

(5)

(6)

From Eqs. (4) to (6), it can be seen that the coordinate can be obtained by measuring each distance \( D_1, D_2 \) and \( D_3 \).
In order to realize the low-cost 3D measuring system, the wire type linear potentiometer is useful because the estimated cost of their parts is less than 10 US dollars. However, the tested linear potentiometer has a little problem that the rotary shaft of the spool and the helical potentiometer are not connected directly. This flexible connection of both shafts causes measuring error. In addition, to measure the relative coordinate between handling stages of the actuator based on a principle of triangulation, it is necessary to realize a high resolution measurement in the appropriate range. Thus, an improved wire type linear potentiometer as shown in Fig. 9 was proposed and tested. Compared with the previous potentiometer, the spool is directly connected to the shaft of the potentiometer. Accordingly, it realized a smooth motion. Further, the diameter of the clockwork wire spool is shortened from 22 mm to 10 mm. As a result, the resolution of the sensor was improved to 0.45 mm. The size of the potentiometer becomes smaller, that is 30 mm in height, 30 mm in width and 42 mm in length. The mass of the potentiometer is 44 g. The maximum length for measurement is about 260 mm.

![FIGURE 9. View of the improved wire type linear potentiometer.](image)

Figure 10 shows an experimental setup of the 3D measuring device to investigate the measuring accuracy of the device. The experimental setup consists of the tested 3D measurement device and two kinds of ring-shaped disks with different inner diameters of 100 and 160 mm. Two disks are set on the base plate with various height by using spacer rods. In the experiment, the tip of the measuring device moves along the inner bore of the disks.

![FIGURE 10. Experimental setup of 3D measuring device.](image)

Figure 11 shows the experimental results of the 3D coordinate measurement using the device. In the experiment, the tip of the device was moved from 0 to 360 degrees of rotation angle \( \theta \) along the inner bore of disks with two kinds of diameters. In Fig. 11, solid and broken lines show the trajectory of measuring point and the true value, respectively. From Fig. 11, it can be seen that the device can measure the coordinate within error of 5 mm. This measuring accuracy is sufficient to apply the spherical actuator that does not require precise positioning.
In order to install the 3D coordinate measuring device in the spherical actuator, the size of the device becomes serious concerns. Therefore, a built-in type 3D coordinate measuring device is proposed and tested. Figures 12 (a) and (b) show the view of whole spherical actuator and the built-in 3D coordinate measuring device, respectively. As shown in Fig.12 (b), by changing the setting direction of each wire type linear potentiometer, the compact 3D coordinate measuring device can be realized even if the distance from the measuring origin to each wire start point is kept at a same distance $d$ of 35 mm. The size of the measuring device is 100 mm in length, 47 mm in width and 36 mm in height. The mass of the measuring device is about 165 g. The required space for setting of the device decreased to 46% of the previous one. From Fig. 12 (a), the device is set so as to meet both coordinates of the device and actuator each other.

Figure 13 shows the schematic diagram of the position control system of the tested flexible spherical actuator. The control system consists of the spherical actuator with the 3D coordinate measuring device, four quasi-servo valves [12] to operate two flexible pneumatic cylinders and an embedded controller (Renesas Co. Ltd., SH7125). The position control of the device is done as follows. First, the embedded controller gets output voltages from three wire type potentiometers. The coordinates of the measuring point $(x, y)$ is calculated by them. By comparing them with the desired position set in advance, the deviation from the desired position can be calculated. As a control scheme, the following simple proportional control scheme was used.

$$u_i = k_p(r_i - c_i) \quad (i = x, y), \quad (7)$$

$$d_i = u_i + 22.5 \quad (i = x, y), \quad (8)$$
where, \( u_i \), \( r_i \), \( c_i \), and \( d_i \) mean the control input, the desired coordinate and the measured coordinate for \( x \) and \( y \) direction and the input duty ratio of the PWM valve, respectively. The switching valves are changed based on negative or positive value of the control input \( u_i \). The input duty ratio for PWM valve \( d_i \) is added 22.5% to compensate the dead zone [12]. By this method, quasi-servo valves are driven and the flexible pneumatic cylinders are controlled in \( x \) and \( y \) direction independently. In the control, the sampling period of 5 ms was used. The PWM period of the quasi-servo valve is 10 ms.

Figure 14 shows the transient response of \( x \) and \( y \) coordinate of the handling stage in the tracking position control. In Fig. 14, the broken and solid lines show the desired coordinate and controlled position for \( x \) and \( y \) directions, respectively. In this case, same desired coordinate was given for both \( x \) and \( y \) directions. From Fig.14, it can be seen that the position of the cylinder can be controlled by using the feedback signal from the 3D coordinate measuring device. However, we can observe a non-negligible error between the desired and measured position. The positioning accuracy of the device will be improved by using a control scheme with consideration of compensating the frictional force and time delay of the actuator.

Figure 13. Position control system of the device.

Figure 14. Transient response of \( x \) and \( y \) coordinate of the handling stage.

**CONCLUSIONS**

In order to prevent crash accident of patient’s hands, the 3D coordinate measuring device that can obtain the relative coordinate between two stages in the flexible spherical actuator was proposed and tested. The measuring device consists of three wire type linear potentiometers and an embedded controller. To confirm the validity of the device, the preliminary measurement using the theoretical model was carried out. As a result, it was confirmed that the tested device was possible to measure the coordinate between both handling stages within error of 5 mm. The spherical actuator with the built-in 3D coordinate measuring device was also constructed. The tracking position control of the actuator using the measuring device was carried out. We confirmed that the
position of the cylinder can be controlled by using the feedback signal from the tested device. As future work, we are going to improve the positioning accuracy by applying a control scheme with consideration of compensating the frictional force and time delay of the actuator.

ACKNOWLEDGMENTS

This study was supported by the Education Ministry of Japan through a Financial Assistance Program for QOL Innovative Research (2012-2016) and a Grant-in-Aid for Scientific Research (C) (Subject 16K06202 & 17J09802).

REFERENCES

A Low Cost Motion Servo Control System with Pneumatic Muscle Actuators based on Pressure Observer and High speed on/off valve

Hao LIU *, Xuping YAO*, Jun TAO*, Xinwei ZHOU*, Pan LYU* and Kun LIU*

* State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University, 38 Zheda Road, Hangzhou, 310027, China
(E-mail: hliu2000@zju.edu.cn)

Abstract. A pneumatic muscle actuator (PMA) is flexible and acts like a human muscle. It has a good feature of high force-to-weight ratio and is applied to robots and other industrial fields widely. This paper presents a new low-cost motion servo control system driven by a pneumatic muscle actuator, with a pair of high speed on/off solenoid valves instead of the expensive proportional valves and a pressure observer instead of the pressure sensors which are used in the pneumatic servo system generally. The nonlinear factors including the hysteresis of PMA and the compressibility of air flow are the great disadvantages for control with high accuracy. To approximate those main nonlinear factors, a single-input single-output (SISO) model of the system is carefully developed, based on analysis of the relationship of the air flow, the pressure, and the force dynamics of the system. Based on the model, a controller combining sliding mode control (SMC) and nonlinear pressure observer has been proposed for the system. After theoretical discussions, an experiment apparatus is built up and the trajectory tracking control of the system under sinusoidal input signals is carried out. The results of the experiments demonstrate the performance of the controller is very good in both precision and robustness, under different frequency tracking signals and loads.

Keywords: Motion Servo Control, Pneumatic Muscle Actuator, Pressure Observer, High Speed on/off Valve, Sliding Mode Control.

INTRODUCTION

PMA has been widely used in recent years due to its clear advantages [1]. The outstanding features of PMA, such as high force-to-weight ratios, cleanliness, compactness, and ease of maintenance, meet the special demands of many industrial designs. However, there are some disadvantages of it, especially the nonlinear characteristics caused by air compressibility and friction, make the motion of it is difficult to be controlled with high accuracy. In recent years, a number of control strategies have been proposed to overcome those disadvantages. Such as PID control, adaptive control strategies, nonlinear PID, neural networks, sliding mode control (SMC), and so on. Among those studies of PMA motion control, the SMC control strategy was proved to be accurate, efficient and robust [2]. In one case, the SMC was applied with a proportional flow valve in an opposing pair configuration, and the simulation result demonstrated that SMC is a very promising method for the control of PMA. In a 2-dof scara-type robot driven by PMA, SMC strategy based on 2-nd order model was applied and a static joint accuracy of 0.02 o was realized [3]. In most researches of SMC on PMA motion control, the PMA are controlled by proportional flow valves. But because of its expensive price, the proportional flow control valve are replaced by high speed on/off valves in some cases and their performance are close. In [4], a master-slave tele-operation system driven by PMA with inexpensive solenoid valves was developed. JOUPPILA, V. T., used high speed on/off valve to control pneumatic system and proved that the system can work better than the system with proportional valves or servo valves [2]. Also, a pair of high speed on/off valves will be applied to control the PMA with SMC in pulse width modulation in this paper.

As the pressure in pneumatic muscle is usually used as a state variable in the PMA servo control, a pressure sensor is necessary in the system. Sometimes it is becoming a burden for the system, because high precision pressure sensors are expensive or too large to be installed. In order to reduce costs and the complexity of the PMA control system, pressure sensor should be used as little as possible. So, pressure observers are used to replace pressure sensor [5]. WU, J. H. analyzed the feasibility of using nonlinear pressure observer instead of pressure sensor in pneumatic servo actuator system and got the conclusion that the system may lost local observation when the system is static or reversed [6]. NAVNEET, G. proposed a global Lyapunov pressure observer based on the assumption that the process is an isothermal process [7]. But most researches of pressure observer are based on proportional valves or pneumatic cylinder systems. In this paper, a pressure observer based PMA motion control with high speed on/off valves will be studied.

The rest of the paper is organized as follows. Section 2 presents the modeling of the PMA control system. Section 3 describes the design of a sliding mode controller with pressure observer. The result of experiment is given in section 4. Section 5 provides some conclusions.
MODELLING OF THE MOTION SERVO CONTROL SYSTEM WITH PMA

Description of the system

The schematic of motion servo control system with PMA is depicted in FIGURE 1. This setup consists of a PMA (FESTO RMSP-20-180N-RM-CM), a pair of high speed on/off solenoids valves (MHE2-MS1H-3/2G), a pressure sensor (Huba control 5436) and a laser displacement sensor (OptoVCDT 1700 ILD 1700-500). The PMA is hung freely from a rigid support with a load fixed to its free end. The movement of the load is recorded by the displacement sensor when PMA is contracting in control processes. Charging or discharging of the PMA is controlled by two high speed on/off solenoid valves respectively. With this configuration, both valves can be closed when the state of the tracking signal is close to the desired signal. It also can be used to save gas and increase lifetime of valves. Both of the valves work in the pulse-width modulation (PWM) mode.

The equation of motion for the system can be expressed as

\[ M \ddot{x} = F_m - Mg \]  

(1)

Where \( M \) is the loads, \( x \) is the contraction length of pneumatic muscle. \( F_m \) is the force of pneumatic muscle.

Model Identification of PMA

The model of PMA defines a relationship between the contraction rate of the PMA, its internal pressure and the axial force it exerts on the load. Because the exerted force holds a non-linear relationship with pneumatic contraction rate and a linear one with pressure [8], Pujana-Arrese proposed a generalized empirical model for static mechanical force exerted by muscle of form [9] :

\[ F(\varepsilon, p) = (a_1 + a_2 p) + (a_3 + a_4 p) \varepsilon + (a_5 + a_6 p) \varepsilon^2 + a_7 \varepsilon^3 + a_8 \varepsilon^4 \]  

(2)

Where \( F \) is the force of pneumatic muscle and \( p \) is the internal pressure of pneumatic muscle, \( \varepsilon \) is the contraction rate equalling to \( x/l_0 \) with \( l_0=0.18 \)m. \( a_1, a_2, \ldots, a_8 \) are coefficients and will be identified through experiments. The experiment set up is illustrated in FIGURE 2.
Based on the data of pressure-force experiments, pressure-contraction experiments and force-contraction experiments, the parameters of the PMA model are identified, with the fitting tools of MATLAB/SIMULINK. They are showed in TABLE 1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a_1$</td>
<td>$46.56$</td>
<td>$a_3$</td>
<td>$109431.76$</td>
</tr>
<tr>
<td>$a_2$</td>
<td>$225.33$</td>
<td>$a_6$</td>
<td>$-5026.69$</td>
</tr>
<tr>
<td>$a_3$</td>
<td>$-11038.45$</td>
<td>$a_7$</td>
<td>$-527269.10$</td>
</tr>
<tr>
<td>$a_4$</td>
<td>$245.55$</td>
<td>$a_8$</td>
<td>$1103778.34$</td>
</tr>
</tbody>
</table>

The dynamic component of PMA model is mainly caused by the thread-on-tube friction and hydrodynamic friction between nylon. The inertia of the actuator itself can be ignored. However, these two kinds of friction are difficult to be built up accuracy models. So the above static mechanical force model of PMA is used in the designing of the SMC controller in the following section, while the thread-on-tube friction and hydrodynamic friction are treated as the disturbances of the system in the controller.

**Pressure Dynamics**

For building the thermodynamic model of the air in PMA, the thermodynamic process of the air is assumed to be an isotherm process, as the control process of PMA is always under a low frequency and the volume change of the PMA is not fast. Then the pressure differential equation of the gas in PMA can be show below, combining ideal gas law, mass continuity equation and the first law of thermodynamics.

$$\frac{\dot{p}}{V} = \frac{kR \dot{T}}{V} - \frac{kp \dot{V}}{V}, \quad T = T_s \tag{3}$$

Where $p$ is the pressure in PMA, $k$ is the ratio of specific heat which equals to $c_p/c_v$. $V$ donates the volume in pneumatic muscle, $R$ is the gas constant and equals to 287.1 J/kg/K, $T$ is the temperature in pneumatic muscle, $T_s$ is the temperature of gas source, $m$ is the net mass in pneumatic muscle.

As shown in Eq. (3), the volume of the muscle $V$ is one of the key factors that determine the pressure inside the PMA. To verify the variations of volume in the control process, the diameter and displacement of the muscle was measured while filling the muscle with different amount of air. Treating the muscle as a cylindrical shape, the corresponding
volume of the muscle can be calculated with the data. FIGURE 3 shows the calculated results from experiment data. It reveals the volume has a linear relationship with the displacement, which can be describe as:

\[ V(x) = V_0 + V_1 x \]  

(4)

where \( V_0 \) and \( V_1 \) are the fitting coefficient, \( V(x) \) is the volume of the muscle, \( x \) is the displacement.

![Experiment data](image)

![Fitting curve](image)

**FIGURE 3.** Muscle volume in correlation with Displacement

### Model of High Speed On/off Solenoid Valve

The mass flow rate of the gas entering or leaving the PMA is controlled by the high speed on/off solenoid valves. As the valves work in the PWM mode, SANVILLE’s flux formula of the gas passed valve port is accepted to describe the process in this paper [10]. The valve port area controlled by PWM equals to the product of duty-cycle and the max area of valve port. Assuming an ideal gas law and an adiabatic process, the flux formula is expressed as follows:

\[
\dot{m} = \begin{cases} 
  u_d S_{\text{max}} \frac{p_u}{\sqrt{T}} \left( \frac{2}{R(k-1)} \left( \frac{p_d}{p_u} \right)^{\frac{k}{k-1}} - \left( \frac{p_d}{p_u} \right)^{k+1} \right) & \frac{p_d}{p_u} > 0.528 \\
  u_d S_{\text{max}} \frac{p_u}{\sqrt{T}} \left( \frac{2}{k+1} \right)^{\frac{k}{k-1}} \left( \frac{2}{R(k+1)} \right) & \frac{p_d}{p_u} \leq 0.528 
\end{cases}
\]

(5)

Where \( u_d \) is duty-cycle of PWM, \( p_u \) and \( p_d \) is the absolute pressure of upstream and downstream of the valve port. \( S_{\text{max}} \) is the max area of the valve port which can be measured by experiment.

### Model of the Motion Servo Control System

Considering the whole system, a single-input single-output (SISO) model can be obtained as follow from Eq. (1) and Eq. (2).

\[
\ddot{x} = \frac{1}{M} F_m (\dot{p}, x, \dot{x}, \ddot{x}) = f(x, \dot{x}, \ddot{x}) + g(x, \dot{x}, \ddot{x}) \dot{p}
\]

(6)

Make linearization near the stable point \( x = [x, \dot{x}, \ddot{x}] = [0, 0, 0] \) due to coupling of state varies, then:

\[
f(x, \dot{x}, \ddot{x}) = \frac{1}{Ma_2} (-Ma_4 - a_2a_3) \dot{x} \]

(7)

\[
g(x, \dot{x}, \ddot{x}) = \frac{a_2}{M}
\]

(8)

Transform the 3-order differential Eq. (6) to state space equation, then
\[
\dot{X} = \begin{bmatrix} 0 & 1 & 0 \\ 0 & 0 & 1 \\ 0 & -6218.42 & 0 \end{bmatrix} X + \begin{bmatrix} 0 \\ 0 \\ 22.53 \end{bmatrix} \dot{p} \quad (9)
\]

\[
\begin{align*}
\dot{x}_1 &= x_2 \\
\dot{x}_2 &= x_3 \\
\dot{x}_3 &= a(x) + bu
\end{align*} \quad (10)
\]

Where \( a(x) = -6222.1 x_2 \), \( b = 22.53 \), \( u = \dot{p} \) with \( M = 10 \) kg and \( l_0 = 0.18 \) m.

**SLIDING MODE CONTROLLER WITH PRESSURE OBSERVER**

SMC is a discontinuous nonlinear control algorithm which can be used to make the system vary according to current states by keep the states values close to desired state plane termed sliding surface. When the states values is away from the surface a switch gain would be used to decrease the error between states values and desired values. Once the states values is on the sliding surface, the state slid among the surface is called slinger mode. SMC is proved to be robust, high accurate and stable in the control of PMA. A boundary layer around sliding surface is used to decrease the chattering. In this paper, a SMC controller will be built for the PMA motion control, and the value of the pressure inside the muscle that will be used in the controller is given by a pressure observer, to eliminate the requirement of pressure sensor.

**Design of a Sliding mode controller**

The sliding surface is set as

\[
s = c_1 e + c_2 \dot{e} + \ddot{e} \quad (13)
\]

Where \( s \) is the sliding surface, \( c_1 \) and \( c_2 \) are the system parameters, and \( e \) is the error.

Take the derivative of Eq.(13), then

\[
\dot{s} = c_1 \dot{e} + c_2 \ddot{e} + \dddot{e} = c_1 \dot{e} + c_2 \ddot{e} + \dddot{x}_1 - \dddot{x}_d = c_1 \dot{e} + c_2 \ddot{e} + a(x) + bu - \dddot{x}_d \quad (14)
\]

A robust control law can be obtained by combining the equivalent control with the switching control.

\[
u(t) = u_{eq} + u_{sw} \quad (15)
\]

Where \( u_{eq} \) is the equivalent control rate and \( u_{sw} \) is the switching control rate.

In this application \( u_{eq} \) and \( u_{sw} \) can be obtained by setting \( \dot{s} = \eta sat(s) \). It is easy to be found out that the control rate function is:

\[
u(t) = u_{eq} + u_{sw} = \frac{1}{b} (-c_1 \dot{e} - c_2 \ddot{e} - a(x) - \eta sat(s) + \dddot{x}_d) \quad (16)
\]

Where \( sat(s) \) is a saturation function and equals to:

\[
sat(s) = \begin{cases} 
1 & s > \Delta \\
k s & \left| s \right| \leq \Delta \\
-1 & s < -\Delta 
\end{cases} \quad k = 1/s
\quad (17)
\]

Where \( \Delta \) is the boundary layer of sliding mode surface.
Design of a Pressure observer

The compressed air is considered as ideal gas here and the ideal gas law is

\[ p = \rho RT \]  

(18)

Where \( p \) is absolute pressure in PMA, \( \rho \) is density, \( T \) is absolute temperature and \( R \) is gas constant. The thermodynamic process can be seen as polytropic process, its thermodynamic model is considered as:

\[ \frac{p}{\rho^n} = c \]  

(19)

\( c \) is a constant and \( n \) is the polytropic index. Combining the differential format of Eq. (18) and Eq. (19), it is easy to be obtained that

\[ \dot{p} = nRT\dot{\rho} \]  

(20)

Assuming the volume in PMA is invariant, the gas mass flow rate equation is derived as follows:

\[ \dot{m} = \dot{\rho}V + \rho\dot{V} \]  

(21)

Where \( m \) is the quality of gas and \( V \) is the volume of gas. Substituting equation (21) in equation (20) leads to the following result:

\[ \dot{p} = \frac{nRT\dot{m}}{V} - \frac{nPV}{V} \]  

(22)

The thermodynamic process of gas in pneumatic muscle is considered as isotherm process in this paper and \( n \) is considered as 1. Then Eq. (22) can be simplified as

\[ \dot{p} = \frac{RT\dot{m}}{V} - \frac{P\dot{V}}{V} \]  

(23)

It is easy to build up a closed loop pressure observer like Eq. (24) because the relation between \( P \) and \( \dot{m} \) is a closed loop.

\[ \hat{p} = \frac{RT\hat{m}}{V} - \frac{\hat{P}\dot{V}}{V} \]  

(24)

Where \( \hat{P} \) and \( \hat{m} \) are the estimated pressure and mass flow rate. A Lyapunov function chosen to verify the pressure observer is convergent:

\[ V_i = \frac{1}{2}\left[\hat{P}\dot{V}(x)\right]^2 \]  

(25)

Where \( \hat{P} = \hat{p} - P \).

Differentiating Eq. (25), it could be gained as follows:

\[ \dot{V}_i = \hat{P}\dot{P}\dot{V}\dot{x} + \hat{P}^2V(x)\dot{V}(x) \]  

(26)

Put Eq. (23) and Eq. (24) into Eq. (26), then:
\[
\dot{V}_i = V(x)RT(\dot{P} - P)(\dot{m} - \dot{m})
\]  

(27)

Synthesizing the above studies, FIGURE 4 illustrates the block diagram of the overall control system with a SMC controller, based on a pressure observer.

![FIGURE 4. The Control scheme of SMC Controller with A Pressure Observer](image)

**EXPERIMENT AND ANALYSIS**

To verify the aforementioned control strategy, an experiment facility is established as FIGURE 5, basing on the schematic discussed in previous section of this paper.

![FIGURE 5. Experimental Facility for Motion Servo Control with PMA](image)

In this section, a series of experiments were carried out to analyze the differences of control effects with a pressure sensor or with a pressure observer in SMC control mode. FIGURE 6 shows a comparison of the observed pressures and the actual pressures in the PMA, while the system is tracking a frequency 0.25Hz and amplitude 20mm sinusoidal signal with a 2kg load. It demonstrates the values of the pressure in PMA recorded by the pressure sensor and the observer at the same time during the control process. As we can see, the data of the pressure observed are very close to the actual pressure inside the PMA, though there are some differences between them. The differences are mainly...
come from the inaccuracy of the model of mass flow passing the high speed solenoid on/off valve and the model of pressure observer. It can be improved by further studies in those fields.

![Graph](image)

**FIGURE 6.** Data of the Actual and observed pressure of the PMA when the system tracking a frequency 0.25Hz and amplitude 20mm sinusoidal signal with a 2kg load.

The performances of the system controlled with pressure sensor or observer in the experiments are show in **FIGURE 7** to **FIGURE 10.** In **FIGURE 7,** the system is tracking a frequency 0.25Hz and amplitude 20mm sinusoidal signal with a 2kg load, and (a) describes the results of the controller with a pressure sensor while (b) describes the results of the controller with a pressure observer. In **FIGURE 8,** the system is working under almost the same conditions except the frequency of the sinusoidal signal is enhanced to 0.5Hz.

![Graph](image)

**FIGURE 7.** (a) Tracking results and error of the system with a pressure sensor under the conditions of 0.25Hz sinusoidal signal and 2kg load. (b) Tracking results and error of the system with a pressure observer under the conditions of 0.25Hz sinusoidal signal and 2kg load.

![Graph](image)

**FIGURE 8.** (a) Tracking results and error of the system with a pressure sensor under the conditions of 0.5Hz sinusoidal signal and 2kg load. (b) Tracking results and error of the system with a pressure observer under the conditions of 0.5Hz sinusoidal signal and 2kg load.
In FIGURE 7, it can be seen that both systems work well with the high speed on/off solenoid valves under SMC control. The results obtained from the system using pressure sensor and the one using pressure observer demonstrates essentially the same tracking performance under the conditions of low frequency and low load. The precision of those results are close to the similar system which is controlled by proportional valve and the pressure is obtained from pressure sensor according to the experiment results in [2]. In FIGURE 8, although the frequency is enhances up to 0.5Hz, the performances of the system with pressure sensor or with pressure observer are as good as they under the lower frequency, while the errors rise a little. And the error gap of the motion control between the two systems is less than 7.5% of the whole amplitude in sinusoidal tracking. FIGURE 9 and FIGURE 10 test the performances of the systems under a larger load. They are working under 0.25Hz and 0.5Hz separately carry a 10Kg load.

In FIGURE 9, the results show that both of the system can also obtain good performances even with a much larger load under a low frequency, and the error of them are as lease as they are in 0.25Hz. But the results are not going well when the frequency is raised to 0.5Hz with the large load in FIGURE 10. Although the actual trajectories are close to the designed trajectory as before, the errors rise near 50%. The reason for those changes is that the flow control ability of the high-speed on/off solenoid valves is limited, as we analyzed. The PMA that carrying a larger load working at a higher frequency needs the pressure in the internal chamber of it changes much faster at a much larger
range. It requires a high capability of instant air flow supply of the control valves. Obviously, the valves that are used in this system cannot meet the demands of the task very well.

![Graphs showing displacement and error over time for different conditions.](image)

**FIGURE 11.** (a) Tracking results and error of the system with pressure sensor moving at 0.5Hz with 2kg load when an additional 2Kg load is added at 2.3s. (b) Tracking results and error of the system with pressure observer moving at 0.5Hz with 2kg load when an additional 2Kg load is added at 4.1s.

The robustness to external disturbances of the system was also verified as **FIGURE 11** shows. It presents the performance of the system when an additional 2Kg load was put on the top of the existing 2Kg load suddenly, while the system was in a stable motion state following a 0.5Hz and 20mm sinusoidal trajectory. The results in **FIGURE 11**(a) is come from the system with pressure sensor and the disturbance was added at the time 2.3s. And **FIGURE 11**(b) is come from the system with pressure observer and the disturbance was added at the time 4.1s. As we can see, the disturbance does cause a sudden error rise near 50% at the right time, but the lasting time of the influence is very short and the systems both return to the low error stable motion state very soon. It indicates that the system with pressure observer has a good robustness as the system with pressure sensor does, under the SMC controller.

**CONCLUSION**

In this paper, a sliding mode controller with a pressure observer is presented to accomplish a high precision motion control system driven by PMA. A nonlinear system model was derived based on modelling the PMA, high speed on/off solenoid valve, and kinetics of the system. And a sliding mode controller with pressure observer is designed after analyzing the state variables of the system carefully. Experiments were carried out to verify the performance of the controller. The experiment results show that both of the system with pressure sensor and pressure observer can work as good as the system driven by proportional valve in motion control accuracy and have good robustness. And the error gap of the motion control between the two systems is less than 7.5% of the whole amplitude in sinusoidal tracking. It demonstrates that the pressure observer can replace the pressure sensor in this case very well and provides a low cost option for industrial application.

**ACKNOWLEDGMENTS**

This paper is supported by the National Natural Science Foundation of China (Project No. 51521064) and the Natural Science Foundation of Zhejiang Province (Project No. LY13E050004).

**REFERENCES**

DEVELOPMENT OF PORTABLE REHABILITATION DEVICE USING FLEXIBLE EXTENSION TYPE SOFT ACTUATOR WITH BUILT-IN SMALL-SIZED QUASI-SERVO VALVE AND DISPLACEMENT SENSOR

So SHIMOOKA*, Shujiro DOHTA *, Tetsuya AKAGI *, Wataru KOBAYASHI*, Masataka YONEDA*

* Department of Intelligent Mechanical Engineering, Okayama University of Science
1-1 Ridai-cho, Kita-ku, Okayama 700-0005, Japan
(E-mail: t14rm01ss@ous.jp, {dohta, akagi, kobayashi}@are.ous.ac.jp)

Abstract. Today, a welfare pneumatic equipment to support a nursing care and to execute a rehabilitation for the elderly and the disabled are actively studied and developed by many researchers. The total weight of a wearable device increases according to the degree of freedom of the device. In this study, we proposed and tested a flexible extension type actuator with longer displacement. The maximum displacement of about 270 mm (235%) can be obtained. Using the actuator, the flexible robot arm was developed. As a result, we confirmed that the robot arm is able to extend straight and bend in any directions. An analytical model of the arm was proposed to predict the attitude of the arm. To control the actuator, the quasi-servo valve with built-in driving circuit using an embedded controller was also proposed and tested. In addition, the portable rehabilitation device using the robot arm, three quasi-servo valves and three displacement sensors was developed.

Keywords: Flexible extension type actuator, Built-in quasi-servo valve, Rehabilitation device, Wire type liner potentiometer

INTRODUCTION

Today, a welfare pneumatic equipment to support a nursing care and to execute a rehabilitation for the elderly and the disabled are actively researched and developed by many researchers[1-4]. The purpose of our study is to develop a home rehabilitation device that includes a controller, valves, sensors and actuators. In such a device, the total weight of a wearable device increases according to the degree of freedom of the device. Therefore, to decrease the burden of the user, a small-sized and light-weight pressure control type quasi-servo valve was developed in our previous study[5-6]. In the next step, we aim to develop a portable rehabilitation device with larger moving area which can give passive exercise for human shoulder[7]. To realize such a device with a compact configuration, it is necessary to develop an extension type actuator with a longer stroke[8]. In addition, to install a whole pneumatic driving system into the device, a built-in servo valve and flexible displacement sensor are also required[9-10]. In this paper, we propose and test a flexible extension type actuator with longer displacement. As a result, the maximum displacement of about 270 mm, that is more than 200% extension from original length, can be obtained when the input pressure of 400 kPa is applied. To control the actuator, the flow rate control type quasi-servo valve with built-in embedded controller is proposed and tested. Figure 1 shows an image of the proposed portable rehabilitation device. As shown in Fig. 1, we aim to develop a portable rehabilitation device giving a force to human arm and shoulder such as an expander and bender. The construction and operating principle of the device is described.

FIGURE 1. Image of the proposed portable rehabilitation device
EXTENSION TYPE SOFT ACTUATOR

Figure 2 shows the view and schematic diagram of the proposed flexible extension type actuator. The tested actuator consists of a rubber tube covered with a ruffled fabric sleeve. The rubber tube has an inner diameter of 6 mm, outer diameter of 9.5 mm and length of 200 mm. The original length of the ruffled fabric sleeve in the stretched condition is 450 mm. Figure 3 shows the relation between supply pressure and displacement of the actuator. In the experiment, the actuator was pressurized from 0 to 400 kPa every 20 kPa by using a pressure regulator. The measurement was carried out for three times. The maximum displacement of the actuator of about 270 mm, that is 235\% extension of original length, can be obtained when the input pressure of 400 kPa is applied. It is also found that the hysteresis which is caused by the friction between the tube and the sleeve can be observed. In order to reduce of the influence of hysteresis, it is necessary to execute a position feedback control using a flexible displacement sensor with a long stroke. The pushing force of the actuator while being pressurized is small because of its flexibility. However, the pulling force in case of decompression is large. It is an elastic force of the rubber tube. The maximum of pulling force is about 40 N.

FIGURE 2. View and schematic diagram of tested flexible extension type soft actuator

FIGURE 3. Relation between supply pressure and displacement of the actuator

FLEXIBLE ROBOT ARM

As a rehabilitation device with a wider moving area, a flexible robot arm using tested actuators as shown in Fig. 4 is proposed and tested. The robot arm consists of three extension type actuators with the original length of 200 mm arranged every 120 degrees at 30 mm from the center of the device. Both ends of the actuator are fixed with a triangle-shaped plastic plate. The device has 40 thin plates with the width of 1 mm to keep a parallel arrangement of three actuators and bending stiffness of the robot arm. The size of the robot arm is 230 mm in length, 90 mm in width and 90 mm in height. The mass of the arm is 420 g. Figure 5 shows the movement of the arm when three actuators are pressurized. From Fig. 5, we can observe that the robot arm can bend in any directions by decompressing one or two actuators in three actuators.

FIGURE 4. Overview of movement of flexible robot arm
In order to design and control the robot arm, an analytical model which can predict the characteristics of the robot arm is required. Figure 6 shows the analytical model of the flexible robot arm[11]. As shown in Figs. 6 (b) and (c), the flexible extension type actuator which is on X axis is defined as “actuator 1” and the other actuators arranged in a counter clockwise direction are defined as “actuator 2” and “actuator 3”, respectively. \( L_1, L_2, \) and \( L_3 \) are the actuator length for actuator 1, actuator 2 and actuator 3, respectively. From the geometric relationship as shown in Figs. 6(b) and (c), the following equations can be obtained.

\[
L_1 = (R - r \cdot \cos \alpha) \cdot \beta, \quad (1) \\
L_2 = (R - r \cdot \cos \left(\frac{2\pi}{3} - \alpha\right)) \cdot \beta, \quad (2) \\
L_3 = (R - r \cdot \cos \left(\frac{4\pi}{3} - \alpha\right)) \cdot \beta \quad (3) \\
R = \frac{L}{\beta}, \quad (4)
\]

where \( L \) means the central length of the robot arm, \( R \) is the radius of curvature of the arm, and \( r \) which is 30 mm is the distance between the center of the arm and the center of each actuator. \( \alpha \) and \( \beta \) mean the bending direction angle and bending angle, respectively. By using Eqs. (1) to (4), the central length of the robot arm \( L \), the bending direction \( \alpha \) and the bending angle \( \beta \) can be expressed as follows.

\[
L = \frac{L_1 + L_2 + L_3}{3}, \quad (5) \\
\alpha = \tan^{-1} \left( \frac{\sqrt{3}(L_3 - L_2)}{L_2 + L_3 - 2L_1} \right), \quad (6) \\
\beta = \frac{L - L_3}{r \cdot \cos \alpha}. \quad (7)
\]

When \( \cos \alpha = 0 \), the bending angle \( \beta \) can be obtained by the following equation from Eqs. (2) and (3).

\[
\beta = \frac{|L_3 - L_2|}{\sqrt{3}r}. \quad (8)
\]

From Eqs. (4) and (7), the radius of curvature of the arm \( R \) is given by

\[
R = \frac{L \cdot r \cdot \cos \alpha}{L - L_1}. \quad (9)
\]

When each actuator is pressurized, each length of the actuator \( L_i \) is given by

\[
L_i = L_{0i} + \frac{A(P_i - P_{\text{min}})}{k}, \quad (10)
\]

where \( L_{0i}, P_i, P_{\text{min}}, k \) and \( A \) mean the initial length of each actuator, supply pressure, minimum supply pressure (160 kPa), the elastic coefficient and the sectional area of the tube in an actuator, respectively.

Next, the calculated posture of the arm using above equations was compared with the experimental one. Figure 7 shows the comparison of the arm shape when the following pressure was given for each actuator. The supply
pressure $P_1$, $P_2$ and $P_3$ are 200 kPa, 400 kPa and 400 kPa, respectively. Figures 7(a) and (b) show the experimental result and calculated one, respectively. When the elastic coefficient $k$ is 345 N/m, the bending angle $\beta$ of 70 from Fig. 7(b) agrees with the experimental result. Figure 8 shows the relation between differential pressure and bending angle of the arm. The differential pressure means the pressure difference between $P_1$ and $P_2$ or $P_3$. The symbols show the experimental results and the lines show the calculated results. From Fig. 8, it is found that the calculated result does not agrees with the experimental result when the differential pressure is higher than 250 kPa. This is because the elastic coefficient $k$ changes with the supply pressure as shown in Fig. 3.

![Analytical model of robot arm](image1)

![Definition of actuator length](image2)

![Definition of angles $\alpha$ and $\beta$](image3)

**FIGURE 6.** Analytical model of robot arm

![Experimental result](image4)

![Calculated result](image5)

**FIGURE 7.** Comparison of robot arm posture

![Relation between differential pressure and bending angle](image6)

**FIGURE 8.** Relation between differential pressure and bending angle

**QUASI-SERVO VALVE BUILT-IN EMBEDDED CONTROLLER**

Figure 9 shows the construction and the schematic diagram of the quasi-servo valve that we developed[12]. It consists of two standard on/off type control valves and an embedded controller. The output port of the first valve is connected to the input port of second one. The first valve is a three-port valve that can change the direction of fluid flow from the supply port to the output port or the fluid flow from the output port to the exhaust port. We call it a “switching valve”. The second valve is a two-port valve driven by PWM(pulse width modulation) method in order to adjust the valve opening per time. The PWM valve can adjust output flow rate like a variable
fluid resistance. In order to decrease the size and cost of the valve, the smaller-sized on/off control valve (SMC Co. Ltd., S070C-SDG-32) was used. Compared with the previous on/off control valve[6], the price of new valve is about a half, that is about 17 US dollars. The maximum flow rate of the valve is 8.5 liter/min when the supply pressure of 400 kPa is applied. This value of flow rate is enough to drive the flexible extension type actuator. As shown in Fig. 9, to realize the compact construction of connector between two on/off valves, the acrylic flow passages were used. The size of the valve without typical tube connector is 36×25×17 mm. The mass of the valve is only 24 g.

FIGURE 9. Construction and schematic diagram of quasi-servo valve

Figure 10 shows the construction and the schematic diagram of the tested flow rate control type quasi-servo valve with a built-in driving circuit using an embedded controller. The driving circuit consists of a tiny embedded controller (Renesas Co. Ltd., RL78/G10) and two transistors (Fairchild Semiconductor Co. Ltd., TO-92 2N7000). The flow rate control is done as follows. First, the embedded controller can get an analogue signal through A/D converter on the embedded controller. In the embedded controller, the input duty ratio is calculated based on the empirical formula[13]. The required input duty ratio is given through PWM port of the controller.

FIGURE 10. Construction and schematic diagram of the tested flow rate control type quasi-servo valve with built-in driving circuit using an embedded controller

Figure 11 shows the relation between the valve opening and the output flow rate of the tested valve. In the experiment, to measure the output flow rate based on the empirical formula[13] and to observe control parameters such as input normalized flow rate and output duty ratio, the embedded controller (Renesas Co. Ltd., SH7125) that has several serial communication ports was used. From the result shown in Fig. 11, the relation between the valve opening and output flow rate has a linear relationship. It means that the tested flow rate control type quasi-servo valve can change the sectional area of the valve linearly according to the control input.

FIGURE 11. Relation between input valve opening and output flow rate
PORTABLE REHABILITATION DEVICE

Figure 12 shows the overview of the control system of the portable rehabilitation device. The system consists of the tested robot arm, three quasi-servo valves with a built-in embedded controller, three wire type linear potentiometers[14] and a micro-computer (Renesas Co. Ltd., SH7125) for control the rehabilitation device. Even if the actuator bends, the sensor can measure the displacement of the actuator because the wire can move along to shape of the actuator through holes on thin plates. This sensor can also measure the long stroke of 210 mm. Figure 13 shows the schematic diagram of the control system of the portable rehabilitation device. The position control is done as follows. First, the wire type linear potentiometers measure each displacement of the actuators. The output value is taken by A/D converter in the micro-computer. The error between the desired and measured displacement is calculated by the micro-computer for control. The control input for the quasi-servo valve is also calculated based on the simple P control scheme. The control input for each valve is given as an analog signal from 0 to 5 V though D/A converter with SPI controller (Linear Technology Co. Ltd., LTC1660CN). In the quasi-servo valve, the built-in controller can control the valve opening (output flow rate) according to the input voltage based on the empirical formula as shown in Fig.11. In the case that the input voltage for the valve is higher than 2.5 V, the valve works as a supply valve. In the opposite case (lower than 2.5 V), the valve works as an exhaust valve. In the case of 2.5 V, the valve is closed. The valve opening is controlled according to the absolute value form 2.5 V. By this method, the attitude of the device is controlled. In the experiment, the reference displacement of 200 mm is given for each actuator. Then, the target value of 50 mm for the actuator is changed every 10 seconds.

Figure 14 shows the operation of attitude control for the portable rehabilitation device. Figure 15 shows the transient response of the displacement of the actuator. In the Fig.15, red line and blue line show the controlled displacement of the actuator and reference, respectively. It can be found that the controlled displacement agrees with the reference. This means that the device can follow up the target attitude well. In future work, we are going to develop a device which can give passive motions to patients who hold the device by hands like an active bender and expander.
CONCLUSIONS

As a rehabilitation device with moving area, the portable rehabilitation device using extension type actuators was proposed and tested. As a result, we confirm that the tested actuator extends until more than 200% of original length. The flexible robot arm using the proposed actuators was proposed and tested. Also, in order to design and control the robot arm, the analytical model was proposed. As a result, it is confirmed that the calculated result does not agree with the experimental result when the supply pressure become higher. This is become the elastic coefficient change with the pressure. However, the calculated result shows the effect of the radius $r$ on the bending angle well. To control the actuator, the flow rate control type quasi-servo valve was proposed and tested. The quasi-servo valve with built-in driving circuit using an embedded controller was developed. As an application, the portable rehabilitation device was tested. Also, in order to measure the displacement of the actuator, the wire type linear potentiometer was used in the device. As a result, it was confirmed that the controlled displacement agrees with reference. As future work, we will develop the device which can give passive motion to patients who hold the device by hands like an active bender and expander.
ACKNOWLEDGMENTS

This study was supported by the Education Ministry of Japan through a Financial Assistance Program for QOL Innovative Research (2012-2016) and a Grant-in-Aid for Scientific Research (C) (Subject No. 24560315 & 16K06202).

REFERENCES

Oral Presentation | Pneumatics

Chair:Tetsuya Akagi(Okayama University of Science), Mitsuhiro Nakao(Kagoshima University)
Thu. Oct 26, 2017 3:30 PM - 4:50 PM Room B (ACROS Fukuoka)

[2B12] PERFORMANCE EVALUATION OF SUPPORTING ARM FOR REDUCING BODY LOAD USING SURFACE ELECTROMYOGRAPHY
*Tetsuro Miyazaki¹, Takuya Iijima², Yuuichi Hirahara², Kazushi Sanada² (1. Tokyo Medical and Dental University, 2. Yokohama National University)
3:30 PM - 3:46 PM

[2B13] A HUMAN-MACHINE COOPERATION CONTROL BASED ON ELECTROMYOGRAPHY FOR UPPER LIMB POWERED EXOSKELETON DRIVEN BY PNEUMATIC MUSCLE
*Jun Tao¹, Hao Liu¹ (1. State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University)
3:46 PM - 4:02 PM

[2B14] EVALUATION OF AIR COMPRESSING METHODS FOR DEVELOPMENT OF A PORTABLE PNEUMATIC POWER SOURCE
*Manabu Okui¹, Yuki Nagura², Shingo Iikawa¹, Yasuyuki Yamada², Taro Nakamura² (1. Graduate School of Science and Engineering, Chuo University, 2. Faculty of Science and Engineering, Chuo University)
4:02 PM - 4:18 PM

[2B15] WRIST REHABILITATION SIMULATOR FOR P.T. USING PNEUMATIC PARALLEL MANIPULATOR (Regulation of Wrist Viscoelastic Property and Therapy Motion Evaluation)
*Masahiro Takaiwa¹, Hiroyuki Imanaka¹ (1. Tokushima University)
4:18 PM - 4:34 PM

[2B16] DEVELOPMENT OF TENDON-DRIVEN CARE ASSISTANCE ROBOT ARM DRIVEN BY AIR PRESSURE CONTROLLING
*Daichi Kimura¹, Osamu Oyama² (1. first year master's student who belongs to Professor Oyama's laborator, 2. Meiji University)
4:34 PM - 4:50 PM
PERFORMANCE EVALUATION OF SUPPORTING ARM FOR REDUCING BODY LOAD USING SURFACE ELECTROMYOGRAPHY

Tetsuro MIYAZAKI*, Takuya IIJIMA**, Yuuichi HIRAHARA***, Kazushi SANADA****

* Institute of Biomaterials and Bioengineering, Tokyo Medical and Dental University
2-3-10 Kanda Surugadai, Chiyoda-ku, Tokyo 101-0062 Japan
** Graduate School of Engineering, Yokohama National University
79-5 Tokiwadai, Hodogaya-ku, Yokohama-shi, Kanagawa 240-8501 Japan
*** School of Engineering, Yokohama National University
79-5 Tokiwadai, Hodogaya-ku, Yokohama-shi, Kanagawa 240-8501 Japan
**** Faculty of Engineering, Yokohama National University
79-5 Tokiwadai, Hodogaya-ku, Yokohama-shi, Kanagawa 240-8501 Japan

Abstract. This paper proposes a supporting arm which is worn by a factory worker for reducing the worker’s body load, and performance of the supporting arm was evaluated by several experiments. The whole mechanism of the supporting arm contains two leg mechanisms for supporting the load and one external frame backpack for connecting the worker’s body and the supporting arm. The supporting arm was worn by several subjects, and its supporting effect was evaluated by measuring electromyogram (EMG) waveforms of leg muscles. When the worker takes half-sitting posture, its leg load was measured in two cases: (i) a part of the worker’s body load is supported by the supporting arm, and (ii) the whole worker’s body load is supported only by the worker’s legs. The EMG waveforms of these cases were compared, and as a result, the supporting effect of the supporting arm was demonstrated.

Keywords: Supporting arm, Power assist robot, Factory worker support, Pneumatic cylinder, Electromyogram

INTRODUCTION

Power assist robots, which support human activities, have been developed in previous researches. For example, there is an exoskeleton type power assist robot [1]. The exoskeleton type power assist robot is attached to human body limbs, and it supports human joint torques by generating actuator forces along with the human motion. BLEEX (Berkeley Lower Extremity Exoskeleton) [2, 3] was developed for enhancing carrying-capacity of soldiers with heavy equipment, and this robot is driven by hydraulic actuators. Sankai [4] developed a robot suit HAL (Hybrid Assistive Limb) targeted for medical welfare applications and support for heavy object handling. Kobayashi et al. [5] developed a muscle suit driven by an artificial muscle, and evaluated muscle fatigue by measuring oxygen concentration in the blood in order to evaluate the assist effect [6]. However, these exoskeleton type power assist robots are designed to follow the human limb motion. Therefore, when the human body is forced to maintain an unstable posture, it will sometimes become difficult to support the human body.

Also, as a different type robot from the exoskeleton type power assist robot, an environment type power assist robot has been proposed. For example, a power assist chair having a virtual spring characteristic that supports the standing motion of a human has been developed [7, 8]. Okazaki et al. [9] developed a power assist arm using large pneumatic muscles, that conveys heavy objects with a worker at a production site. In these type robots, the environmental side becomes a power assist robot, so that human is not required to wear any equipment on its body, and it is one of the effective strategies. However, since the power assist robot is fixed to the environment side, it will be difficult to assist tasks accompanying with human transfer like walking around.

In addition, an additional body type power assist robot has been also developed, whose robotic parts can move separately from the human limbs. For example, a wearable robotic finger [10] was developed for the purpose of supplementing the gripping function of the human hand. A crutch-type walking assist machine was developed for the purpose of assisting the walking of the lower limb-handicapped person having healthy upper limbs [11].
A supernumerary robotic limb (SRL) [12, 13] was developed. The SRL grips a work or an environment by its hands, and it supports a factory worker task. By using the additional leg-type robot like the SRL, the human body will be stabilized and prevented falling accidents. Since the additional body type power assist robot is a type to be worn on the human body, work assistance is possible even when the human transfers. However, from a safety and an efficiency point of view, the additional body type assist robot and the human limbs need to cooperate with each other, and the robot motion in each task needs to be designed.

A design and control method of a supporting arm which reduces factory worker load was proposed [14]. The supporting arm is an additional body type assist robot, which is driven by pneumatic cylinders. In some situation, the factory worker is forced to work with an uncomfortable posture. In this case, the supporting arm undertakes a part of the worker load. By using the supporting arm, the worker leg loads are relaxed, and the worker posture is stabilized. However, from a safety and an efficiency point of view, the additional body type assist robot and the human limbs need to cooperate with each other, and the robot motion in each task needs to be designed.

In this paper, a whole mechanism of a supporting arm is manufactured. The supporting arm is worn by several subjects, and its supporting effect is evaluated by measuring electromyogram (EMG) waveforms of leg muscles. When a muscle generates large force, the EMG waveform will also become large. The EMG waveforms are measured by a surface electromyography, and the leg load will be measured quantitatively. When the worker takes half-sitting posture, its leg load is measured in two cases: (i) a part of the worker’s body load is supported by the supporting arm, and (ii) the whole worker’s body load is supported only by the worker’s legs. The EMG waveforms of these cases are compared, and the supporting effect of the experimental system is validated.

**NOMENCLATURE**

- $d$: Foot position [m]
- $e$: Time series electromyogram (EMG) data [V]
- $F_c$: Cylinder output force [N]
- $F_s$: Support force [N]
- $g$: Gravitational acceleration [m/s$^2$]
- $h$: Hip height [m]
- $h_{ref}$: Target hip height [m]
- $K_p$: Proportional gain [N/m]
- $l$: Parameter of left-side arm of worker's body
- $M$: Mass of load to be borne by supporting arm [kg]
- $M_0$: Target mass [kg]
- $n$: Sample number using in t-test
- $\text{RMS}$: Root mean square (RMS) value of EMG data [V]
- $\text{RMS}_\text{MVC}$: RMS value of maximum voluntary contraction (MVC) state [V]
- $r$: Parameter of right-side arm of worker's body
- $T$: Time interval for calculating the RMS value [s]
- $T_v$: Test statistic of t-test
- $\%\text{MVC}$: RMS value normalized by that of MVC state
- $\Delta\%\text{MVC}$: Difference of $\%\text{MVC}$ mean values between the case with support and with no support
- $\Delta\%\text{MVC}$: Mean value of $\Delta\%\text{MVC}$
- $\sigma_{\Delta\%\text{MVC}}$: Standard deviation of $\Delta\%\text{MVC}$
- $\theta_1$: Angle of joint 1 [rad]
- $\theta_2$: Angle of joint 2 [rad]

**SUPPORTING ARM FOR REDUCING BODY LOAD**

**Link System and Leg Mechanism of Supporting Arm**

In [14], a link system and leg mechanism of the supporting arm were proposed, and these are utilized for manufacturing a whole mechanism of the supporting arm in next section. The link system of the supporting arm is shown in **Fig. 1 and 2**. In **Fig. 1**, left figure A represents the worker taking half-sitting posture. This posture forces the worker leg and hip joints to generate large torques, and the worker will feel uncomfortable. On the other hand, right figure B represents the worker using the supporting arm. In this case, the supporting arm undertakes a part of the worker load. By using the supporting arm, the worker leg loads are relaxed, and the...
worker posture is stabilized. In Fig. 2, a horizontal view of the supporting arm is shown. This link system contains an L-shaped link and two leg mechanisms which are represented as an arm 1 and 2 enclosed in dashed line. It is assumed that frictions between the supporting arm and floor are sufficient, and the supporting arm will not slip on the floor. In order to prevent slip, non-slip rubber sheets made of nitrile rubber are attached to the sole of the supporting arm. In Fig. 2 left, a workspace of the arm 1 is limited on a plane parallel to x-y plane. In the same way, a workspace of the arm 2 is limited on a plane parallel to y-z plane. By connecting the arm 1 and arm 2 with the L-shaped link, the degrees of freedom of the whole mechanism of the supporting arm becomes one. Therefore, the working space of the L-shaped link is limited to the worker hip height direction, which is shown as the y-axis direction in Fig. 2. The L-shaped link is fixed on the worker hip.

The leg mechanisms which are represented as the arm 1 and 2 in Fig. 2 are designed and manufactured as shown in Fig. 3. In Fig. 3, left figure represents the manufactured leg mechanism, and right figure represents the link system of the leg mechanism. This link system becomes a closed-loop six link system, which contains five rotational joints and two linear joints. The link 4 and 5 are a rod and cylinder part of a pneumatic cylinder. The joint 5 will be actuated by pneumatic pressure, and all other joints are passive joints. The joint angle 1 and 2 are measured by potentiometers. The link 3 is assumed as a part of the L-shaped link, and a weight, which is simulated the human body load, is fixed on the link 3. A motion of the link 3 is limited on y-direction by using a linear guide. The link 0 is a foot-like link that contacts with the floor and is assumed as a static link that will not slip on the floor by the static frictional force. In this paper, assuming that the worker model is 1.7 m in height and 70 kg in weight, the hip height when the worker takes a half-sitting posture is about 0.7 m. These parameters are almost same value to standard Japanese adult male. The target load supported by the supporting arm is 35 kg corresponding to the half weight of the worker model, and this load is supported by the two leg mechanisms. In this case, the target load per the leg mechanism is 17.5 kg. The pneumatic cylinder and link system are selected and designed to satisfy that requirement. The manufactured system in Fig. 3 was evaluated its supporting performance in [14]. In the weight lifting experiment, it is confirmed that the desired support force and stiffness characteristics can be obtained by using the leg mechanism.

Manufacturing Whole Mechanism of Supporting Arm

A whole mechanism of the supporting arm is manufactured as shown in Fig. 4. The whole mechanism of the supporting arm contains two leg mechanisms for supporting the load and one external frame backpack for connecting the worker’s body and the supporting arm. The external frame backpack is often used for mountain climbers to carry their items, and its load bearing capacity is 50 kg. In this paper, a target weight of the supporting arm is set as 35 kg, and the external frame backpack can support the target weight. A total weight of the supporting arm is 16.6 kg. The manufactured mechanism is a prototype for verifying that the desired load supporting effect can be obtained by the designed mechanism. As a result of design the mechanism to have
sufficient strength, the mass of the mechanism increased. However, because it is desired to a lightweight mechanism to carry the supporting arm, we will also work on the lightweight design of the mechanism.

**FIGURE 4.** Developed supporting arm (left : side view, right : rear view).

**Control of Support Force and Stiffness Characteristic**

The illustration of the supporting arm undertaking the worker’s body load is shown in **Fig. 5**. In **Fig. 5**, the subscript $R$ represents the parameter of the right-side arm of the worker’s body, and the subscript $L$ represents the parameter of the left-side arm of the worker’s body. $F_s$ is the support force, $M$ is the mass of the load to be borne by the supporting arm, $g$ is the gravitational acceleration, $h$ is the hip height, $d$ is the foot position, and $F_c$ is the cylinder output force. The support forces $F_{sR}$ and $F_{sL}$ are controlled using a two degrees of freedom control system so as to satisfy the following two requirements: (i) support force requirement and (ii) stiffness characteristic requirement. The target load is supported by the output of the feed forward controller, and the desired stiffness characteristic is given by an adjustable proportional gain of the feedback controller.

In this paper, we use the method of the paper [14] and control the supporting arm so as to satisfy the requirements of support force and stiffness characteristic when supporting the target load with two arms as shown in **Fig. 5**. The control method is described as follows.

**FIGURE 5.** Illustration of supporting worker load.  
**FIGURE 6.** Two degrees of freedom control system of supporting arm.

### Support Force Requirement

Target mass $M_0$ of the load borne by the supporting arm is given and will be supported by the support forces $F_{sR}$ and $F_{sL}$ of the left and right arms which are outputs of the feedforward controller as:

$$\begin{align*}
F_{sR} + F_{sL} &= M_0 g, \text{ and} \\
F_{sR} &= F_{sL}.
\end{align*}$$

Eq. (1) is an equilibrium condition of translational forces in the hip height direction, and Eq. (2) is an equilibrium condition of rotational forces for not giving the rotational moment to the worker by the support forces. From Eq. (1) and (2), the support forces of the right and left arms are as:

$$F_{sR} = F_{sL} = M_0 g / 2.$$
However, in practice, the load that the supporting arm undertakes from the worker is not always constant, and it will be considered to change during work. Since the output of the feedforward controller cannot adapt to the change of the load, the output of the feedback controller described in the next section is added.

**Stiffness Characteristic Requirement**

The load that the supporting arm undertakes from the worker body changes when the worker moves or receives an external force. In these case, it is desirable to be able to give appropriate stiffness to the support force for each work situation. For example, when the worker actively changes the posture, the worker should be supported with soft stiffness, and when the worker takes a certain posture to concentrate on the work, the worker should be supported with hard stiffness. Therefore, as the stiffness characteristic requirement, the output of the feedback controller is added to the support forces \( F_{sR} \) and \( F_{sL} \) in the Eq. (3) as:

\[
F_{sR} = F_{sL} = M_0 g / 2 + K_P (h_{ref} - h) / 2 .
\]

Where \( K_P \) is an adjustable proportional gain, and \( K_0 / 2 \) is a stiffness itself in the waist height direction of the supporting forces \( F_{sR} \) and \( F_{sL} \). \( h_{ref} \) is a target hip height. Finally, the target mass \( M_0 \) will be supported by the support forces of the two arms as:

\[
F_{sR} + F_{sL} = M_0 g + K_P (h_{ref} - h) .
\]

and the stiffness characteristic during the worker load support becomes \( K_P \). When allowing changes in the worker posture, \( K_P \) will be set to zero or a small value. On the other hand, when fixing to a certain posture, \( K_P \) will be set to a large value.

**Control System of Supporting Arm**

A two degrees of freedom control system shown in Fig. 6 is used to control the supporting arm. The control sequence in Fig. 6 is described as follows. Firstly, the target mass \( M_0 \), the proportional gain \( K_P \) and the target hip height \( h_{ref} \) are given, and the support forces \( F_{sR} \) and \( F_{sL} \) are calculated using the controller of the Eq. (4) so as to reduce the error between the actual hip height \( h \) and \( h_{ref} \). The mechanism of the supporting arm can be calculated static analysis by considering the left and right arms as independent planar link mechanisms as shown in [14]. In this paper, support force control, static analysis and forward kinematic analysis are calculated on each of the left and right arms using the leg mechanism model shown in Fig. 3. The hip height \( h \) used for the support force control is calculated as \( h_R \) and \( h_L \) by forward kinematic analysis on each of the left and right arms. Secondary, the cylinder output forces \( F_{cR} \) and \( F_{cL} \) required gaining the support forces \( F_{sR} \) and \( F_{sL} \) at the current hip heights \( h_R \) and \( h_L \) are calculated by static analysis of the leg mechanism model. In order to gain the desired cylinder output forces \( F_{cR} \) and \( F_{cL} \), the cylinder pressures are controlled using pressure control valves. \( F_{cR} \) and \( F_{cL} \) are controlled, and the supporting arm changes its posture. The angle \( \theta_1 \) of the joint 1 and the angle \( \theta_2 \) of the joint 2 are measured by the potentiometers on each arm, and the current hip heights \( h_R \) and \( h_L \) are calculated by forward kinematic analysis using these angle values. While updating the hip heights \( h_R \) and \( h_L \) in the control loop of Fig. 6, the control operation is performed at a constant cycle so as to approach the target hip height \( h_{ref} \).

**PERFORMANCE EVALUATION OF SUPPORTING ARM USING SURFACE ELECTROMYOGRAPHY**

**Measuring Procedure of Surface Electromyography**

A supporting effect of the supporting arm is evaluated by measuring electromyogram (EMG) waveforms of leg muscles. Recording electrodes of the surface electromyography are attached on the leg muscles as shown in Fig. 7. The surface electromyography has four channels (CH). CH 1 and CH 3 are attached on lateral great muscles. CH 2 and CH 4 are attached on anterior tibial muscles. GND electrodes are attached on insteps. The human leg muscles have many categories, and many channels are necessary to simultaneously measure all these EMG waveforms. In this paper, in order to show the effect of reducing the body load from the EMG measurement data of small number channels, the lateral great muscles of the quadriceps femoris muscle and the anterior tibial muscles of the shins are selected as measuring parts, because these muscles will output large forces when the
worker is maintaining the half-sitting posture. Another reason of the selection is easiness of attaching recording-electrodes. The measurement of the EMG data is performed for each subject by the flowchart as shown in Fig. 8. Firstly, subject’s EMGs are measured in a relaxed state which is sitting on the chair. Secondary, when the subject using the supporting arm is taking a half-sitting posture, the subject’s EMGs are measured. Since it is considered that the target hip height \( h_{ref} \), the target mass \( M_0 \) and the proportional gain \( K_p \) used for control will be different from each subject, these values are adjusted for each subject so as to feel comfortable for maintaining the half-sitting posture. Thirdly, when the subject without support is taking a half-sitting posture, the subject’s EMGs are measured. After that, EMGs with the maximum voluntary contraction (MVC) state are measured, and the subject takes a break about 10 minutes. In this paper, the MVC state is in which the subject taking a half-sitting posture holds a total 20 kg iron weight on both hands and consciously outputs the leg forces. In these procedures, the measurements are made 5 s each. These procedures are repeated several times per each subject. From the obtained EMG data, root mean square (RMS) values are calculated as one of indicator of voltage magnitude. The larger the RMS value, the more muscle fibers are about to contract, and the muscles are trying to generate a large force. The RMS value is calculated as:

\[
RMS = \sqrt{\frac{1}{T} \int_{0}^{T} e^2(t) \, dt}.
\]  

(6)

where \( T \) is a time interval for calculating the RMS value, and \( e \) is a time series EMG data. After obtaining the RMS values in each channel and each trial, these values are normalized by the RMS values of the MVC state, and these normalized values are called %MVC values. The %MVC values are calculated as:

\[
%MVC = \frac{RMS_{ij}}{RMS_{MVCij}}.
\]  

(7)

Where the subscript \( i \) represents the number of CH, and \( j \) represents the \( j \)-th data of the measurement loop of Fig. 8. \( RMS_{MVC} \) is the RMS value of the MVC state. The RMS value \( RMS_i \) of each channel is expressed as a ratio to the RMS value \( RMS_{MVCj} \). By using this normalization, it becomes possible to compare the %MVC values between the different test muscles or between the different test subjects.

![FIGURE 7. Recording-electrode positions.](image)

![FIGURE 8. EMG measuring flowchart.](image)

**Validation of Supporting Effect for Several Different Workers**

The supporting effect of the supporting arm was verified by experiments with several different subjects. The experiments were executed three times for each of four subjects by the procedure of Fig. 8. Experimental results of each subject are shown in Fig. 9, and body parameters and experimental settings of each subject are shown in Table 1. In Fig. 9, the vertical axis represents %MVC values which are calculated from three set trials in each channel. The horizontal axis represents the electrode channel numbers. Bold bars represent the %MVC average values in each measuring situation. Blue bars are results of the relaxed state, white bars are results with support, and gray bars are results without support. Thin black error bars represent the standard deviations.

Firstly, an experimental result of a subject 1 shown in Fig. 9 upper left is discussed. As shown in Table 1, a height and weight of the subject 1 are 1.70 m and 66 kg, and the target mass \( M_0 = 30 \) kg (45% of the body weight) was supported by the supporting arm. The proportional gain \( K_p \) was set as 2000 N/m. All white bars are less than gray bars, therefore the body load of the subject 1 seems to be relaxed by using the supporting arm. It is verified that whether there is a significant difference in the %MVC average values between the case with support and no support by the t-test. The samples used for the t-test are twelve sets of experimental data measured three trials on the four channels recording electrodes, and each set includes the data of the case with...
support and no support. The null hypothesis is defined as that there is no support effect of the supporting arm, that is, there is no significant difference in the \%MVC average values between the case with support and no support. And the alternative hypothesis is defined as that there is a support effect of the supporting arm. The test statistic $T_v$ is calculated from the experimental data as:

$$ T_v = \frac{\Delta \%MVC}{\sigma_{\Delta \%MVC} / \sqrt{n}}. $$  \hspace{1cm} (8) 

Where $\Delta \%MVC$ is the difference of the \%MVC average values between the case with support and with no support obtained for each data set. $\overline{\Delta \%MVC}$ and $\sigma_{\Delta \%MVC}$ are the average value and standard deviation of $\Delta \%MVC$, and $n$ is the sample number. The test statistic $T_v$ obtained from the experimental data of the subject 1 was 6.88. In this paper, for each of the four subjects, the t-tests were performed with a significance level of 5% on one side test. In this case, the reference value of the twelve data sets is 1.80 from the t distribution. Since the test statistic $T_v$ of the subject 1 is larger than the reference value, the null hypothesis is rejected. Therefore, there is a significant difference of the \%MVC average values between the case with support and without support.

In the same manner, experimental results of other three subjects were also verified by the t-test. The test statistic $T_v$ obtained from the data of the subject 2, 3 and 4 was 4.12, 7.17 and 3.76. These values are larger than the reference value 1.80, therefore there are significant differences of the \%MVC average values between the case with support and without support in all subjects. It was verified that the load reduction effect by the supporting arm is effective for several different workers. As a result of the four subjects, the \%MVC average values of all four channels became smaller when there was support than when without support.

**FIGURE 9.** \%MVC values of 4 subjects in each electrode channel and each measuring situation. Figure (1) to (4) represent results of subject 1 to 4. The vertical axis represents %MVC average values and standard deviations which are calculated from 3 set trials in each channel. The horizontal axis represents electrode channel numbers. Bold color bars represent the \%MVC average values in each measuring situation (blue: relaxed state, white: with support, gray: no support). Thin black error bars represent the standard deviations. All white bars are less than gray bars, therefore the body load of the subject 1 to 4 seems to be relaxed by using the supporting arm.

**TABLE 1.** Body parameters and experimental settings of each subject.

<table>
<thead>
<tr>
<th>Subject</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Height [m]</td>
<td>1.70</td>
<td>1.80</td>
<td>1.72</td>
<td>1.73</td>
</tr>
<tr>
<td>Body Weight [kg]</td>
<td>66</td>
<td>70</td>
<td>70</td>
<td>57</td>
</tr>
<tr>
<td>Target Hip Height [m]</td>
<td>0.70</td>
<td>0.70</td>
<td>0.70</td>
<td>0.75</td>
</tr>
<tr>
<td>Target Mass [kg] (% of Body Weight)</td>
<td>30 (45 %)</td>
<td>25 (36 %)</td>
<td>30 (43 %)</td>
<td>30 (53 %)</td>
</tr>
</tbody>
</table>
CONCLUSION

This paper proposed the supporting arm which is worn by the factory worker for reducing the worker’s body load, and performance of the supporting arm was evaluated by several experiments. The main results are as:

1. The whole mechanism of the supporting arm was manufactured which contains two leg mechanisms for supporting the load and one external frame backpack for connecting the worker’s body and the supporting arm. This mechanism was designed for supporting 50% body weight of the worker who is the standard Japanese adult male.
2. The supporting arm was worn by several subjects, and its supporting effect was evaluated by measuring EMG waveforms of leg muscles. When the worker takes half-sitting posture, its leg load was measured in two cases: (i) with support, and (ii) without support. The EMG waveforms of these cases were compared, and finally, the supporting effect of the supporting arm was demonstrated by the t-test.

As a future work, it has to be considered the situation when the foot of the supporting arm slips on the floor accidentally. When the foot slips, the supporting effect of the supporting arm cannot be obtained, and the worker will face a falling risk. For example, when the foot of the supporting arm slips, load support will be possible if the joint 1 (or 3) is actuated as an active joint in addition to the joint 2. Also, it is an important task to reduce the weight of the mechanism to improve the portability of the supporting arm.

ACKNOWLEDGMENTS

This work was supported by SMC Corporation for providing pneumatic equipment.

REFERENCES

4. SANKAI, Y., HAL: Hybrid Assistive Limb Based on Cybernics, Robotics Research, the 13th International Symposium ISRR, 2010, p.25-34.
A Human-Machine Cooperation Control Based on Electromyography for Upper Limb Powered Exoskeleton Driven by Pneumatic Muscle

Jun TAO* and Hao LIU*

*State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University
38 Zheda Road, Hangzhou, 310027, China
(E-mail: hliu2000@zju.edu.cn)

Abstract. Although the merits of powered exoskeleton system based on surface EMG signals have been demonstrated, it is not easy to achieve man-machine coordinated motion control. This paper presents a motion control algorithm based on electromyography to control exoskeleton in accordance with the operator’s intention. The algorithm’s main body is the biomechanics model, a simplified Hill-type-based musculoskeletal model, which combines joint kinematics with EMG signals to predict the operator’s intention. The genetic algorithm is used to optimize the internal parameters of the model for individual users so that the model can be adapted to any users. In this paper, we designed an exoskeleton for assistance of upper limb movement to complete the validation of the model. Pneumatic muscles were chosen as the actuator of joint motion due to their high power to mass ratio. Experimental results demonstrates the proposed method’s effectiveness.

Keywords: Powered Exoskeleton, Pneumatic Artificial Muscle, Electromyography, Hill-type Muscle Model, Human-Machine Cooperation Control

INTRODUCTION

Over the past few decades, there have been a large number of research teams dedicated to the development of exoskeleton robots [1]–[7]. Although the application situations are not the same, they are faced with the same problem in the development process: how to design a human-machine interface to understand the operator’s intention, so as to give appropriate movement support.

In general, there are two ways to achieve the estimation of the movement intention — classification and regression [8]. The classification method classifies the operator’s intention into several types of actions, and the regression method can estimate the continuous reference trajectory for the implementation of continuous feedback control. One way to achieve continuous control is to estimate the joint desired torque required to perform the desired movement, which is natural and intuitive for human.

A feasible way to estimate the joint torque is to set the control system at the neurophysiological level, such as the surface electromyography (sEMG) [9]. The sEMG is generated by nerve impulses, and highly correlated with muscle activation and intention of movement. The sEMG could be detected prior to the occurrence of muscle movements [10], thereby the exoskeleton movement’s delay could be reduced if it is applied in the control system.

Several methods have been developed to evaluate the desired joint torques from the sEMG, for examples, the desired joint torque of upper limb exoskeleton is linearly related to the EMG signal in [11], and seven degree of freedoms’ movements of an upper limb are calculated with a neuro-fuzzy controller in [12].

The EMG and joint torque can be correlated at the biomechanical level by using the muscle’s biomechanical model, one representation of which is the Hill-type musculoskeletal model [13]. The model can combine the joint kinematic and the EMG that reflects the muscle activation to predict the joint torque. Rosen [4] presented a torque controller based on the Hill-type muscle model to control a two-link exoskeleton arm to lift loads with the hand and made a performance comparison with an another torque controller based on the black box model (neural network) [14]. An big problem with this biomechanical model is that a lot of parameters in the model, which rely on individual physiological condition, have to be determined [15].

This paper presents an upper limb exoskeleton robot developed to decrease the operator’s effort of lifting loads. The joints’ movement are driven by pneumatic artificial muscles (PAM) due to their high power-mass ratio and flexible characteristic. Exoskeleton’s human-machine cooperation controller is the torque controller based on the Hill-type muscle model. The muscle biomechanical model is simplified for easier model parameter calibration procedures and embedded implementation. The genetic algorithm is used to calibrate the internal parameters of the biomechanical model for individual users so that the model can be adapted to any user. At present, the controller has been validated on the elbow joint of the exoskeleton.
EXOSKELETON HARDWARE

As can be seen in FIGURE 1, the upper limb of the exoskeleton robot consists of two two-link robotic arms and a support frame located on the back. When the operator wears exoskeleton to lift the load, the load will be transmitted through the back support frame to the lower limb of the exoskeleton, and then to the ground. The mechanical structure of the exoskeleton robot is designed according to the body bionic so that the device can fit the operator well. Each robot arm of upper limb has two degrees of freedom to be driven, including flexion/extension on the elbow and shoulder joint. In order to simplify the mechanical structure and control system, the adduction/abduction and internal/external rotation of the shoulder joint are not designed to be driven by actuators, but are kept as movable so that the operator could still feel free in the movement of the two degrees of freedom.

Compared with other actuators, pneumatic artificial muscles (PAM) have a high power to mass ratio, and the elastic properties of the PAM can increase the flexibility of the system. In this paper, the exoskeleton joint motion is driven by PAM (FESTO DMSP-20-180N-RN-CN) with maximum output force of 1500N. A cable-driven transmission system (see FIGURE 1) is adopted to convert the linear contraction motion to the articular motion. Since the extension motion of the joint could be driven by gravity, each joint is driven by only one PAM. The potentiometers are mounted on the joints to measure the current joint angle and the pressure sensors are mounted on the robot arms to measure the inner pressure of the PAM. The intake and exhaust of the PAM are controlled by two position three-way high speed on/off valve (MHE2-MS1H-3/2G-QS-4-K). The strong nonlinearity of the PAM is difficult to model precisely, so our research group has developed empirical models based on a large number of experimental data as suggested in [16]. The model reflects the relationship between the air pressure, the amount of contraction and the output force of the PAM:

\[ F = (a_1 + a_3 P) + (a_2 + a_4 P)e + (a_5 + a_6 P)e^2 + a_7 e^3 + a_8 e^4 \]  

The bipolar surface electrodes are attached to the skin to measure the surface EMG signal. A signal conditioning device with a gain of 1000V/V has been developed to amplify the sEMG signal. All sensor signals are sampled at 1000Hz using a signal acquisition card (National Instrument USB-6001).

FIGURE 1. The structure of the exoskeleton. The lower limb is plotted to explain the principle.

METHOD

This Section describes the muscle biomechanical model and the procedures of estimating joint moment in detail. The model parameter calibration method and the exoskeleton elbow joint control scheme are also discussed in this section.

Biomechanical Model of Muscle

The muscle biomechanical model is based on the Hill-type muscle model, which uses the raw EMG signal and joint kinematics to estimate individual muscle forces and joint torque. Actually, in this paper, the model is simplified for embedded implementation and simple model parameters calibration procedure. The muscle biomechanical model, as shown in FIGURE 2, consists of three modules [17]: 1) EMG to Muscle Activation Model, which converts the EMG signal to muscle activation level; 2) Anatomical Model, which calculates muscle length and moment arms by using the joint angular position and anatomical information; 3) Hill-type Muscle Model, which estimates the muscle force and moment by using the muscle length, moment arm and activation level.
EMG Signal to Activation Model

The post-processed EMG signal is nonlinearly related to the muscle activation level. The nonlinear relationship is modeled as:

$$a(t) = \frac{e^{\Delta t} - 1}{e^A - 1}$$

Parameter $A$ defines the scale of nonlinearity between the post-processed EMG and the activation level. The value of parameter $A$ is constrained to $-3 < A < 0$. The function will become highly exponential function when $A$ is equal to $-3$, and approximate a linear relationship when $A$ approximates zero.

The complete post-processed procedure of the raw EMG signal consists of: a) high-pass filtering to remove the dc offset of the raw EMG signal; b) full wave rectification; c) low-pass filtering to obtain the linear envelope; d) normalization with respect to the peak value of EMG linear envelop obtained during maximal voluntary contraction.

Anatomical Model

The joint angular position data are used as input for the anatomical model to determine individual muscle tendon lengths and moment arm of related DOF for the simplified Hill-type muscle model. In order to obtain these outputs, the anatomical data of the muscle must be used. Using SIMM™ [18], an upper limb anatomical model was developed by Holzbaur et al. [19], which describes the geometry of bones and the complex path of muscle on bones. The anatomical model allows to compute the length of the musculotendon and moment arm in accordance with current joint angular position. In order to implement the real-time estimate of musculotendon’s length and moment arm on the control system, the cubic B-spline curve is used to interpolate the muscle anatomical data of anatomical model, as the work in [20]. The cubic B-spline curve has enough high interpolation accuracy and is computationally inexpensive.

Hill-type Muscle Model

The individual muscle tendon length and activation level were used as input to a simplified Hill-type muscle model to calculate individual muscle forces. Once individual muscle forces are determined, it is multiplied by the muscle moment arm to determine the joint moment generated by the individual muscle. The Hill-type muscle model describes the muscle tendon unit as two components (as shown in FIGURE 3): a muscle fiber in
series with a tendon [21]. The muscle fiber consists of a contractile component in parallel with an elastic element. The force $F^m$ in muscle fiber is computed as:

$$F^m = F_A^m + F_P^m$$  \hspace{1cm} (3)

$$F_P^m = f_p \left( \frac{l^m}{l^o} \right) \cdot F_o^m$$  \hspace{1cm} (4)

$$F_A^m = a \cdot f_a \left( \frac{l^m}{l^o} \right) \cdot F_o^m$$  \hspace{1cm} (5)

where $F_A^m$ is the active force produced by the contractile element when the muscle is activated and $F_P^m$ is the passive force produced by the parallel elastic element when the muscle is stretched. The normalized passive force length curve $f_p$ [22] and the normalized active force-length curve $f_a$ [22] represent the relationship between fiber length and force in muscle fiber. Muscle could generate a peak force when it is at an optimal fiber length $l^o$, and the peak force is called as the maximum isometric force $F_o^m$. $l^m$ is the current muscle fiber length.

The determination of the muscle activation has been described above. If the current muscle fiber length is known, the force $F^m$ of muscle fiber could be determined. As shown in FIGURE 3, there is a relationship between muscle fiber length, tendon length and musculotendinous length:

$$l^m = l' + l^o \cos \varphi$$ \hspace{1cm} (6)

where $\varphi$ is the current pennation angle between the tendon and the muscle fibers, which changes with instantaneous muscle fiber length. The following function is used to calculate pennation angle [17]:

$$\varphi = \sin^{-1} \left( \frac{\sin \varphi}{l^m} \right)$$ \hspace{1cm} (7)

where $\varphi_o$ is the pennation angle at the optimal muscle fiber length. Thus, the muscle fiber length can be determined, given the musculotendon length and the tendon length, that means the muscle force can be computed.

Although tendons are passive elements that act like the passive parallel elastic element of muscle fiber, due to the fact that the tendon’s strain is only about 3% of the tendon slack length when the muscle generates maximum muscle force, the tendon is treated as inextensible [21], [23]. The modeling simplification makes it possible to determine the muscle fiber length and pennation angle. Finally, the output force of the muscle is computed as:

$$F^m = F^m \cos \varphi$$ \hspace{1cm} (8)

Once individual muscle forces are determined, these are multiplied by the muscle moment arm and summed to determine the total joint moment.

$$T = \sum_i T_i$$ \hspace{1cm} (9)

$$T_i = r_i \cdot F^m_i$$ \hspace{1cm} (10)

where $r_i$ is the $i$th muscle’s moment arm obtained by the anatomical model.

**Parameter Calibration**

As previously described, there are some object-specific parameters in the muscle biomechanical model that has to be adjusted. In the present work, 7 muscles about elbow joint flexion/extension motion have been modeled, including Brachialis (BRA), Biceps Brachii long head (BLH), Biceps Brachii short head (BSH), Brachioradialis (BRD), Triceps Brachii long head (TLgH), Triceps Brachii medial head (TmH) and Triceps Brachii lateral head (TLtH).
Adjustable model parameter

There are 10 adjustable parameters in the calibration process. The muscle activation parameter is shared among all muscle models. Tendon slack length varies obviously among individuals and has a great influence on the calculation result of muscle force, so it is allowed to be adjusted. The data from the literature [19] are used to determine the initial values for each muscle’s tendon slack length and constrained to ±15% of the initial values. Two strength coefficients are used to scale flexor and extensor muscles, respectively, to account for differences in muscle strength between people, as the work in [17]. These two global coefficients are used to maintain relative strength across flexor and extensor muscles, respectively, and constrained to ±50%. The other parameters, including optimal fiber length and pennation angle at optimal fiber length, are set according to the values in literature [19].

The 9 muscle model parameters are anatomical and physiological parameters and only required to be calibrated once per person. The muscle activation parameter has to be calibrated once for every experimental session, because it is affected by electrode position on skin and skin condition.

Calibration Method

It was assumed that the elbow joint moment calculated by the biomechanical model of muscle should be equal to the results calculated by the inverse dynamic model. Then, an optimization algorithm can be used to calibrate the internal parameters of model to accurately estimate the joint moment. The inverse dynamic model to estimate elbow joint moment according the joint kinematics is described as [24]:

$$ T = M(\theta)\ddot{\theta} + V(\theta, \dot{\theta}) + G(\theta) $$  \hspace{1cm} (11)

where parameter $M$ is the inertia matrix, $V$ is the Coriolis force and centrifugal force vector, $G$ is the gravity vector and $\theta$ is the joint angular position.

Genetic algorithm is used to optimize the model parameters to achieve the closest estimation of the elbow joint FE moments compared to that calculated using an inverse dynamics approach. Genetic algorithm is easy to implement and irrelevant to the problem domain. The algorithm begins with a population, which represents a solution set, and performs the heuristic selection based on the fitness function. The cross and mutation manipulation are used to produce new population and the process is repeated until the convergence condition is satisfied.

In the present work, the root mean square error (RMSE) is selected as the fitness function:

$$ \text{Fit}_{GA} = \frac{1}{N} \sum_{i=1}^{N} (T_i - \hat{T}_i)^2 $$  \hspace{1cm} (12)

where $T$ is the torque calculated by the biomechanical model, $\hat{T}$ is the reference torque calculated by the inverse dynamic model, $N$ is the number of sample points.

Control Structure

The control scheme is shown in FIGURE 4: the EMG signal and elbow joint angle position signal are sent to the biomechanical model to estimate the joint torque, which is multiplied with an assistance ratio to obtain the exoskeleton joint torque controller’s reference torque. Due to the mechanical restriction, there is no force sensor mounted to measure current PAM output force. Thus, the joint torque controller couldn’t be achieved directly. Alternatively, for implementing the human-machine cooperation control based on the operator’s intention, the reference torque is converted to reference internal air pressure value of PAM in accordance with the PAM model and the current joint angle.

$$ \text{Fit}_{GA} = \frac{1}{N} \sum_{i=1}^{N} (T_i - \hat{T}_i)^2 $$

where $T$ is the torque calculated by the biomechanical model, $\hat{T}$ is the reference torque calculated by the inverse dynamic model, $N$ is the number of sample points.

$$ P_r = \frac{T_r}{r_{PAM}} - a_1 - a_2e - a_3e^2 - a_4e^3 - a_5e^4 $$

where $P_r$ is the reference internal air pressure value of PAM, $T_r$ is the estimated reference torque. The parameter $r_{PAM}$ is the moment arm of the PAM output force and the parameter $e$ is the contraction amount of the PAM computed in accordance with the joint angle. Finally, the error between reference air pressure and air...
pressure measured by air pressure sensor is sent to low level controller to achieve the continuous close loop feedback control.

At present, the low level controller is designed as a proportional controller. It is assumed that a simple low level controller could make it easier for the operator to adapt to the behavior of the system and has a little effect on the movement pattern of the operator.

**EXPERIMENTS AND DISCUSSION**

All experiments are divided into two categories: the first category is done to calibrate the model parameter and test the model’s effectiveness; the second category is done to validate the effectiveness of human-machine cooperation control algorithm based on EMG signal for power-assisted exoskeleton robot actuated by PAM. All experiments’ operator was the same person (male, 22 years, 1.70 m, 60 kg). All experiments require the operator to wear the exoskeleton to perform the elbow joint’s flexion and extension motion. Electrodes are placed following the recommendations in [25].

**Model Validation**

The exoskeleton actuators were not driven in this experiment and there was no any limitation within the motion range of exoskeleton joint for the operator. The operator was required to perform the elbow flexion and extension motion with 0kg load. Each experiment lasted 5~7 seconds. The EMG signal and joint angle signal were recorded and sent to generic algorithm to calibrate model parameter. The joint torques predicted by the model before and after parameter calibration are plotted in FIGURE 5.

As is shown in FIGURE 5, the biomechanical model could capture the trend of joint torque and the RMSE is about 0.75 Nm after parameters calibration. However, the uncalibrated model does not capture the trend and the RMSE is about 1.58 Nm. So, it can be concluded that the model can be adopted to estimate the joint moment after proper parameter calibration.
Validation of Human-Machine Cooperation Control

In this experiment, the actuators were driven to provide the power support. More specifically, the operator was required to perform flexion and extension of the elbow joint with three different kinds of loads (3kg, 6kg and 10kg) and finish continuous FE movement and staged FE movement. The staged FE movement means that the operator should lift his or her arm to a certain position firstly. After remaining stationary for a period of time, the operator lift the arm to another position. The assistant ratio was set to 0, 0.5, 1.0 and 1.5 respectively to compare the power support effect at different assistant ratio.

The experiment results of continuous elbow joint FE movement with 6kg load are plotted in FIGURE 6. The results of assistant ratio set at 0, 0.5, 1.0 and 1.5 are shown, respectively, in the FIGURE 6(a), FIGURE 6(b), FIGURE 6(c), and FIGURE 6(d). In every figure, the top represents the elbow joint trajectories and the torque contributions of the operator estimated by the model, while the bottom represents the normalized linear envelope of the biceps brachii and triceps brachii muscle’s EMG signal. The experiment results of staged elbow joint FE movement with 6kg load are plotted in FIGURE 7. The results of assistant ratio set at 0.5, 1.0 and 1.5 are shown, respectively, in the FIGURE 7(a), FIGURE 7(b), and FIGURE 7(c).

As is shown in FIGURE 6, the shape of elbow joint trajectories with power support are similar to the trajectories without power support (assistant ratio is 0), indicating that the desired movement could be performed successfully and the motion pattern could keep unchanged. It is worth noting that the torque

![FIGURE 6. Continuous elbow joint flexion and extension movement. (a) assistant ratio: 0; (b) assistant ratio: 0.5; (c) assistant ratio: 1; (d) assistant ratio: 1.5.](image.png)

![FIGURE 7. Staged elbow joint flexion and extension movement. (a) assistant ratio: 0.5; (b) assistant ratio: 1; (c) assistant ratio: 1.5.](image.png)
contributions and the post-processed EMG signal magnitude of the operator could be reduced with increased assistant ratio, which validates the assistant effectiveness. The staged movement experiments could be used to test muscle’s endurance. The more loads the exoskeleton could bear, longer the operator could hold the loads on a certain position. However, when keeping arm at a certain position, the fluctuation of operator’s joint position is more obvious with the increased assistant ratio. An explanation could be presented: Although the smooth EMG signal linear envelop was used to estimate joint moment, but the fluctuation of linear envelop still existed when the operator didn’t move his or her arm due to the non-stationary property of the EMG signal. Since the exoskeleton power assistance directly relied on the estimation of operator’s joint moment, which relied on the non-stationary EMG signal, the fluctuation was reflected in the controller’s reference input and was amplified with the increased assistant ratio.

CONCLUSION

This paper investigates a human-machine coordinated motion control strategy based on surface EMG signals for exoskeleton robot driven by pneumatic artificial muscle to achieve motion-assisted effect. The control algorithm has been implemented on the elbow joint of the exoskeleton and experiments has validated the algorithm’s effectiveness. When the operator wears the exoskeleton to lift loads, the exoskeleton could mostly reduce the operator’s efforts. However, the operator couldn’t keep the joint stationary well at a certain position. Further work will focus on the optimization of the control algorithm to solve the above problems and the algorithm’s implementation on multiple joints.

ACKNOWLEDGMENTS

This paper is supported by the National Natural Science Foundation of China (Project No. 51521064) and the Natural Science Foundation of Zhejiang Province (Project No. LY13E05004).

REFERENCES


Copyright (C) JFPS All Rights Reserved. - 2B13 -
EVALUATION OF AIR COMPRESSING METHODS FOR DEVELOPMENT OF A PORTABLE PNEUMATIC POWER SOURCE

Manabu OKUI*, Yuki NAGURA**, Shingo IIKAWA*, Yasuyuki YAMADA*** and Taro NAKAMURA**

*Department of Precision Mechanics, Graduate School of Science and Engineering, Chuo University
1-13-27 Kasuga, Bunkyo-ku, Tokyo, 112-8551 Japan
(E-mail: m_okui@bio.mech.chuo-u.ac.jp)

**Department of Precision Mechanics, Faculty of Science and Engineering, Chuo University
1-13-27 Kasuga, Bunkyo-ku, Tokyo, 112-8551 Japan

Abstract. Pneumatic driven systems are widely used in various fields, and portable pneumatic sources have been developed to make the system portable. However, some of their characteristics still need to be improved before they can be applied. Herein, various air compressing methods are evaluated to fabricate a portable and practical pneumatic source. The methods compared include a small compressor activated by a battery, material phase change, air tank, and chemical reaction. The results show that all methods have advantages and disadvantages. The compressor works stably but has low energy efficiency when it generates high pressure. Material phase change can generate a large amount of air per mass but has low pressure. Conversely, the chemical reaction of sodium bicarbonate and citric acid can generate high pressure but only a small amount of air per mass. The air tank can be used repeatedly but needs to be implemented on a large scale to store a large amount of gas.

Keywords: Pneumatic power source, Robot, Wearable assist, Chemical reaction, Phase change

INTRODUCTION

Pneumatic driven systems are widely used in various fields. Recently, they have been applied to wearable assist systems [1-3] because of their light weight, high-power, and ability to retain structural softness. To activate these pneumatic actuators, an air compressor driven by a large motor or engine is generally used. These heavy and large systems prevent pneumatic systems from being portable. By using portable pneumatic power sources, the pneumatic system can have a broader field of application.

Portable pneumatic power sources have been developed [4-7]. The most common method is activating a small compressor by using batteries. This method can generate constant air pressure [4]; however, the energy efficiency of this method is poor since the compressed air is generated from electric energy. Moreover, pneumatic power sources using either gas explosion [5] or battery charge/discharge [6] have been developed. While these methods can generate high pressure, they cannot realize the stable generation of compressed air; therefore, using a compressed air tank instead of a pneumatic power source is another possibility. However, high-pressure-proof air tanks are heavy for enduring high pressure and low-pressure-proof air tanks are large for storing a sufficient amount of air. To overcome these problems, a pneumatic power source using the phase transition at the triple point of carbon dioxide has been developed [7]. Although this method can generate 0.42 MPa of compressed air without making a loud noise, it cannot preserve the energy of compressed air because the dry ice, used as the power source, sublimes at room temperature. In addition, this device can cause frostbite if the user touches the vessel containing dry ice. In this paper, a comparison of pneumatic sources is presented to find solutions to the aforementioned problems [8]. The comparison is useful in selecting a pneumatic source for untethered soft robots at 0 ~ 0.2 MPa; however, this comparison cannot be considered in relation to existing pneumatic systems, such as pneumatic-powered tools or pneumatic artificial muscles, since these require a pressure of approximately 0.7 MPa.

In this study, we aimed to develop a portable pneumatic power source, which would be able to withstand practical use. As an initial step, various air compressing methods were evaluated in terms of portability, and the results are presented in this paper. Methods for comparison are a small compressor activated by battery, phase change of material, air tank, and chemical reaction.
AIR COMPRESSING METHOD FOR COMPARISON

In this chapter, different types of compressed air generation methods are described for the purpose of comparison. They were selected in terms of practical use and include existing methods, such as small battery driven compressors, as well as novel methods such as chemical reaction and material phase change. Gas explosion and the chemical reaction of hydrogen peroxide are not discussed in this paper because their characteristics were measured in [8] and also because they may cause injuries to users due to the heat generated during the air compression process.

Small Compressor Driven by Lithium-Ion Battery

A lithium-ion battery drives a compact compressor. In a motor, the piston or bellows are pressed to compress the air. This method can produce higher pressure by connecting compressors in series, since the method creates a difference between inflow and outflow. This method can also stably generate compressed air.

Phase Change of CO₂

Carbon dioxide generated by the sublimation of dry ice is used as a pressure source. Dry ice is has been routinely used as a coolant. Carbon dioxide is a byproduct of the industries such as the production of ammonia. When dry ice is placed in a sealed container and heated, a part of it sublimate as carbon dioxide; therefore, pressure gradually increases.

Phase Change of Dimethyl Ether

The dimethyl ether (DME) phase change from liquid to gas is used widely in air dusters. Although this substance is gaseous at room temperature, it exists in liquid state under relatively low pressure; therefore, it can be stored as a liquid inside a lightweight can with high capacity. A commercially available air duster, which uses 100% DME as the gas source, is considered in this paper.

Air Tank

The tank is filled with air. Although it needs another pneumatic power source, the tank can be used repeatedly. If the tank can endure high pressure, a larger amount of gas can be stored.

Chemical Reaction of Sodium Bicarbonate and Citric Acid

Carbon dioxide, generated by mixing citric acid and sodium bicarbonate, is used as compressed air. Water is also used as a solvent. Eq. 1 shows the reaction formula. These substances are generally used for cleaning. They are cheap, easy to store, easy to obtain, and easy to dispose. Additionally, the carbon dioxide, water, and trisodium citrate, produced by this reaction, are non-toxic and suitable for consumption.

\[
C_6H_8O_7 + 3NaHCO_3 \rightarrow Na_3C_6H_5O_7 + 3H_2O + 3CO_2 \uparrow \quad (1)
\]

EXPERIMENTAL CONDITIONS

Criteria of Portability

A portable pneumatic power source should be lightweight and small enough so as not to exert a big load on the wearer. Additionally, having the ability to exert enough pressure to activate the pneumatic device is important. Thereby, two evaluation criteria were chosen: the maximum pressure provided by each method and the flow capacity, which is the total flow rate per mass of the pneumatic source. Flow capacity is calculated by dividing the amount of generated gas by the pneumatic power source mass. The mass for the calculation is defined as the...
mass of consumed part of the system. Mass of components such as compressors or tanks is not included. Table 1 shows the calculation methods for each method.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mass for calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compressor and battery</td>
<td>Battery</td>
</tr>
<tr>
<td>Tank</td>
<td>Gas charged in tank</td>
</tr>
<tr>
<td>Chemical reaction</td>
<td>Citric acid and sodium bicarbonate</td>
</tr>
<tr>
<td>Dry ice</td>
<td>Dry ice</td>
</tr>
<tr>
<td>Phase change</td>
<td>Vaporized material</td>
</tr>
</tbody>
</table>

**Experimental Setup**

The experimental setup is depicted in Figures 1 and 2. Two types of flow meters were used in the experiment: a thermal type flow meter, PFMB7201-02-A-M (SMC) for the measurement of CO₂ and atmosphere and a volume type flow meter GF1010 (GL science) for the measurement of other gases. All experiments were conducted at 25°C. The pressure of the atmosphere was 1.013 MPa. Each measurement was performed thrice and the average value was calculated.

Fig. 1 shows the thermal type flow meter experimental setup for the air tank, chemical reaction, and dry ice. The pressure sensor was SEU 11-6UA (PISCO Co.,) and its output was recorded by a control desk. Fig. 2 shows the volume type flow meter experimental setup for DME. Since the GF1010 doesn’t have an output, the value on the display was recorded using a video camera.

**FIGURE 1.** Schematic diagram of the experimental device (Thermal mass)

**FIGURE 2.** Schematic diagram of experimental device (Volume type)

**Measurement Conditions for Each Method**

*Small Compressor Driven by Lithium-Ion Battery*

The schematic of the experimental setup is shown in Fig. 3. First, the compressed air, which was discharged from the compressor, was stored in a pressurized tank. Then, two port valves were opened when the interior of the tank reached the desirable pressure. The battery was fully charged at the beginning and measurement was conducted until the battery was exhausted. The battery used was a 24VC100T2 type manufactured by NISSEN.
Corporation; it weighed 860 g and had a capacity of 100 Wh. The compressor was MP-2-C manufactured by SUQUSE Corporation and weighted 500 g. The pressure inside the tank was set to 0, 0.2, and 0.4 MPa. When the pressure reached or exceeded the desirable pressure, the valves were opened and the flow rate was measured while keeping the pressure constant. In this method, the total flow of the generated air was divided by the battery mass.

**Phase Change of CO\(_2\)**

Dry ice weighing 50 g was placed in a 400 ml bottle. The total flow generated from the dry ice was measured. In this method, the total flow rate was divided by the mass of dry ice to calculate the flow capacity.

**Phase Change of Dimethyl Ether**

The air duster, which contained dimethyl ether, was ejected for 300 seconds; the pressure, total flow rate, and variation of mass were measured. The measured total flow divided by the variation of mass is the flow capacity.

**Air Tank**

A tank with a capacity of 600 ml was filled with 0.5 MPa of air. Then, the tank’s outflow was measured until the tank’s internal pressure reached 0 MPa. The mass was measured before and after the release of air and the difference was used in the calculation of flow capacity.

**Chemical Reaction of Sodium Bicarbonate and Citric Acid**

A schematic diagram of the measurement setup is shown in Fig.4 The mass of materials is shown in Table 2. The experimental setup comprised a 500-ml bottle A with a citric acid solution, a bottle B with a volume of about 500 ml containing sodium bicarbonate, the electric regulator, and the check valve. The citric acid aqueous solution in bottle A flowed into bottle B by the pressure applied from the compressor and mixed with sodium bicarbonate to generate carbon dioxide. In this experiment, first, all the citric acid aqueous solution was injected into bottle B with the 2-way valve closed. Next, the bottle was shaken to stimulate a chemical reaction and measure the maximum pressure. Then, the flow rate was measured with the two port valve opened. In this method, the total generated flow rate divided by the mass of citric acid and sodium bicarbonate was taken as the flow capacity.

**TABLE 2.** Mass of experimental material

<table>
<thead>
<tr>
<th>Material</th>
<th>Mass (g)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Citric acid</td>
<td>9.6</td>
</tr>
<tr>
<td>Sodium hydrogen carbonate</td>
<td>12.6</td>
</tr>
<tr>
<td>Water</td>
<td>150 ml</td>
</tr>
</tbody>
</table>
RESULTS AND DISCUSSION

Results

Experimental results are shown in Tables 3 and 4. Table 3 shows the measured and theoretical value of total flow and mass of the consumed part. The theoretical values were calculated on the basis of molar volume for the air tank, chemical reaction, phase changes of CO₂, and DME. In this study, the molar volume was taken as 22.4 mol/L. The theoretical values for the compressor and battery were calculated by the drive time and specification of the compressor. The drive time is estimated by dividing battery capacity by measured energy consumption of compressor.

Table 4 depicts the maximum pressure and flow capacity for each method; advantages and disadvantages are also described. The chart in Fig. 5 explains the portability of the pneumatic power source. The vertical axis is the maximum pressure and the horizontal axis is the flow capacity. The more upper right a point is, the better is the portable pneumatic source it represents.

**TABLE 3. Mass of experimental material**

<table>
<thead>
<tr>
<th>Methods</th>
<th>Total (Theoretical) [NL]</th>
<th>Total (Experimental) [NL]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air tank</td>
<td>3.27</td>
<td>3.24</td>
</tr>
<tr>
<td>Chemical reaction</td>
<td>3.36</td>
<td>3.27</td>
</tr>
<tr>
<td>Phase change of CO₂</td>
<td>27.5</td>
<td>23.16</td>
</tr>
<tr>
<td>Phase change of DME</td>
<td>4.47</td>
<td>4.33</td>
</tr>
<tr>
<td>Compressor and battery</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.1 MPa</td>
<td>2343</td>
<td>1319</td>
</tr>
<tr>
<td>0.2 MPa</td>
<td>1362</td>
<td>402</td>
</tr>
<tr>
<td>0.4 MPa</td>
<td>515</td>
<td>65.3</td>
</tr>
</tbody>
</table>

**TABLE 4. Mass of experimental material**

<table>
<thead>
<tr>
<th>Methods</th>
<th>Maximum pressure [MPa]</th>
<th>Flow capacity [NL/g]</th>
<th>Advantage</th>
<th>Disadvantage</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air tank</td>
<td>Depends on spec of tank</td>
<td>0.84</td>
<td>-Repeatedly usable</td>
<td>-Large scale</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-Needs additional pneumatic source</td>
</tr>
<tr>
<td>Chemical reaction</td>
<td>More than 1 MPa</td>
<td>0.152</td>
<td>-Easy to store</td>
<td>-Difficult to generate gas stably</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>-Easy to obtain and dispose</td>
<td></td>
</tr>
<tr>
<td>Phase change of CO₂</td>
<td>More than 1MPa (0.42 MPa at triple point)</td>
<td>0.477</td>
<td>-High flow rate under triple point</td>
<td>-Low flow rate</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-Vessel becoming too cold may harm the user as it can cause frostbite</td>
</tr>
<tr>
<td>Phase change of DME</td>
<td>0.428 MPa (at 25 degree)</td>
<td>0.521</td>
<td>-High flow rate</td>
<td>-Combustible</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-Pressure decreases at low temperature</td>
</tr>
<tr>
<td>Compressor and battery</td>
<td>0.4 MPa (Depends on spec of a compressor)</td>
<td>0.365</td>
<td>-Stable movement</td>
<td>-The higher pressure, the lower flow capacity</td>
</tr>
</tbody>
</table>
**Discussion**

The efficiency of the compressor decreases when high pressure is generated, as shown in Table 3. This is due to the compressor generating pressure by compressing the atmosphere using a piston; therefore, the higher pressure the compressor generates, the lower is the energy efficiency at the same total flow. Two methods using material phase change have similar characteristics and can generate large amounts of compressed air per mass. It is assumed that these methods have the same potential as pneumatic power sources. Solid or liquid materials generate 22.4 NL/mol of gas when vaporized or sublimated. Conversely, the chemical reaction can generate significantly high pressure but only a small amount of compressed air per mass. The air tank seems to be the best method but needs to be implemented on a large scale for a large amount of gas to be stored. Overall, these methods have both merits and demerits; therefore, the combination of multiple methods is justifiable. For example, the combination of DME phase change can generate a large amount of gas but low pressure; the chemical reaction can generate high pressure but a small amount of gas.

**CONCLUSION**

The portability of various air compressing methods was measured. Methods for comparison are compact compressor driven by battery, air tank, phase change of dimethyl ethyl, phase change of CO$_2$, and chemical reaction. The portability criteria considered maximum pressure and integrated flow rate per mass. The results showed that all methods have advantages and disadvantages. The compressor can supply stable compressed air but has low energy efficiency when high pressure is generated. Two methods, using material phase change, can generate a large amount of compressed air per mass but low pressure. Conversely, the chemical reaction can generate high pressure but a small amount of compressed air per mass. The air tank seems to be the best method; however, it needs to be implemented on a large scale for a large amount of gas to be stored.

**ACKNOWLEDGMENTS**

This research was supported by the New Energy and Industrial Technology Development Organization (NEDO).

**REFERENCES**

WRIST REHABILITATION SIMULATOR FOR P.T. USING PNEUMATIC PARALLEL MANIPULATOR

Regulation of Wrist Viscoelastic Property and Therapy Motion Evaluation

Masahiro TAKAIWA, Hiroyuki IMANAKA
Tokushima University, 3-1-1, Tokushima, 700-8530 JAPAN

Abstract. Currently, the total number of P.T. is increasing to be more than 90,000. The problem of shortage of total number is being solved, but skill improvement for them is still remaining as a problem. In this study, we focus on wrist motion necessary for day-life and we develop a wrist rehabilitation training simulator to improve the skill of P.T.. Concretely, patient wrist model is constructed using 6 D.O.F. pneumatic parallel manipulator, which correspond to complex wrist motion. It is important to install an actual mechanical property of a patient on a manipulator. We propose a user friendly interface with which P.T. can regulate the mechanical property of wrist model. After that the rehabilitation motion given by expert and rookie P.T. are stored on a manipulator. By evaluating these motion using 3D graph etc, we provide an educational and skill transfer environment among expert and rookie P.T.

Keywords: Pneumatic parallel manipulator, Rehabilitation, Wrist joint

INTRODUCTION

As is well known, human wrist joint plays an important role in our day-life, such as a meal, change of clothes, and so on. According to a Japan physical therapy white paper in 2014[1], there are currently about 93,000 of P.T.. Comparing with that in a decade ago, indeed, the total number of P.T. has been increasing but skill improvement and skill transfer of P.T. are still remaining as an important issue. Some wrist rehabilitation devices have been investigated [2][3][4][5], but most of them are training devices not for P.T. but for patients.

Hearing from P.T., rookie or its candidates are doing their wrist rehabilitation practice by taking another P.T.’s wrist each other. Therefore robot technology has possibility to bring more efficient training environment, which is a motivation of this study. Then we develop a training simulator for P.T. in wrist rehabilitation using a pneumatic parallel manipulator from a view that it has 6 D.O.F. sufficient to correspond to complex wrist motion and has back-drivability resulted from air compressibility, which works as safe function. In order to realize such a training simulator, a manipulator is required to realize a mechanical property of various types of patient wrist. Our basic idea is to realize a physical patient wrist model by implementing a position-based impedance control on a manipulator. However it is not easy to give an appropriate impedance parameter corresponding to various types of patient. So we propose a method where an expert P.T. themselves can regulate the mechanical property(impedance model) with feeling reaction torque from a manipulator during their therapy motion to close to that of patient's wrist based on their expert knowledge and long-term experience. In order to do so, we provide a user friendly interface for P.T. to regulate the mechanical property easily.

Subsequently, an expert P.T. and a rookie one give their therapy motion to the completed wrist physical model on a manipulator. By comparing the recorded training motion quantitatively using a visual interface like 3D graphics, we provide an environment to skill improvement and skill transfer between an expert P.T. and a newcomer one. The validity of the proposed systems are confirmed through some experiments.

WRIST REHABILITATION SIMULATOR

Schematic Diagram

A developed wrist rehabilitation simulator is shown in Fig.1 (a). So called Stewart type platform is introduced, where an upper platform is driven with 6 low friction type pneumatic cylinders (Airpel Co. Ltd., 9.3 mm in internal diameter, 150 mm in rod stroke). And a mannequin wrist model is mounted on the upper platform via 6 axis force/moment sensor. Fig.1 (b) shows the coordinate system of the manipulator \( \mathbf{h} = [x, y, z, \phi, \theta, \psi]^T \), where an origin of \( \mathbf{h} \) is agree with a center point of wrist joint of a mannequin model. The rotational angle is expressed using roll-pitch-yaw angle notation. Fig.2 shows a wrist motion, where pronation/supination, radial flexion/ulnar flexion and flexion/extension motion correspond to the rotational direction \( \psi, \theta \) and \( \phi \), respectively.
Force/moment vector at an origin of $h$ is defined as $\mathbf{f}_h = [f_{he}^T \tau_{he}^T]^T = [f_x, f_y, f_z, \tau_\phi, \tau_\theta, \tau_\psi]^T$, which is obtained through coordinate transformation based on the measured force/moment with a 6 axis force/moment sensor. $\mathbf{f}_h$ also works on a link equivalently as disturbance $\mathbf{f}_l$, which satisfy the following relation from a principle of virtual work.

$$\mathbf{f}_h = \mathbf{J}^T \mathbf{f}_l \quad (1)$$

where $\mathbf{J}$ is a Jacobi matrix which forms the next relation.

$$\frac{d\mathbf{e}}{dt} = \mathbf{J} \frac{dh}{dt} \quad (2)$$

where $\mathbf{e} = [e_1, \ldots, e_f]^T$ is a link vector with a displacement of each piston rod.

Fig. 3 shows the pneumatic driving circuit. The pressure in each cylinder's chamber, $p_1, p_2$ are detected by a pressure sensor and the displacement of a piston rod $\mathbf{e}$ is measured by a wire type rotary encoder (0.025mm in resolution). A pressure in chambers are regulated by a flow control type servo valve (FESTO MPYE-5), where a supply pressure $p_s$ is set to 500 kPa. Control algorithm is implemented on RTAI a real-time extension of Linux with sampling interval of 5 ms.

Proposed Rehabilitation Environment with Training Simulator

Fig. shows the schematic diagram of the proposed rehabilitation environment with a training simulator. The function indicated by an arrow with a number is described as follows.

1. An expert P.T. regulates a mechanical property of the wrist model on a manipulator to close to that of the actual patient's wrist property by feeling reaction torque based on their expert knowledge and long term experience. In order for a P.T. to do so, we provide a user-friendly interface described later. These regulation motion is done for the whole area of movable joint angle as an iteration work.
### TABLE 1: Control parameters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_p, T_{pn}$</td>
<td>Time constant of pressure response and its nominal value</td>
</tr>
<tr>
<td>$K_p, K_{pn}$</td>
<td>Steady gain of pressure response and its nominal value</td>
</tr>
<tr>
<td>$K_v$</td>
<td>Steady gain between piston velocity and pressure</td>
</tr>
<tr>
<td>$m, m_n$</td>
<td>Equivalent mass for one cylinder and its nominal value</td>
</tr>
<tr>
<td>$b, b_n$</td>
<td>Viscous coefficient and its nominal value</td>
</tr>
<tr>
<td>$f_h$</td>
<td>Force/moment applied by P.T.</td>
</tr>
<tr>
<td>$f_f$</td>
<td>Force equivalently applied on a link</td>
</tr>
<tr>
<td>$f_g$</td>
<td>Generated force from cylinder</td>
</tr>
<tr>
<td>$f_d$</td>
<td>Reference of generation force</td>
</tr>
<tr>
<td>$A_1, A_2$</td>
<td>Cross sectional area of head and piston side</td>
</tr>
<tr>
<td>$p_1, p_2$</td>
<td>Air pressure in head and piston side</td>
</tr>
<tr>
<td>$\ell$</td>
<td>Displacement of piston rod</td>
</tr>
<tr>
<td>$T_u, T_{pq}$</td>
<td>Time constant of filter</td>
</tr>
<tr>
<td>$u$</td>
<td>Control input (input voltage of valve)</td>
</tr>
</tbody>
</table>

### FIGURE 4: Educational rehabilitation environment

2. An expert P.T. implements their therapy motion for the completed wrist model which behaves as a patient wrist. The given therapy motion is recorded on a manipulator quantitatively.

3. Similarly, a rookie P.T. also implements their therapy motion for the wrist model and it is recorded on a manipulator quantitatively.

4. Comparing the given therapy motion from an expert P.T. and a rookie one, a place for skill improvement and skill transfer between them can be provided.

### Control System

In order to realize a mechanical property of wrist joint on a manipulator, a position based impedance control system is employed as shown in Fig.5, where system parameters are explained in Table 1. A force/moment $F_h$ applied by a P.T. is fed back through an inverse impedance model $I_{mp}$ and a position control system based on a generation force control loop described in Fig.5. (b) is composed[6]. In a position control loop, PD controller is employed and a disturbance observer works to improve robustness against external force acting as a disturbance. A disturbance observer is also utilized in a generation force control system to declare the influence of nonlinear pressure response and to follow a reference force $F_d$ as 1 type servo system. Before setting an impedance model $I_{mp}$ as that of the actual wrist property, the wrist property of a patient is roughly identified in advance. Here a manipulator is used as a measurement tool of the wrist property. Concretely, a mannequin hand model is changed with a hand hold equipment with a patient's hand inserted and a position control is implemented with turning off the signal from impedance model. Giving the reference position/orientation $H_r$ at constant low angular velocity within the moving area of a wrist, the wrist mechanical property is obtained from the joint deflection angle and reaction torque from a wrist joint. Fig. shows the obtained static mechanical property of a pseudo patient for $\phi$ axis as an example.
Regulation of Wrist Mechanical Property on Manipulator

In order to make a mechanical property of wrist model to close to an actual wrist property of a patient, we realize a regulation function, where a P.T. themselves can regulate the wrist mechanical property with doing their therapy motion. In order to do so, we provide a user-friendly interface as shown in Fig. 7. Two joysticks are equipped to a forearm model. During therapy motion, a P.T. feels reaction torque from a manipulator at a current joint angle and they regulate a proper reaction torque by operating a corresponding joystick on an interface shown in Fig. 7.

Seeing from the result in Fig., and if we focus the relation at flexion / extention direction, we set the mechanical property of the wrist joint as the following equation.

\[
\tau_{\phi} = a_1 \phi^3 + a_2 \phi + b \dot{\phi}
\]  

where \(a_1, a_2\) are the parameter for static relation and \(b\) is the viscous coefficient. In the interface shown in Fig.7, the left joystick increase/decrease the \(a_1\) that dominantly regulates the large joint angle while right joystick does the \(a_2\) that dominantly regulates the small joint angle range. The viscous coefficient \(b\) is kept constant for ease in this study.
Eq.(3) is written as the following by discretization with sampling period $T_s$

$$\tau_\phi(k) = a_1\phi^3(k) + a_2\phi(k) + b\frac{\phi(k) - \phi(k-1)}{T_s}$$

(4)

Eq.(4) is modified as Eq.(5).

$$\tau_\phi(k) = a_1\phi(k) + a_{2v}\phi(k)$$

(5)

where $\tau_v = \tau_\phi + b\frac{\phi(k-1)}{T_s}$, $a_{2v} = a_2 + \frac{b}{T_s}$

Solving Eq.(5) algebraically, we obtain the reference joint angle $\phi$, corresponds to the output of $I_{mb}$ in Fig.5 (a).

**EXPERIMENTAL RESULTS**

Position Control Performance

The control performance of a position based impedance control depends on that of the inner position control system. Fig.8 shows a step response for horizontal and rotational direction, where rise time is about 0.5 sec. No overshoot and enough steady state property are obtained, which shows the effectiveness of the proposed control system.

Regulation of Wrist Mechanical Property

Fig.9 shows the result of regulation of wrist mechanical property on a manipulator for the direction of flexion/extension, where figure (a) and (b) is the static relation with $b = 0$. Figure (a) is the case of changing parameter $a_1$, while the figure (b) shows the case of $a_2$. Good agreement shows that a mechanical property can be realized arbitrarily. Figure (c) shows the visco-elastic property. The viscous property also can be confirmed.

Providing Place for Skill Improvement and Skill Transfer
In this section, the function described by arrow with 2),3) and 4) in Fig.4 are investigated. Fig.10 shows a wrist joint bones. Some small pieces of bones called curpus are gathered to form a wrist joint. In a general therapy of wrist joint, contraction force along with a forearm is indispensable to prevent friction in joint bones during a rotational motion. The Contraction force is also one of the therapy skills.

In order to train the proper contraction force, we introduce AR environment as shown in Fig.11. A P.T. give the contraction force by seeing the screen displayed by USB camera as shown in figure (b). At the AR marker, a reference contraction force that is obtained in advance by an expert P.T. is shown with a white colored vector and the current contraction force is shown with green one in real-time. A subject can gives proper contraction force with visual support.

Fig.12 shows the comparison of contraction force, where red and black line shows the case with and without visual support. It is clear that proper contraction force can be given with visual support. Centering the developed training simulator, we can provide a place for the skill improvement or skill transfer between an expert and rookie P.T..

FIGURE 10: Wrist joint bone

FIGURE 11: Training with AR environment

FIGURE 12: Comparison of training skill
CONCLUSION

In this study, a training simulator for P.T. in wrist rehabilitation is developed. A pneumatic parallel manipulator is introduced as an equipment from view point of its features of multiple D.O.F. suitable for complex wrist motion and safety property due to the air compressibility.

In order to realize a patient's wrist mechanical property on a manipulator, a position based impedance control system is employed and we proposed a scheme where a regulation of the mechanical property is done by an expert P.T. themselves based on their expert knowledge and long-term experience.

By comparing the recorded training motion of an expert P.T. and a rookie one for the completed wrist model of a patient using such as 3D graph quantitatively, the place for the skill improvement and skill transfer can be provided.

ACKNOWLEDGMENT

This work was supported by JSPS KAKENHI Grant Number 15K0589900. An appreciation is represented by authors.

REFERENCE

Development of Tendon-Driven Care Assistance
Robot Arm Driven by Air Pressure Controlling

Daichi KIMURA*, Osamu OYAMA**

Student of the Graduate School of Meiji University
E-mail: ce62023@meiji.ac.jp
** Meiji University
E-mail: oyama@meiji.ac.jp

Abstract. Currently there is a move to introduce robots to nursing care. I realized a robot arm using a pneumatic cylinder for nursing care. The robot arm can change the rigidity of the joint by changing the pressure inside the cylinder, and flexible operation is possible. When care is taken with a robot arm, a reference value for starting the operation of the device is necessary. By setting this value properly, comfortable nursing care can be done. In this experiment, the pressure inside the cylinder was adjusted to prepare three kinds of joint hardness, and the optimum threshold value was further obtained by using psychological evaluation value by a tester. By using the proper threshold revealed in this experiment, comfortable nursing care becomes possible and it can be considered to be useful for future nursing robot arms developed in the future.

Keywords: Keywords; Robot Arm, Air Pressure Controlling, Care Assistance, Start operation, Psychological evaluation value

INTRODUCTION

Currently, the declining of birthrate and the aging population are progressing in Japan. So it is necessary to reduce the burden of caregiver to elder populations. We focused the burden of caregiver when he assist the standing up motion of human from the seat position as shown Figure 1. In order to avoid the load stress to caregiver, this action must be substituted by the robot arm assist, of course it must be safe and comfort. In this report, we will show the pneumatic and tendon driven care robot arm which assist the stand up motion of human.

ROBOT ARM

The proposed robot arm can do flexible/stiff operation as desired, since the compliance/stiffness of arm joint motion can be selected to needed magnitude in wide range by using construction pneumatic and tendon drive. The arm is shown in Figure 2. It has two joints of shoulder and forearm axis. Each joints are tensioned by opposite wire tendons respectively. The stiffness of each arm joint can be changed by adjusting the pressure of both pneumatic cylinder connected to counter wire which drive same axis.
FIGURE 2. Robot Arm

EXPERIMENTAL METHOD

When the robot arm is operating, how the movement should be needed to care the standing up motion of a person from chair. Experimentally tests by the human testers were conducted and the verification of human arm movement was carried out. The experimental angles of shoulder and forearm of the human motion are shown in the Figure 3.

FIGURE 3. Joint Action

In this Figure, θ1 is the shoulder joint angle and θ2 is the forearm angle. In the experiments, the shoulder moves from 30 ° to 50 °, on the other hand the forearm moves slightly and is almost 60 °.

Considering above result of human test, the robot arm movement is set as follow in the experiment. The initial arm joint angles are set at θ1 = 30 °, θ2 = 60 ° and then if human weight is loaded to arm, arm moves to θ1 = 50 ° and θ2 = 65 ° at the end of operation.

The main work of this report is to know the care start timing, i.e. to know the human action signal when he want to stand up. Of course, it seems to be most certain method that human or caregiver put the provided bottom when stand up, although we use the internal pressure change when human lean on the arm. We investigate the relationship between threshold level of trigger pressure change and human comfortability. The actual experimental procedure is shown as follows.
1. We settle the robot arm at $\theta_1 = 30^\circ$ and $\theta_2 = 60^\circ$.
2. Tester lean his weight on the robot arm and then the pressure increase inside the cylinder caused by load adding.
3. When the pressure exceeds a certain threshold level, the robot arm begin to moves toward $\theta_1 = 50^\circ$ and $\theta_2 = 65^\circ$ at the end point.

For simplicity, we removed tendon 2 and tendon 4 in Figure 2 then the each angle is driven appropriate single cylinder 1 and 3. The threshold values were chosen of three values of 40 kPa, 60 kPa, and 80 kPa of cylinder 1. The psychological evaluation of comfort is done by five testers. The evaluation values are a maximum of 5 points that is 5 points were most comfortable. Figure 4 shows a flowchart of the robot arm operation program.

In this experiment, it is considered that the load applied to the robot arm varies depending on the user’s posture. When threshold values were individually set for the cylinder 1 and the cylinder 3 respectively, we thought that the accurate settings of threshold values become difficult. Therefore, a threshold value is set for cylinder 1 only which is considered to apply a large load securely.

As the control method of robot arm, consideration was given to the safety of the tester, and the operation was performed with gentle PI control.

![Flowchart of the robot arm operation program](image)

**FIGURE 4.** Flowchart
Experimental results showed that the threshold of 60 kPa is most proper and the tester feels most comfortable for standing up. The 40 kPa is not well in overall, it is thought that the robot arm reaction is too excessively sensitive and the arm has too immediately operated. At 80 kPa, the threshold value is most large and the load of human part in contact with the robot arm becomes large, although the arm does not begin to work at his desired start up point then tester may feel unrest. In the case of this experiment, it can be said that there is most proper setting of threshold level of the cylinder pressure and is about 60kPa.

**CONSIDERATION**

In this study, it is shown that we can use the internal pressure change when human lean on the arm as start signal of arm motion and the most proper threshold pressure is 60 kPa in cylinder 1. In present experiment, for the convenience of test, we removed the one side of composed tendon link, although the arm works correctly because the reaction force to original position is yielded by weight of arm self in this case, but the feature of opposite tendon drive such as shown the beginning of report was lost. The study about fully linked tendon arm is planned and it is supposed that the proper threshold will be replaced by pressure deference of cylinder 1 and 2.

**ACKNOWLEDGMENTS**

I am deeply grateful to the great contribution from my friends regarding the program. Also, members of the lab will appreciate constructive opinions and support. And I thank my friends who cooperated as testers during the busy research activities.

**REFERENCES**

THE CHARACTERISTIC ANALYSIS OF WATER SPRAY COOLING COMPRESSED AIR

*Guanwei Jia\(^1,2\), Maolin Cai\(^1,2\), Yan Shi\(^1,2\), Weiqing Xu\(^1,2\), Ziyue Du\(^1,2\), Yunhua Li\(^1\), Liman Yang\(^1\), Yaoxing Shang\(^1\), Dongkai Shen\(^1\) (1. Beihang University, 2. Pneumatic and Thermodynamic energy storage and supply Beijing Key Laboratory)

5:00 PM - 5:16 PM

IMPROVEMENT OF LIFTING FORCE IN VORTEX LEVITATION BY ATTACHING A CIRCULAR COLUMN

*Yuta Yamanouchi\(^1\), Chikahisa Kawakami\(^2\), Mitsuhiro Nakao\(^1\), Minoru Fukuhara\(^1\) (1. Kagoshima University, 2. Panasonic Co., Ltd.)

5:16 PM - 5:32 PM

A NEW VACUUM GENERATOR BASED ON TORNADO-LIKE VORTEX FLOW

*Jyh-Chyang Renn\(^1\), Jian-Siang Zeng\(^1\) (1. National Yunlin University of Science and Technology)

5:32 PM - 5:48 PM

MATHEMATICAL MODELING OF A PNEUMATIC VANE MOTOR IN MATLAB/SIMULINK

*Stephan Merkelbach\(^1\), Joan Vidal Mas\(^3\), Hubertus Murrenhoff\(^1\) (1. RWTH Aachen University, Institute for Fluid Power Drives and Controls (IFAS))

5:48 PM - 6:04 PM

NUMERICAL SIMULATION OF AIR JET IMPINGEMENT FOR ARCH BREAKING IN HOPPER

*Yige Fang\(^1\), Yajun Liu\(^1\), Cunyang Zuo\(^1\) (1. South China University of Technology, Department of Mechanical and Automotive Engineering)

6:04 PM - 6:20 PM
THE CHARACTERISTIC ANALYSIS OF WATER SPRAY COOLING COMPRESSED AIR

Guanwei JIA\textsuperscript{1,2}, Maolin CAI\textsuperscript{1,2}, Yan SHI\textsuperscript{*1,2}, Weiqing XU\textsuperscript{*1,2}, Ziyue DU\textsuperscript{1,2}, Yunhua LI\textsuperscript{1}, LiMan YANG\textsuperscript{1}, Yaoxing SHANG\textsuperscript{1}, Dongkai SHEN\textsuperscript{1}

1 School of Automation Science and Electrical Engineering, Beihang University, Beijing 100191, P.R. China
2 Pneumatic and Thermodynamic energy storage and supply Beijing Key Laboratory, Beijing 100191, PR China
*Corresponding author: weiqing.xu@buaa.edu.cn and yesoyou@gmail.com

Abstract. With the extensive development of compressed air systems, the problem of reducing energy consumption in the compression system is popular. And water spray cooling compressed air is one of the effective methods, the compression process is close to the isothermal compression after water spray cooling and the compression power is reduced. In this experiment, the effect of water cooling was verified, and the influence of water spray flow on cooling effect was studied. And the experiment proves that the water spray pressure has little effect on the performance of water spray cooling. This experimental study provides a reference for the energy saving of the compression system.

Keywords: Compressed air/ Compression power/ Air temperature/ Spray pressure/ Water spray cooling

INTRODUCTION

Air compression system is widely used in metallurgy, machinery, mining, electricity, textile, petrochemical and other industries. Air compressor power consumption occupies a large proportion of energy consumption in the device, compressed air requires a high cost, it accounts for 10$\%$ to 20$\%$ of the power consumption in large industrial equipment, and some even up to 30$\%$ [1]. Therefore, saving air compressor energy consumption is an important way to reduce the overall energy consumption and product cost [2].

Spray water cooling is one of the important ways to save energy and reduce consumption in piston air compressor [3]. Spray water cooling is spray water into compressed air in the inlet or compression process. Water spray as a refrigerant direct contact with compressed air and absorbs heat, cooling efficiency is significantly higher than the external cooling method [4\textendash}5]. Effective cooling of the compressed air can make the compression process close to isothermal compression [6\textendash}7], thereby reducing the power of compression [8\textendash}9].

The air cooling method can be sorted by contact and non-contact [10]. Contact cooling mixes compressed air and coolant, the coolant directly absorbs the heat of the compressed air and then filters the coolant in the separator [11]. The non-contact cooling method cools the internal compressed air outside the compression chamber, compressed air and coolant are isolated during heat exchange.

Water spray cooling is to mix the water spray and compress the air, they exchange heat directly in the tank. This cooling method greatly increases the heat transfer area between the compressed air and the spray droplets, which reduces the heat transfer resistance between the hot and cold fluids, and the cooling effect is remarkable. A. J. White and A. J. Meacock used water spray cooling to enhance the performance of gas turbines [12]. R. K. Bhargava and his partner discussed the problem with droplet dynamics, and the factors influencing the droplet size were analyzed by experiment [13]. Sepehr Sanaye and Mojtaba Tahani studied the effect of evaporative cooling on the performance of gas turbines, and proposed the prediction equation of net power [14]. M. W. Coney et al. used a large amount of water to enter the compressor through the nozzle to achieve a quasi-isothermal compression process [15].

In this study, the water-cooling process of the compressor was studied, and the parameters such as water spray pressure, water flow rate and droplet diameter were considered, and the influence of these factors was verified by experiments.

ANALYSIS OF SPRAY COOLING COMPRESSED AIR

The water spray is injected into the compressed air, which produces heat and mass exchange between high temperature and low temperature. The surface between droplet and compressed air will form a saturated air boundary layer, where temperature is between compressed air and water [16].
Calculate the heat transfer between compressed air and surface of water spray droplets:

\[ dQ_x = \alpha (t - t_b) dF \]  

\( \alpha \) - sensible heat coefficient between surface of water spray droplets and compressed air, \( W/(m^2 \cdot K) \);
\( t \) - ambient air temperature, K;
\( t_b \) - the air temperature of the boundary layer, K;
\( dF \) - Contact surface area, m²;

Calculate the heat dissipation of the compressed air in the tank:

\[ G_r = \frac{g \beta (T_s - T_\infty) L^3}{v^2} \]  

\( g \) - gravity acceleration, N \cdot kg⁻¹;
\( \beta \) - Thermal expansion coefficient (ideal gas equal to about 1 / T), K⁻¹;
\( T_s \) - the surface temperature, K;
\( T_\infty \) - the bulk temperature, K;
\( L \) - characteristic length, m;
\( v \) - Kinematic viscosity, W / (m \cdot K⁻¹);

Calculate the convective heat transfer coefficient \( h \) of the gas tank:

\[ h = \frac{\dot{\lambda} \cdot C (G_r P_r)^\alpha}{L} \]  

\( \dot{\lambda} \) - Thermal conductivity, W \cdot m⁻¹ \cdot K⁻¹;
\( C \) - Constant of convective heat transfer;
\( G_r \) - Grashof number;
\( Pr \) - Prandtl number;
\( \alpha \) - Flow index, \( \alpha \) is 1/4 in laminar flow; \( \alpha \) is 1/3 in turbulence;
\( L \) - Length, m;

In addition, the heat loss needs to be calculated:

\[ q = h A (T_s - T_\infty) \]  

\( A \) - Contact area of spray droplets and compressed air, m²;
\( T_s \) and \( T_\infty \) have the same meaning as above.

Calculate the energy consumption of the compressor (assuming the compression process is adiabatic reversible process):

\[ W_{CS} = \frac{k}{k-1} R g T_e \left[ \left( \frac{V_1}{V_2} \right)^{k-1} - 1 \right] \]  

\( k \) - Isentropic index;
\( R \) - The molar mass of the air, kg \cdot mol⁻¹;
\( T_e \) - The temperature of the entrance, K;
\( V_1 \) - Initial volume of air, m³;
\( V_2 \) - The volume of compressed air, m³;

Calculate the power loss for water compression into the spray:

FIGURE 1. Heat and mass transfer of compressed air and water droplets.
\[ \Delta W_{a\rightarrow c} = \frac{k}{k-1} R_g (T_{\text{adiabatic}} - T_{\text{spray}}) \left[ \left( \frac{V_1}{V_2} \right)^k - 1 \right] \]  

(6)

\[ T_{\text{adiabatic}} \text{ - The inlet temperature without spray cooling, } K; \]
\[ T_{\text{spray}} \text{ - The inlet temperature with spray cooling, } K; \]

Calculate the energy saving rate using the compressed power:

\[ \varepsilon = \frac{\Delta W_{a\rightarrow c}}{W_{\text{adiabatic}}} \]  

(7)

\[ W_{\text{adiabatic}} \text{ - Adiabatic compression power, } W; \]
\[ W_{\text{spray}} \text{ - Compression power of the spray cooling, } W. \]

**EXPERIMENTAL SETUP**

Figures 2 and 3 show a schematic of a compressed air water spray cooling system, which comprises a high pressure atomization system, an air compression system and a data acquisition system.

Compressed air systems produce compressed air with ATLAS screw compressors. Adjust the compressed air outlet pressure with a pressure valve and adjust the outlet flow with the flow valve. Monitor the data with flow meters and pressure gauges. The experiment was established by SMC FLOW SWITCH PF2A751-04-27 and SMC AR60-10G.

Components of the high-pressure water atomization system include: a water tank, a water filter, a low and a high pressure water pump, pressure gauges, a water pressure regulator, and nozzles. The nozzle diameter can be adjusted from 0.1mm to 0.5mm. They produce water spray is very tiny. Use the ball valve and pressure valve to control the spray flow rate. They are installed between high pressure pumps and different nozzles.

The high-speed data acquisition system is set up on the experimental platform, which includes a 6120 USB board, it can have up to eight input channels. The data acquisition process is programmed with LABVIEW software. 6210 USB board transform temperature of the inlet and outlet water, temperature of the compressed air, pressure of gas tank, pressure of water spray to digital signals.

The gas tank is made of the 304 # stainless steel, which has a diameter of 200 millimeters and a long 500 millimeters. The heat and mass transfer between compressed air and water spray is carried out in this critical device.

**FIGURE 2.** A schematic diagram of an experimental setup system.
RESULTS

Measurement of Water Flow Rate

The spray nozzle flow is measured by weighing. The specific method is as follows:
First, collecting water in the tank. Second, start the timer and detecting the total weight of the spray on the electronic balance. Then, stop the timer. At last, the total weight of the water spray divided by total time is flow rate of water spray.

The seven nozzles produce spray with pressure of 3,4,5 MPa and the spray from nozzle to gas tank with pressure of 0.2 MPa. Figure 4 shows a schematic representation of the nozzle structure. Figure 5 show the flow characteristics of the nozzle. It can be found that the increase in nozzle diameter and inlet pressure leads to a gradual increase in flow rate.

Comparison of Experimental Results

Experiments with no water spray are compared with experiments with water spray. The experimental conditions were as follows: the ambient temperature was 19 °C and the low humidity was 24%. Set the temperature of the compressed air to 91.1 °C, 92.7°C, 93.9°C. The time of the environment and facility to reach the thermal equilibrium takes approximately half an hour. Table 1 shows the temperature difference between the inlet and the outlet and heat transfer surface area, which can calculate the heat transfer coefficient.

<table>
<thead>
<tr>
<th>Inlet temperature °C</th>
<th>Outlet temperature °C</th>
<th>Different between inlet and outlet °C</th>
<th>Heat transfer coefficient W/(m²·°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>93.9</td>
<td>51.1</td>
<td>42.9</td>
<td>256.3</td>
</tr>
<tr>
<td>92.7</td>
<td>48.3</td>
<td>44.4</td>
<td>256.6</td>
</tr>
<tr>
<td>91.1</td>
<td>47.4</td>
<td>43.4</td>
<td>259.8</td>
</tr>
</tbody>
</table>
The experimental conditions for water spray are the same as those without water spray above. The temperature of ambient is 19-20 °C, relative humidity is about 20%-30%, The effluent water temperature of high pressure water atomization system is 24 °C. Adjust the pressure of the water with the ball valve. The data acquisition system collects pressure and temperature signals. The water spray flow has a direction which opposite from compressed air. This opposite flow direction enhances the heat transfer between the water spray and the compressed air.

As shown in Fig.6, at a given inlet temperature and compressed air flow rate, the outlet temperature of compressed air is reduced gradually with the increase in water spray flow rate. The outlet temperature curves overlapping each other reveal that generated pressure of water spray has little influence on the outlet temperature.

![Inlet and outlet temperature of compressed air at different flow rate](image)

**FIGURE 6.** Inlet and outlet temperature of compressed air at different water spray flow rate

It is proved that the increase of the flow rate leads to the increase of the temperature difference. Therefore, increasing the water flow rate to improve the cooling efficiency is an effective way.

**DISCUSSION ON COMPRESSION POWER**

Figure 7 shows a schematic diagram of a two-stage compression system and a water atomization system. Water spray injection system is a water injection between two stages. Air into the compression chamber is compressed, and then into the gas tank to cool, and finally into the compressor.

The experimental steps are as follows:

First, assuming that the compression process is adiabatic compression, the pressure of the compressed air in the first-stage compressor becomes 0.2 MPa and the temperature becomes 90 °C. And then the compressed air will enter the tank, compressed air and water spray contact for heat transfer, water spray will affect the temperature of compressed air. The inlet temperature of the compressed air varies with the water spray in the second stage compressor. Therefore, the compression power becomes smaller due to the effect of water cooling in the second stage of compression. As shown in Figure 8. And as the water pressure increases the compression power will gradually decrease.

![Two-stage compression system](image)

**FIGURE 7.** Two-stage compression system.
As shown in Fig. 9, the power used to compress the air is reduced with the flow rate of water spray increases, due to the cooling effect of the water spray. But as shown in Figure 13 is the system power is not always reduced when the water flow is increased. Because the system power includes two parts, water atomization power and compressed air power. When the power of the water atomization is greater than the power saved by the water spray cooling, the power saving rate will be below the reference value. In the case of a nozzle diameter of 0.4 mm and a water spray pressure of 5 MPa, the compression power is reduced by 23%.

**CONCLUSIONS**

Experiments show the characteristic of water spray cooling compressed air, and the following conclusions are obtained:

1. An increase in nozzle diameter leads to an increase in water flow;
2. The increase in water inlet pressure can lead to an increase in water flow;
3. The water inlet pressure does not have a significant effect on cooling the compressed air.
4. Experimental use of the nozzle with water pressure of 5MPa and diameter of 0.4mm, compression power reduced by 23%.
REFERENCES

IMPROVEMENT OF LIFTING FORCE IN VORTEX LEVITATION BY ATTACHING A CIRCULAR COLUMN

Yuta YAMANOUCHI*, Chikahisa KAWAKAMI**, Mitsuhiro NAKAO* and Minoru FUKUHARA*

*Department of Mechanical Engineering, Graduate School of Scientific and Engineering, Kagoshima University
1-21-40 Korimoto, Kagoshima-shi, Kagoshima, 890-0065 Japan
(E-mail: nakao@mech.kagoshima-u.ac.jp)
** Panasonic Co., Ltd.
1-1-1 Sakata, Oizumi-machi, Ora-gun, Gunma, 370-0596 Japan

Abstract. Vortex levitation is a pneumatic device attaining non-contact handling by injecting air through a tangential nozzle into a vortex chamber. In this paper, we deal with a fundamental characteristic of a vortex cup installed a thin column. According to the experimental set up, the sucking forces and the pressure distributions on the workpiece of the vortex levitations using a traditional type cup and a cup with the column were measured at three kinds of inlet mass flow rates. The result suggested that the sucking force of the cup with the column becomes larger than that of the traditional cup at a certain flow rate. Numerical simulations were conducted to investigate flow field inside the vortex cups. The result suggested that the eccentricity of the center of the swirling flow becomes smaller, and the swirling velocity becomes faster by installing the column.

Keywords: Vortex levitation, Swirling flow, Lifting force, CFD

INTRODUCTION

In semiconductor manufacturing process, it is necessary to protect workpieces being transported from damage and contamination. However, traditional contact type conveyance is unsuitable for this purpose. Therefore, non-contact handling device has attracted attention in recent years.

Several methods such as magnetic, ultrasonic and pneumatic have been developed for the non-contact conveying. The magnetic method has a problem that the workpiece is limited to only the magnetic material, and the ultrasonic method has a problem that the maintenance cost is high [1]. On the other hand, because of its simple structure and operation, the pneumatic method provides minimal maintenance and does not require a control loop to obtain a stable state [2]. Bernoulli levitation is a most popular pneumatic method based on Bernoulli’s theorem. Vortex levitation is another type of the pneumatic non-contact handling methods, and has low air consumption compared with the sucking power compared with the Bernoulli levitation [3].

The vortex levitation uses a cylindrical cup, called vortex cup, as shown in FIGURE 1. The vortex cup is composed of a vortex chamber generating a swirling flow, a nozzle connected with the chamber in the tangential direction, and a skirt part at the bottom. Compressed air is supplied from the nozzle and is injected tangentially into the vortex chamber. Then, ejected air rotates along the wall surface to form a swirling flow, so that centrifugal force is generated. The force makes the pressure near the center in the vortex chamber low. Since pressure loss occurs in the gap flow, positive pressure is generated in the skirt part. As opposed to the negative pressure, this positive pressure acts as a force for pushing out workpieces and the difference between the negative pressure and the positive pressure is the resultant lifting force. The swirling flow velocity becomes faster as the lifting force improves. In general, the flow velocity becomes faster against the same flow rate when flow path area is small. Therefore, if a vortex cup having a column in the vortex chamber is used, the lifting force should be large; however, the lifting force is decreased in a previous study [4]. The authors thought that there is a possibility of improving the lifting force since it was not studied sufficiently in detail.

In this study, we investigated characteristics of a vortex levitation using a vortex cup with a thin column. The sucking power and the pressure distribution on the workpiece were experimentally investigated against three mass flow rates. In addition, we investigated the influence of the installed column on the swirling flow by numerical calculation using CFD.
EXPERIMENT

We employ two types of vortex cups, one of the cup is traditional type and the other cup has a column, as shown in FIGURE 2. The dimensions of the cups are cylindrical diameter $D = 23$ mm, nozzle position $L = 7.20$ mm, cylindrical chamber height $H = 8.05$ mm, cup inner edge angle is 30 degrees. The sizes of the cups were determined by trial and error process. The height and the diameter of the column installed in the vortex chamber is $h = 6$ mm and $d = 4$ mm, respectively. Experimental measurements were performed at three kind of supply flow rate; $1.95 \times 10^{-4}$ kg/s, $3.26 \times 10^{-4}$ kg/s, $4.94 \times 10^{-4}$ kg/s. An experimental apparatus for measuring the lifting force is shown in FIGURE 3. This apparatus consists of a Z stage for setting the gap height and an electronic balance for measuring the lifting force. The clearance height at which measurement starts is set to 0.2 mm, and the Z stage is operated to increase the clearance height to 1.0 mm. The lifting force was measured by means of the electronic balance. FIGURE 4 shows an experimental apparatus for measuring pressure distribution on the workpiece. The pressures were measured with a differential pressure sensor of semiconductor type.

FIGURE 2. Sketch of the vortex cups (left: the cup of the traditional type, right: the sup with the column).

FIGURE 3. Experimental apparatus for measuring the lifting force.
FIGURE 4. Experimental apparatus for measuring pressure distribution on the workpiece.

![Experimental apparatus for measuring pressure distribution on the workpiece.](image)

FIGURE 5. Comparison of the lifting forces against the gap heights in three mass flow rates.

(a) $1.95 \times 10^{-4}$ kg/s

(b) $3.26 \times 10^{-4}$ kg/s

(c) $4.94 \times 10^{-4}$ kg/s

TABLE 1. Maximum lifting forces against the three flow rates.

<table>
<thead>
<tr>
<th>Mass Flow Rate</th>
<th>Cup without column</th>
<th>Cup with the column</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.95 kg/s</td>
<td>0.191</td>
<td>0.189</td>
</tr>
<tr>
<td>3.26 kg/s</td>
<td>0.481</td>
<td>0.502</td>
</tr>
<tr>
<td>4.94 kg/s</td>
<td>0.919</td>
<td>0.928</td>
</tr>
</tbody>
</table>
FIGURE 6. Line on which the pressures and the velocities were obtained.

FIGURE 7. Comparison of the pressure distribution between the two cups.

FIGURE 5 shows the lifting forces against the gap heights in the three mass flow rates. TABLE 1 shows the maximum lifting forces against the three flow rates. Each lifting force increases with the increasing the gap height until the maximum lifting forces are obtained. After that, the lifting forces are gradually decreased as the gap height increased.

When the mass flow rates are $1.95 \times 10^{-4}$ kg/s and $4.94 \times 10^{-4}$ kg/s, the maximum lifting forces of the cup with the column are as equal to that of the traditional cup. On the other hands, when the mass flow rate is $3.26 \times 10^{-4}$ kg/s, the maximum lifting force of the cup with the column becomes about 5% larger than that of the traditional cup. In all three mass flow rates, the gap heights of the cup with the column, obtaining the maximum lifting forces, are smaller than that of the traditional cup.

FIGURE 6 shows a line on which the pressures and the velocities were obtained for comparison. The velocities were compared only in the case of numerical simulation. FIGURE 7 shows Comparison of the pressure distribution between the two cups when the mass flow rate is $3.26 \times 10^{-4}$ kg/s. As we might have surmised from FIGURE 5, the pressures around the center of the cup with the column are smaller than those of the cup without the column.

**NUMERICAL SIMULATION**

The above experiment has demonstrated that the maximum lifting force is increased by attaching the column in the vortex chamber when the inlet mass flow rate is $3.26 \times 10^{-4}$ kg/s. However, it is unknown how the flow field inside the vortex chamber changes due to the installation of the column, and how it influences the lifting force. Therefore, in order to clarify the effect of the column to the flow field and the sucking force, a numerical simulation was performed.

FIGURE 8 shows the computation grids of the vortex cup and the vortex cup with the column to be analyzed. The size of the vortex chamber and both of the height, $h$, and the diameter, $d$, of the column were set at the same values as the experiment. The gap height was set to the value at which almost the maximum lifting force was obtained; 0.4 mm in the traditional cup, 0.3 mm in the cup with the column. The number of the grid points is
23,3078 in the case of the cup with the column, and the number of the grid points in the case of the cup without the column is 226,526. There are three types of boundary conditions; inlet, outlet, and wall. We set the nozzle inlet as the inlet region, the outer side of the gap as the outlet region, and the other region as the wall region. The working fluid is air of ideal gas in turbulent flow. As the turbulent model, we employed the standard $k$-$\varepsilon$ mode for stable and efficient calculation. We set the mass flow rate to the three mass flow rates, the temperature of the fluid to 300 K, the turbulent kinetic energy, $k$, to 1,095,558, and the dissipation rate of $k$, $\varepsilon$, to 108.1 m$^2$/s$^3$ for the boundary condition of the inlet region, and the outlet pressure to 101300 Pa. The other variables were considered to be zero-gradient for the boundary conditions.

![FIGURE 8. Computational Grids.](image)

![FIGURE 9. Comparison of the pressure distributions between the numerical simulation and the experiment.](image)

| Table 2. Lifting forces computed with the simulation when $G = 3.26 \times 10^{-4}$ kg/s |
|---------------------------------|---------------------------------|
| **Cup without the column**      | **Cup with the column**         |
| 0.16 N                          | 0.17 N                          |

![FIGURE 10. Computed streamlines inside the vortex chambers.](image)
In order to verify the validity of the numerical simulation, the pressure distribution on the line at the bottom of the gap were compared between the experiment and the numerical simulation in FIGURE 9. The inlet mass flow rates were set to the three mass flow rates. Both graphs in FIGURE 9 demonstrate that the pressure distributions does not match perfectly, but coincide roughly. Therefore, the results of the numerical simulations are considered to be approximately reliable. The lifting force is calculated by numerically integrating differential pressure between the upper surface and the back surface of the workpiece. The pressure on the back surface on the workpiece is considered to be atmospheric pressure. The lifting forces obtained with the numerical simulation were calculated when \( G = 3.26 \times 10^{-4} \) kg/s, and were arranged in TABLE 2. The lifting force of the cup with the column is about 8 % larger than that of the cup without column. The lifting forces calculated from the numerical simulation are roughly agreed with those of experiments.

In order to investigate whether the setting of the column actually affects the swirling flow velocity inside the vortex chamber, the streamlines inside the vortex cups when \( G = 3.26 \times 10^{-4} \) kg/s are illustrated in FIGURE 10, and the velocity distributions on the line are illustrated in FIGURE 11. In the case of the cup without column, the centers of the swirling fluid inside the cup are biased. On the other hand, the swirling fluid has smaller bias. This trend is the same with the previous study [4]. FIGURE 11 shows the velocity of the cup with the column is larger than that of the traditional cup. These results dovetailed with the fact that the swirling flow velocity becomes faster as the lifting force improves.

CONCLUSION

We investigated characteristics of a vortex levitation using the vortex cup with the thin column with the diameter of 4 mm, and the length of 6 mm. The sucking force and the pressure distribution on the workpiece were experimentally investigated against three mass flow rates. Only when the inlet mass flow rate is \( 3.26 \times 10^{-4} \) kg/s, the sucking force of the cup with the column is about 5 % larger than that of the traditional cup. The numerical simulation suggested that the inner velocity inside the cup with the column becomes faster than that of the traditional cup.

REFERENCES

A NEW VACUUM GENERATOR BASED ON TORNADO-LIKE
VOXERX FLOW

Jyh-Chyang RENN* and Jian-Siang ZENG**

*Department of Mechanical Engineering, National Yunlin University of Science and Technology
123 University Road, Section 3, Douliou, Yunlin, 64002 Taiwan
(E-mail: rennjc@yuntech.edu.tw)

**Department of Mechanical Engineering, National Yunlin University of Science and Technology
123 University Road, Section 3, Douliou, Yunlin, 64002 Taiwan

Abstract. In this paper, a brand new design to generate the vacuum effect is proposed. The basic idea is to imitate the geometry of a tornado. It is well-known that tornado is one of the severest natural disasters in the world. However, tornado is also able to generate a huge vacuum suction force to lift cars and even houses on the ground into the air. Using CFD simulations, a novel vacuum generator based on tornado-like vortex flow is successfully designed and implemented. This new structure is totally different from the traditional vacuum generator based on the Bernoulli’s equation. Finally, two traditional nozzle-type vacuum generators are chosen for the purpose of performance comparing. Experimental studies prove that the new tornado-like vacuum generator can generate acceptably large vacuum pressure while the consumed volumetric airflow-rate is kept at a low level.

Keywords: Pneumatics, Vacuum Generator, CFD, Tornado, Vortex Flow.

INTRODUCTION

In the middle and west area of United States, tornado is perhaps the severest natural disaster because the damage caused by tornados cannot be estimated every year. It is also observed that the geometry of a tornado is similar to a funnel as shown in Fig. 1. That is, the diameter of the tornado is smallest near the ground surface. On the other hand, tornado is also able to generate a huge vacuum suction force to lift cars and even houses on the ground into the air. In this paper, a brand new design to generate the vacuum effect is proposed. The basic idea is to imitate the geometry of a tornado. Such a novel design is totally different from the traditional vacuum generator which is based on the Bernoulli’s equation. In details, traditional vacuum generator utilizes nozzles to produce the airflow jet. The pressure will significantly decrease if the velocity of airflow jet increases. One typical structure is shown in Fig. 2. However, such a technique is quite old though it is still accepted by engineers nowadays. One significant feature of this study is the utilization of CFD-simulation to find the most suitable structure and dimension for a novel tornado-like vacuum generator. An experimental test rig is also constructed to verify the design. In the following, the design using the CFD-simulations will firstly be outlined.

FIGURE 1. (a) A tornado, (b) Funnel shape structure to imitate a tornado.
DESIGN BY CFD-SIMULATIONS

Figure 3 shows the most important design parameters for the CFD simulations, which include the number of eccentric inlet ports, the diameter of the inlet restrictor ($\varphi_r$), the diameter of the vacuum pressure port ($\varphi_v$), the inclined angle of inlet port ($\alpha$), the expansion angle of the tornado-like funnel ($\beta$) and the height of vortex flow ($H$). In addition, only one parameter is adjusted at a time while the rest parameters are kept unchanged. Thus, after numerous CFD simulations, a most suitable design configuration for the tornado-like vacuum generator can be determined as shown in Fig. 4. It is observed that two eccentric inlet ports at both sides and one vacuum pressure port at the bottom are constructed. The former is used to generate the vortex flow. From previous literatures [1-4], it was already proved that the vortex flow can be used to develop non-contact pneumatic suction pads. In this paper, however, the flow path of the vortex is guided to move upwards to establish a flow field which is very similar to a tornado. Table 1 summarizes the corresponding values of design parameters. In details, two inlet ports with restrictor diameter $\varphi_r = 1.0$ mm, the vacuum pressure port diameter $\varphi_v = 2.0$ mm, inclined inlet angle $\alpha = 15^\circ$, funnel expansion angle $\beta = 5^\circ$ and the height of vortex flow $H = 35$ mm are found to be the most suitable structure and dimensions. Figure 5 depicts an example of pressure distribution of the most suitable design using CFD simulation. It is observed that the negative pressure, which is generally called the vacuum pressure, occurs at the vacuum pressure port. The real picture of the developed prototype is shown in Fig. 6. Moreover, to verify this design, a test rig will be established and a series of experiments will be conducted in the following sections.
FIGURE 4. Most suitable design configuration for the tornado-like vacuum generator.

FIGURE 5. An example of pressure distribution of the most suitable design using CFD simulation.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of inlet port</td>
<td>2</td>
</tr>
<tr>
<td>Inclined inlet angle $\alpha$</td>
<td>$15^\circ$</td>
</tr>
<tr>
<td>Funnel expansion angle $\beta$</td>
<td>$5^\circ$</td>
</tr>
<tr>
<td>Diameter of inlet restrictor $d_r$</td>
<td>1.0 mm</td>
</tr>
<tr>
<td>Diameter of vacuum port $d_v$</td>
<td>2.0 mm</td>
</tr>
<tr>
<td>Height of vortex flow $H$</td>
<td>35 mm</td>
</tr>
</tbody>
</table>

TABLE 1. Values for the most suitable design parameters.
FIGURE 6. Real Picture of the developed prototype.

EXPERIMENTAL TEST RIG

To test the performance of the developed vacuum generator based on tornado-like vortex flow, a test rig is constructed and its scheme is shown in Fig. 7. The inlet pressure to the vacuum generator can be adjusted from 0.5 to 10 bar by manually adjusting the pressure reducing valve (No. 4). Besides, a vacuum gauge (No. 10) is utilized to measure the generated vacuum pressure and a flow-rate meter (No. 8) is used to record the consumed volumetric airflow-rate.

FIGURE 7. Scheme of the test rig.

TEST RESULT AND DISCUSSION

First of all, two traditional nozzle-type vacuum generators with type number AVD-15H5S (denoted as Traditional-I) and AV-15H5SJ (denoted as Traditional-II) are chosen as shown in Fig. 8. The generated vacuum pressure and the consumed volumetric airflow-rate are two indices for comparing the performance. The former is obviously the key performance, and the latter represents the degree of energy efficiency. Generally speaking, the absolute value of generated vacuum pressure is higher corresponding to a larger inlet pressure. However, the consumed volumetric airflow-rate will also be higher meaning that more power is needed. From the viewpoint of generated vacuum pressure, as shown in Fig. 9 (a), the performance of the proposed new vacuum generator is better than Traditional-II, but worse than Traditional-I. Moreover, form the viewpoint of energy efficiency or consumed volumetric airflow-rate as shown in Fig. 9 (b), the proposed new vacuum generator is the best among three tested vacuum generators. It is worth mentioning that even though the proposed new vacuum generator is
not the best regarding the key performance, the absolute value of vacuum pressure of 60 kPa at the inlet pressure of 5 bar is good enough for real industrial applications.

**FIGURE 8.** Two traditional nozzle-type vacuum generators chosen for comparison, (a) Traditional-I; (b) Traditional-II.

**FIGURE 9.** (a) Comparison of generated vacuum pressure, (b) Comparison of consumed airflow-rate.

**CONCLUSION**

In this paper, both simulation and experimental studies prove that the new tornado-like vacuum generator can generate acceptably large vacuum pressure while the consumed volumetric airflow-rate is kept at a low level. However, it is still possible to do more research and apply other optimal design criterion in this area to further improve the performance in the future. Nevertheless, it can already be concluded that the currently developed vacuum generator based on tornado-like vortex flow is not only a successful new idea but also a potential alternative for future vacuum generator design. It is expected that such a novel tornado-like vacuum generator can find some potential applications in the real industry.

**ACKNOWLEDGMENTS**

The financial supports of the National Science Council under grant number MOST 105- 2622- E- 224- 016- CC3 is greatly appreciated.

**REFERENCES**

MATHEMATICAL MODELING OF A PNEUMATIC VANE MOTOR IN MATLAB/SIMULINK

Stephan MERKELBACH*, Joan Vidal MAS* and Hubertus MURRENHOFF*

*Institute for Fluid Power Drives and Controls (IFAS), RWTH Aachen University
Steinbachstr. 53, 52074 Aachen, Germany
(E-mail: stephan.merkelbach@ifas.rwth-aachen.de)

Abstract. Air driven motors are used in a variety of applications, for example as drives for tools in manufacturing. Pneumatic vane motors in particular feature high speed and eminent power density. In contrast to the large number of applications, only few efforts regarding dynamic simulation of these motors have been made in the past. Due to this lack of simulation studies concerning pneumatic rotating equipment, the paper presents the development and experimental validation of a one dimensional simulation model for pneumatic vane motors in Matlab/Simulink. The model developed in the study can be used as a basis for future work on the optimization and control of pneumatic vane motors. Additionally, the model may be included in lumped parameter simulation environments to improve their accuracy in the examination of pneumatically driven rotating equipment. The validation is carried out by an experimental setup which consists of the vane motor mounted on a common shaft with a hydraulic gear pump building up the load. The paper presents a validated, one-dimensional model of the dynamic behavior of a pneumatic vane motor. This is of great importance for the simulation of rotating equipment in manufacturing driven by pneumatic vane motors. [1]

Keywords: Pneumatics, Vane Motor, Simulation

INTRODUCTION

Pneumatic vane motors are widely used in industrial applications where robustness, high power-to-weight-ratio or indifference to overload are of importance. Up to today, there are only few examples for the mathematical modelling of these motors known in literature. Most models cover only a limited bandwidth of physical effects appearing in pneumatic vane motors. Beater [2] proposes a simple model in Modelica for the dynamic behavior meant for the use in lumped parameter simulation programs. The presented model does explicitly not feature the dynamic oscillations of the driving torque, as experimentally described, e.g., by Ioannidis [1]. Another model without detailed examination of friction and leakage effects is presented by Luo [3]. Badr established a detailed mathematical description for leakage [5] as well as friction effects [6] but does not present any implementation in a simulation environment. Additionally, there are some models for vane-expanders used especially in cooling applications (cf. e.g. [4]). In these applications most effort is put into the modelling of the multi-phase flow which usually does not occur in pneumatic motors. In literature, there is some work on the dynamic analysis of rotary vane combustion engines, which can in part be used for the analysis of the dynamic behavior of pneumatic motors as well. Examples were published by Librovich et. al. in [8–10].

The Matlab/Simulink-model described in the paper gives an overview of the dynamic behavior of pneumatic vane motors including friction and leakage losses dependent on geometry, working pressure and load. Especially the modeling of leakage has deep impact on the accuracy of the simulation results. The model includes a dynamic simulation of the rotating parts of the motor including the vanes. Pressure and temperature inside the motor chambers are simulated in dependence of the rotating angle and, therefore, the volume of each chamber, the heat transfer to the environment as well as the leakage between chambers can be evaluated.

There are different paths at which internal leakage between chambers occurs (cf. FIGURE 1). TABLE 1 gives an additional overview of the main leakage paths. [4]
The paper shows an estimation of the dimension of the mass flow through these paths for normal operation derived from literature. The main losses due to leakage during normal operation are calculated.

The friction torque is calculated based on the forces acting inside the motor. The friction forces between the vanes and the stator are highly dependent on the angular velocity of the motor as well as the angular position of the vanes. These dependencies and the resulting friction torque are considered in the model. An additional source of friction losses is the contact line between rotor and stator.

<table>
<thead>
<tr>
<th>TABLE 1 Primary leakage paths</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1)</td>
</tr>
<tr>
<td>(2)</td>
</tr>
<tr>
<td>(3)</td>
</tr>
<tr>
<td>(4)</td>
</tr>
<tr>
<td>(5)</td>
</tr>
</tbody>
</table>

The driving torque is calculated as the difference between the torque generated by the pressure differences on the vanes and the friction torque acting against the movement. If the driving torque is higher than the external load, the motor accelerates. The rotating motor can be decelerated by a load torque higher than the driving torque. At stand still, a load higher than the driving torque is interpreted as static friction. Therefore, in this case, the motor stays in rest. To evaluate the angular acceleration, the variable moment of inertia is calculated in dependence of the position of the vanes.

The pressure build-up and reduction in the motor chambers is modeled as a function of the chamber geometry and the heat transfer during the rotation of the motor as well as leakage mass flow out of and into the chamber between the vanes.

As the air temperature has a high impact on pneumatic motors to avoid water condensation or icing at the outlet it is calculated during the rotational movement. This includes a model of the heat transfer between the air and the stator in dependence of the rotating velocity and the inlet conditions of the air. Multi-phase flow is not considered in the model to reduce computing time.

MATHEMATICAL MODELING

The mathematical model of the vane motor is split up in five subparts. First, a geometrical model of the motor chambers is developed. Afterwards, the pressure build-up is modeled in dependency of the chamber volume, the mass flow into and out of each chamber. To generate a valid estimation of the air mass flow, modelling of the leakage through the different paths is necessary. Once the pressure inside each chamber is known, it is possible to estimate the driving torque generated by each vane. As friction losses are very high in pneumatic vane motors, it is essential to model the friction torque in dependency of the rotary position and velocity. The modelling of these subsystems is illustrated in the following. Afterwards, specific results of the simulations carried out in the study are shown.

Geometrical Model

FIGURE 2 shows the cross sectional area $A_{ch}$ of one chamber of the motor. To evaluate the change of volume during the rotational movement, the distance $R_{ch}(\phi)$ between the rotational axis of the rotor and the inner wall of the stator has to be calculated as a function of the rotational angle $\phi$ shown in FIGURE 2 (c). The calculation of $A_{ch}$ is conducted under the assumption that the contact point between vane and stator always lies in the tip of the vane.
**FIGURE 2** Calculation of the cross sectional area between two vanes

$R_V(\varphi)$ can be calculated by equation (1) with the eccentricity $e$, the rotational angle $\varphi$ and the stator radius $R_S$.

$$R_V(\varphi) = -e \cdot \cos(\varphi) + \sqrt{R_S^2 - (e \cdot \sin(\varphi))^2}$$  \hspace{1cm} (1)

First, the area $A_{vect}$ between vanes 1 and 2 depicted in purple in **FIGURE 2** (b) is calculated by equation (2).

$$A_{vect} = \frac{1}{2} \int_{\varphi_1}^{\varphi_2} R_V(\varphi)^2 d\varphi$$  \hspace{1cm} (2)

Due to the square root term in equation (1), equation (2) has two different solutions depending on the angle $\varphi$ as shown in **FIGURE 2** (c). Equations (3) and (4) show the “positive” and “negative” solution for $R_V^2(\varphi)$ which then are inserted in equation (2).

$$R_V^2(\varphi)_+ = e^2 \cdot \cos(2\varphi) + R_S^2 + 2 \cdot e \cdot \sqrt{R_S^2 \cdot \left(\frac{1 + \cos(2\varphi)}{2}\right) - e^2 \cdot \left(\frac{1}{8} - \frac{\cos(4\varphi)}{8}\right)}$$  \hspace{1cm} (3)

$$R_V^2(\varphi)_- = e^2 \cdot \cos(2\varphi) + R_S^2 - 2 \cdot e \cdot \sqrt{R_S^2 \cdot \left(\frac{1 + \cos(2\varphi)}{2}\right) - e^2 \cdot \left(\frac{1}{8} - \frac{\cos(4\varphi)}{8}\right)}$$  \hspace{1cm} (4)

Afterwards, the area $A_{rot} = \frac{R_R^2 \pi}{N}$ between two vanes on the rotor (hatched orange in **FIGURE 2** (b)), calculated with the number of vanes $N$, and the area of the vanes marked in yellow in **FIGURE 2** (a) are subtracted from the area calculated in equation (2).

Multiplication with the length of the stator leads to the slope of the chamber volume shown in **FIGURE 3** for a motor with 8 vanes.

**FIGURE 3** Chamber volume depending on the rotation angle
Pressure and temperature model

The pressure change in each chamber is a function of the volume change of the chamber, the mass flow into and out of the chamber as well as the air temperature. To avoid CFD-modelling of the chambers, the state variables pressure and temperature inside each chamber are considered quasi stationary and location-independent within one chamber. **FIGURE 4** shows a schematic of the p-V-diagram for one chamber during the rotation of the rotor.

As the air mass inside the chamber is not constant due to leakage, the slope from 'B' to 'C' and from 'D' to 'E' cannot be modeled using the polytropic equations. Therefore, the pressure change \( \dot{p}_i \) in each chamber is calculated using equation (5). [7]

\[
\dot{p}_i = \dot{p}_{m_i} + \dot{p}_{HE,i} + \dot{p}_{V_i}
\]  

\[ (5) \]

It is the sum of the pressure change \( \dot{p}_{m_i} \) due to mass flow into and out of each chamber, the pressure change \( \dot{p}_{HE} \) due to the heat exchange between the air and the stator and the pressure change \( \dot{p}_{V} \) caused by the volume change of each chamber. These terms can be calculated according to equations (6) to (8).

\[
\dot{p}_{m_i} = \frac{k - 1}{V} \cdot R \cdot \left[ \sum (\dot{m}_{in,i} \cdot T_{in,i}) + \sum (\dot{m}_{out,i} \cdot T_{out,i}) \right]
\]  

\[ (6) \]

\[
\dot{p}_{HE,i} = \frac{k - 1}{V} \cdot [\alpha \cdot (T_{stator} - T_i) \cdot A_{wall}]
\]  

\[ (7) \]

\[
\dot{p}_{V_i} = -\frac{k}{V} \cdot p_i \cdot V_i
\]  

\[ (8) \]

\( A_{wall} \) describes the area on the stator between two corresponding vanes. To reduce computing time, it is approximated by the mean area (equation (9) with the number of vanes \( N \)) and considered constant for the complete revolution of the rotor. The stator temperature \( T_{stator} \) has the constant value of the environment and therewith the supply air temperature.

\[
A_{wall} = \left( \frac{2\pi \cdot R_S + R_R}{N} \right) \cdot L
\]  

\[ (9) \]

The Nusselt number \( Nu \) delivers the heat transfer coefficient \( \alpha \). Using the Dittus-Boelter-equation, \( Nu \) can be calculated from the Reynolds number \( Re \) and the Prandtl number \( Pr \) according to equation (10).

\[
Nu = \alpha \cdot L/\lambda_{air} = 0.023 \cdot Re^{0.8} \cdot Pr^{0.3}
\]  

\[ (10) \]

The temperature within each chamber is calculated by partially deriving the ideal gas equation to all time dependent terms (equation (11)).
According to [1] the pressure inside one chamber is modeled as a PT$_1$-element when the chamber reaches the inlet to avoid a pressure jump. This approach is common for the numerical simulation of valves [7]. When the chamber pressure has reached the system pressure, its value is considered constant until the following vane leaves the inlet area.

**FIGURE 5** shows the pressure in one chamber during one revolution of the motor, computed with the equations mentioned above without the consideration of leakage losses and gains. Leakage mass flows are included in the total mass flow $\dot{m}_i$ later on. The results are in good accordance to the experimental results shown by Ioannidis [1].

![Graph showing pressure over time](image)

**FIGURE 5** Slope of the pressure in one chamber without leakage mass flow

### Leakage air flow

There are different paths through which air can leak between two adjacent chambers or from one chamber to the outlet of the motor (cf. **TABLE 1, FIGURE 1**) [5]. The different paths have different influences on the total leakage flow. Therefore, not all of the paths are actually modeled to reduce computing time. Due to the very small gap between the vanes and the slots in the rotor, the leakage flow through path (5) is neglected. The gap between the rotor and stator is very small while being relatively long. Additionally, the movement of the vanes and the rotor induces a flow contrary to the pressure difference. This leads to the conclusion that path (4) may be neglected as well.

The leakage mass flow through the gaps between the vanes and the stator (paths (2) and (3)) are calculated as one single ideal nozzle with a constant height of 0.1 mm to reduce computing time according to equation (12).

$$ m_{\text{leak, noz}} = v_2 \cdot A_2 \cdot \rho_2 $$  \hspace{1cm} (12)

Inserting the first law of thermodynamics to compute the air velocity $v_2$ inside the gap for each time-step from the pressure values $p_2$ and $p_1$ inside the adjacent chambers leads to equation (13) with area $A_2$ considered constant with a height of 0.1 mm and the length of the rotor. The density $\rho_2$ is derived from the ideal gas equation. [7]

$$ m_{\text{leak, noz}} = 0.1 \text{ mm} \cdot \text{L} \cdot \sqrt{2 \cdot \frac{k}{k - 1} \cdot p_1 p_1 \left( \frac{p_2}{p_1} \right)^\frac{2}{k} - \left( \frac{p_2}{p_1} \right)^\frac{k+1}{k} } $$  \hspace{1cm} (13)

In addition to the leakage flow through the gaps between the vanes and the stator (paths (2) and (3) considered above), air can leak between the rotor and the stator (path (1) in **FIGURE 1**). These losses can be modeled as an air flow through the gap between a rotating and a stationary disk. The mathematical modeling of this flow is described by Bein in [11]. The mass flow can subsequently be calculated with equation (14) with the two Fourier-coefficients, as derived in [5], described in equation (15) and (16).

$$ m_{\text{leak, r-s}} = - \frac{(p_1 - p_2) \cdot \varepsilon_1^3}{6 \cdot v_m} \sum_{k=1}^{\infty} \left( 1 - \beta^{2k} (A_k \cdot \sin(k \varphi) - \sin(k \varphi - \delta)) 
- B_k \cdot (\cos(k \varphi) - \cos(k \varphi - \delta)) \right) $  \hspace{1cm} (14)
\begin{equation}
A_k = \frac{1}{k^{2 \pi (1 + \beta_k^2)}} \left( \frac{1}{\phi_{\text{out}} - \phi_{\text{in}} - \delta} \left[ \cos(k \phi_{\text{out}}) - \cos(k(\phi_{\text{in}} - \delta)) \right] - \frac{1}{2\pi - \phi_{\text{out}}} \left[ 1 - \cos(k \phi_{\text{out}}) \right] \right) \tag{15}
\end{equation}

\begin{equation}
B_k = \frac{1}{k^{2 \pi (1 + \beta_k^2)}} \left( \frac{1}{\phi_{\text{out}} - \phi_{\text{in}} - \delta} \left[ \sin(k \phi_{\text{out}}) - \sin(k(\phi_{\text{in}} - \delta)) \right] - \frac{1}{2\pi - \phi_{\text{out}}} \left[ 1 - \sin(k \phi_{\text{out}}) \right] \right) \tag{16}
\end{equation}

**Dynamic model of the rotor-vane-system**

To simulate the dynamic behavior of the rotor, first it is mandatory to model the inertia \( J(\phi) \) of the rotating parts, i.e., the rotor and the vanes, in dependency of the angular position.

Using equation (17) the angular acceleration of can be calculated for each time step.

\begin{equation}
\ddot{\phi}(t) = \frac{M_{\text{tot}}(t)}{J(\phi)} \tag{17}
\end{equation}

As the center of gravity of each vane does not fall into the center of gravity of the rotating system, their inertia has to be calculated using Steiner’s law (mass \( m_v \), width \( a_v \) and length \( b_v \)). This leads to equation (18) for the total moment of inertia of the rotating parts.

\begin{equation}
J(\phi) = J_r - \sum_{i=1}^{N} J_{v,i}(\phi) = \frac{1}{2} m_r \cdot R_r^2 + \sum_{i=1}^{N} \left( \frac{1}{12} m_v (a_v^2 + b_v^2) + m_v d_{\text{COG},i}^2 \right) \tag{18}
\end{equation}

The radial position of each vane’s center of gravity \( d_{\text{COG},i} \) is calculated by equation (19).

\begin{equation}
d_{\text{COG},i} = R_{v,i}(\phi) - \frac{b}{2} = -e \cdot \cos(\phi) + \sqrt{R_v^2 - (e \cdot \sin(\phi))^2} - \frac{b}{2} \tag{19}
\end{equation}

The total torque working on the rotor is calculated using equation (20)

\begin{equation}
M_{\text{tot}}(\phi) = M_p(\phi) - M_{fr,v-r}(\phi) - M_{fr,r-s}(\phi) - M_{\text{load}} \tag{20}
\end{equation}

which is the difference between the torque \( M_p(\phi) \) generated by the pressure differences \( \Delta p_i \) over the vanes and the friction between the vanes and rotor \( M_{fr,v-r}(\phi) \) and between the rotor and stator \( M_{fr,r-s}(\phi) \), respectively as well as the constant load \( M_{\text{load}} \). \( M_p(\phi) \) is calculated by equation (21)

\begin{equation}
M_p = \sum_{i=1}^{N} M_i = \sum_{i=1}^{N} (\Delta p_i \cdot A_i \cdot d_i) \tag{21}
\end{equation}

with the lever arm \( d_i \) and the pressurized area \( A_i \) for each vane calculated according to equations (22) and (23).
\[
d_i(\varphi) = \frac{R_f(\varphi) + R_R}{2} = \frac{1}{2} \left( -e \cdot \cos(\varphi) + R_R + \sqrt{R_S^2 - (d \cdot \sin(\varphi))^2} \right) \\
A_i(\varphi) = L \cdot \left( -e \cdot \cos(\varphi) - R_R + \sqrt{R_S^2 - (d_i \cdot \sin(\varphi))^2} \right) 
\]

\(22\)  
\(23\)

**Friction modelling**

As the output torque of the motor is highly dependent on the losses within the motor, detailed modeling of the friction is necessary to achieve reliable results.

In the study, two main sources for friction losses are considered. First, velocity proportional friction is calculated. This includes the losses caused by steady, lubricated contacts which appear mainly in the bearings.

\[
M_{fr,r-s}(\varphi) = \mu_{fr,r-s} \cdot \dot{\varphi} 
\]

(24)

The second source for major friction losses is the contact between the vanes and the stator. The contact force and, therewith, the friction force at each vane has to be calculated for each vane separately. It can be seen in FIGURE 6 that tilting of the vanes occurs in the slot.

![FIGURE 6 Vane tilting and resulting friction forces at the vane](image)

The forces resulting from the tilting (the friction forces between the vane and the rotor \(F_{fr1}\) and \(F_{fr2}\) (cf. FIGURE 6 (b) which are depicted as well, can be calculated for each vane in dependence of the rotational position. Solving the mechanical system (including the radial force \(F_R\)) leads to equation (25) for the angle-dependent friction torque, including the radial contact forces \(F_{N,i}\) for each vane. The assumption of a permanent contact between the tip of the vane and the stator is necessary.

\[
M_{fr,v-r}(\varphi) = \sum_{i=1}^{N} M_{(fr,v-r),i} = \sum_{i=1}^{N} \mu_{fr} \cdot F_{N,i} \cdot R_v(\varphi) \\
= \sum_{i=1}^{N} \mu_{fr} \cdot F_{N,i} \cdot (-e \cdot \cos(\varphi) + \sqrt{R_S^2 - (e \cdot \sin(\varphi))^2}) 
\]

(25)

**RESULTS & EXPERIMENTAL VALIDATION**

To validate the results of the simulation study, a test rig was constructed in the IFAS lab. The test-rig consists of a reversible pneumatic vane motor with eight vanes without use of the expansion energy, i.e., the motor only has two ports for the air inlet and outlet respectively. Both ports cover a range of 90°. The inlet port starts at an angle of \(\varphi_{in_1} = 30°\), the outlet port begins at \(\varphi_{out_1} = 240°\). A hydraulic gear pump with a displacement of \(V_{pump} = 6 \text{ cm}^3\) is used as a manually adjustable load. The load torque applied by the pump is adjusted by a
pressure relief valve with a maximum pressure of \( p_{\text{hy, max}} = 80 \) bar. A schematic of this test rig is shown in FIGURE 7. The hydraulic pressure generated by the pump and the pneumatic supply pressure at the motor inlet are measured as well as the torque and the angular velocity. The motor outlet is connected to the environment. Additionally, the air flow to the motor is measured by a calorimetric mass flow sensor.

To validate the simulations different loads are applied to the motor working at a driving air pressure of 5 bar relative to the environment. FIGURE 8 ff. show exemplary results of the simulations in comparison to the experiment.

It is obvious that the values for the rotational speed are in acceptable accordance (deviation around 10 \%) for higher time values whereas the rise of the rotational speed in the start-up phase does not fit the experimental values. The same applies for the slope of the mean torque. The high deviations during start-up are caused by the modeling of the load torque. In the model, the load is a constant value acting right from the start of the rotational movement. In reality, the hydraulic pump first needs to build up pressure. Due to this lack of load at the beginning, the motor is able to accelerate very fast. In the simulation, a higher load slows down the acceleration at the beginning. When the pressure build-up in the hydraulic system is completed, the rotational speed stays constant. This constant value lies in a range of 8 \% and 10 \% to the simulated constant values at low and medium load, respectively.
FIGURE 9 Measured (purple dashed) and simulated (blue) rotational speed for medium load

FIGURE 10 shows a comparison of the measured and simulated torque for medium load. It is obvious, that the simulation shows higher fluctuation of the torque than the experiment. Nevertheless, the mean values are in good accordance. One of the main reasons for the oscillations in the simulation is the modeling of the outlet of the motor. When the front vane reaches the outlet, the pressure falls from 6 to 1 bar in just one timestep. This leads to high numerical oscillations seen in the simulated torque. Future work on the simulation model will focus on this problem.

FIGURE 10 Measured (purple dashed) and simulated (blue) torque for medium load

The simulation includes only one degree of freedom for the vanes. Therefore, a loss of contact between the vanes and the stator cannot be modeled. Especially at higher loads, a lift-off of the vanes may occur. This leads to higher leakage and subsequently to a lower driving torque in the experimental setup. A lower driving torque then leads to a lower maximum speed, which shows in the results above. Future work on the simulation model will focus on this problem by adding a second degree of freedom for the vanes.

SUMMARY & OUTLOOK

The paper presents a detailed model of the dynamic behavior of pneumatic vane motors. Especially the modelling of the pressure inside each chamber including a detailed description of the leakage losses between the chambers and through the gap between rotor and stator is shown. A detailed model for the friction between the vanes and the stator in dependence from the rotational position is shown as well. In combination with the geometrical model of the inertia and the chamber volumes between the vanes, it is possible to obtain a good accordance for the steady state behavior of the motor whereas the behavior during the start-up phase lacks accuracy.

In future projects, the dynamic behavior of pneumatic vane motors should be examined more closely. Therefore, a validation of the pressure inside each chamber should be developed. Also, a more detailed mathematical description of the friction at the tips of the vanes should be added to the model. Therefore, a second degree of freedom should be added to the simulation of the vanes’ positions. By doing so, it is possible to describe the loss of contact between the vanes and the stator. When the vane lifts off of the stator surface, the friction torque is reduced massively while the leakage losses increase largely. This has a large influence on the dynamic behavior at high load as well as low rotational speeds.

Detailed knowledge of the dynamic behavior can be used to implement model based control algorithms or for the simulative design of closed loop control for pneumatic motors which are mostly used in simple, open loop controlled applications today.
NOMENCLATURE

\( a \)  
Width of the vane

\( A_{\text{wall}} \)  
Wall area between two vanes

\( b \)  
Length of the vane

\( e \)  
Eccentricity of the rotor

\( J \)  
Moment of inertia

\( k \)  
Polytropic coefficient of air

\( L \)  
Length of the motor

\( m_r, m_v \)  
Mass (of the rotor/vane)

\( m \)  
Mass flow

\( N \)  
Number of vanes

\( N_u \)  
Nusselt number

\( p_i \)  
Pressure within chamber \( i \)

\( Pr \)  
Prandtl number

\( R \)  
Specific gas constant of air

\( R, R_B, R_S \)  
Radius (of the rotor/stator)

\( Re \)  
Reynolds number

\( T_i \)  
Air temperature within chamber \( i \)

\( V_i \)  
Geometric volume of chamber \( i \)

\( \alpha \)  
Heat transfer coefficient

\( \lambda_{\text{air}} \)  
Heat conductance of air

REFERENCES

NUMERICAL SIMULATION OF AIR JET IMPINGEMENT FOR ARCH BREAKING IN HOPPER

Y.G. Fang*, Y.J. Liu*, C.Y. Zuo*

* Department of Mechanical and Automotive Engineering, South China University of Technology
Guangzhou, 510641, China
(E-mail: yajun@scut.edu.cn)

Abstract. Air jets are employed for breaking the arch formed by clogging powder material in the hopper. In this work, a two-fluid model combining with kinetic theory of granular flow is used to simulate the penetration of air jets. The simulation of jet impingement on powder using only one air jet is performed and compared with the experimental result, which indicates that only one air jet is unable to break down all the powder agglomerations in the hopper. Then two air jets which are located at the same height and a three-dimension hopper geometry are utilized in the CFD simulations. The influences of jet parameters such as jet velocity and jet injection angle are discussed by analyzing the simulated results. The gas penetration length increases with the increase of air jet velocity or the decrease of jet injection angle.

Keywords: Air Jet, Arch Breaking, Numerical Simulation, Two-Fluid Model, Jet Penetration

INTRODUCTION

Arch is usually formed above the outlet of hopper during the discharge of granular materials. It has been often stated that jamming in granular materials results from a complex mechanical process due to interlocking of grains which ensures force chains propagation on large distances [1-3]. In other words, the stability of clogging arch depends on the equilibrium state of forces developed among the particles. When an external force acts on the arch which is formed by clogging powder material, the stable structure of arch will be broken and then the clogging arch will collapse [4-6].

One of effective methods to breaking arch is that one or more nozzles are arranged on the inclined wall of hopper, venting high-velocity air jet uniformly or abruptly to the zone where the jamming has the ability to occur. The air jet transfers momentum to the granular materials, and generates a force on the arch. Therefore, the arch will be destroyed and the jamming phenomenon will be eliminated.

Apart from this, air jet is also applied successfully in food, agricultural sciences and chemical industry. In many chemical processes, gas jets are used to enhance mixing and stimulate solid flow in fluidized bed reactors. Merry [7] carefully investigated the influence of jet velocity, etc. on the jet penetration length using three fluidized beds of different diameters and different kinds of particles. He obtained an equation for calculating the horizontal jet penetration length. Hong [8-9] analyzed the inclined jet characteristics (jet velocity, nozzle diameter, nozzle inclination angle, and nozzle position) and obtained a correlation for the inclined jet penetration length from experimental data. The inclined jet penetration length has also been determined by computer simulation based on a two-phase model. Li et al. [10] simulated the horizontal gas injection into a cylindrical fluidized bed using a three-dimensional multiphase CFD code based on an Eulerian-Eulerian Granular Kinetic Model. The transient simulation results show that the jet penetration lengths of different jet velocities are in good agreement with published experimental data and predictions of empirical correlations.

Now there are growing interests to use CFD to understand dense gas–solid two-phase flows and analyze the interaction between gas and solid [11]. The Eulerian–Eulerian (two-fluid) model with kinetic theory of granular flow (KTGF) is the most applicable approach to compute gas–solid flow. In the two-fluid model, the particles are treated as a continuum as in the gas phase. Thus, there are two interpenetrating phases (gas and solid) where each phase is characterized by its own conservation equation of motion. The interactions between the two phases are expressed as additional source terms added to the conservation equations. The kinetic theory of granular flow is used to define the fluid properties of the solid phase through constitutive equations.

In our work, air jet with high velocity is injected into hopper and impinges on the powder in order to break the clogging arch above the outlet of hopper. Because the detailed information about gas-solid flow such as the interaction mechanism of air jet and granular materials in the hopper, is very difficult to obtain due to the complicated behavior including impinging and penetration. Thus we apply two-fluid model to simulate the penetration of air jet impinging on the arch. The jet impingement region and distributions of solid volume fraction are investigated for different jet velocities and jet injection angles.
MODEL DESCRIPTION

Basic Assumptions

The two-fluid model solves \( n \) momentum and continuity equations for gas and particle phases which are both treated as continuous phases. In the present simulation, the solids are assumed to be mono dispersed spheres, so the max volume fraction of clogging powder material in the hopper is about 0.63 [12]. For simplify, it is further assumed that the gas phase is incompressible, gas-solid flow is isothermal without reaction and there is no mass transfer between phases. The gas–solid momentum exchange coefficient is a function of drag, virtual mass and lift force. However, since the particle size of material has a relatively small diameter of 0.05 mm, the lift force is neglected. The virtual mass force is insignificant when the solid density is much larger than gas density [13]. Thus, the momentum exchange coefficient comprises only the drag force. The Eulerian approach is applied to simulate the complex gas-solid flows in the clogging hopper, taking into account all possible intra- and inter-phase interaction.

Conversation Laws

The continuity equation for the gas phase:

\[
\frac{\partial}{\partial t}(\varepsilon_g \rho_g) + \nabla \cdot (\varepsilon_g \rho_g \vec{u}_g) = 0
\]  

The continuity equation for the solid phase:

\[
\frac{\partial}{\partial t}(\varepsilon_s \rho_s) + \nabla \cdot (\varepsilon_s \rho_s \vec{u}_s) = 0
\]  

\( \rho_s \) and \( \rho_g \) denote the densities of the particle and gas phase respectively. \( \vec{u}_g \) and \( \vec{u}_s \) are the velocity of the particle and gas phase respectively. \( \varepsilon_g \) and \( \varepsilon_s \) are the volume fractions of the particle and gas phase respectively.

The momentum equation for the gas phase:

\[
\frac{\partial}{\partial t}(\varepsilon_g \rho_g \vec{u}_g) + \nabla \cdot (\varepsilon_g \rho_g \vec{u}_g \vec{u}_g) = -\varepsilon_g \nabla P_g + \nabla \cdot \tau_g + \varepsilon_g \rho_g \vec{g} + \beta (\vec{u}_g - \vec{u}_s)
\]  

The momentum equation for the solid phase:

\[
\frac{\partial}{\partial t}(\varepsilon_s \rho_s \vec{u}_s) + \nabla \cdot (\varepsilon_s \rho_s \vec{u}_s \vec{u}_s) = -\varepsilon_s \nabla P_s - \nabla P_g + \nabla \cdot \tau_s + \varepsilon_s \rho_s \vec{g} + \beta (\vec{u}_s - \vec{u}_s)
\]

\( \vec{g} \) is the gravitational acceleration. \( P_s \) is particle pressure which represents the particle normal forces due to particle-particle interactions, given as Eq. (T1-3) in Table 1. \( \tau_s \) and \( \tau_g \) are the stress tensors of the particle and gas phase respectively, which can be calculated by Eq. (T1-1) and Eq. (T1-2). The solids stress tensor \( \tau_s \) can be expressed in terms of the shear solids viscosity \( \mu_s \) and bulk solids viscosity \( \xi_s \), and given as Eq. (T1-4) and Eq. (T1-5) [14-15]. \( g_0 \) is the radial distribution function which is interpreted as the probability of particle collisions, defined as Eq. (T1-10). \( e \) is the restitution coefficient, \( d \) is the particle diameter, \( \theta \) is the granular temperature and \( \psi \) is the angle of internal friction. \( \beta \) is the drag coefficient, namely the momentum exchange coefficient between the solid and gas phases [16]. Gidaspow [17] proposed a drag coefficient model that is a combination of the Ergun [18] and the Wen and Yu [19] experimental correlations, which is given as Eq. (T1-11-14). The equation of conservation of solid fluctuating energy:

\[
\frac{3}{2} \frac{\partial}{\partial t}(\varepsilon_s \rho_s \theta) + \nabla \cdot (\varepsilon_s \rho_s \theta) \vec{u}_s = \nabla \cdot \{ \nabla P_s + \nabla \cdot \tau_s \} \vec{u}_s + \nabla \cdot (k_s \nabla \theta) - \gamma_s + \sigma_s + D_s
\]  

\( k_s \) is thermal conductivity of particles, given as Eq. (T1-6). From kinetic theory of granular flow, particles are assumed to be smooth spheres and slightly inelastic, the dissipation fluctuating energy, \( \gamma_s \), only comes from...
binary inelastic collisions [17], which is expressed by Eq. (T1-7). The rate of energy dissipation per unit volume, $D_{\text{en}}$, is energy consumption for exchange between particles and gas fluctuation per unit volume, as given in Eq. (T-8). And the net fluctuation energy exchange between the gas and solids, $\sigma_c$, is expressed by Eq. (T1-9) [20].

### TABLE 1. Constitutive Equations

<table>
<thead>
<tr>
<th>Equation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_{\epsilon} = e_{\epsilon} T_{\epsilon}$</td>
<td>Gas Phase Stress</td>
</tr>
<tr>
<td>$T_{\epsilon} = e_{\epsilon} T_{\epsilon}$</td>
<td>Solid Phase Stress</td>
</tr>
<tr>
<td>$T_{\epsilon} = e_{\epsilon} T_{\epsilon}$</td>
<td>Solid Pressure</td>
</tr>
<tr>
<td>$\mu_s = \frac{4}{5} \rho_s (1 + \epsilon) \frac{\beta}{\pi} \left( \frac{1}{2} \right)^{1/3}$</td>
<td>Shear Viscosity of Solids</td>
</tr>
<tr>
<td>$\mu_s = \frac{4}{5} \rho_s (1 + \epsilon) \frac{\beta}{\pi} \left( \frac{1}{2} \right)^{1/3}$</td>
<td>Bulk Viscosity of Solids</td>
</tr>
<tr>
<td>$k_s = \frac{75}{64} \rho_s \left( \frac{\beta}{\pi} \right)^{2/3}$</td>
<td>Thermal Conductivity of Solids</td>
</tr>
<tr>
<td>$\gamma_c = 3(1 - \epsilon') \rho_s \left( \frac{\beta}{\pi} \right)^{1/3}$</td>
<td>Dissipation Fluctuating Energy</td>
</tr>
<tr>
<td>$D_{\text{en}} = \frac{18 \mu_s}{4 \sqrt{\rho_s}} \left( \frac{\beta}{\pi} \right)^{1/3} \left( \frac{\beta}{\pi} \right)^{1/3}$</td>
<td>Rate of Energy Dissipation per unit Volume</td>
</tr>
<tr>
<td>$\sigma_c = -3 \beta \theta$</td>
<td>Exchange of Fluctuating Energy between gas and particles</td>
</tr>
<tr>
<td>$\beta = (1 - \phi) \beta_{\text{in}} + \phi \beta_{\text{rr}}$</td>
<td>Radial Distribution Function</td>
</tr>
<tr>
<td>$\phi = \arctan \left[ 150 \times 1.75 (0.2 - r') \right] + 0.5$</td>
<td>Gidaspow Drag Coefficient Model</td>
</tr>
<tr>
<td>$\beta_{\text{in}} = 150 (1 - e') \left( 1 - e' \right) \left( \frac{s'}{d'} \right)$</td>
<td></td>
</tr>
<tr>
<td>$\beta_{\text{rr}} = \frac{3}{4} \beta_{\text{in}} \left( \frac{1 - e'}{d'} \right) \left( \frac{s'}{d'} \right)$</td>
<td></td>
</tr>
</tbody>
</table>

### Geometry and Simulation Setup

The discharging hopper, as shown in Fig. 1, is a cylindrical column with an inside diameter of 492 mm and a height of 240 mm, and with a tapering angle of 50° in the conical base with a height of 240 mm. The diameter of the conical base at the bottom is 90 mm. The static bed of clogging powder material is 250 mm in height. The convergent nozzle with an outlet diameter of 3 mm, which was mounted on the conical wall 60 mm above the bottom of hopper. The nozzle inclination angles $\theta_i$ can be set as 40°, 20° or 0° for CFD analysis. Compressed air flows through the convergent nozzle and forms a high velocity air jet, whose velocity can be adjusted to 0~250 m/s by regulating the exhaust pressure of air compressor in experiment.

The whole computational domain was divided into 523,490 elements and the boundary conditions utilized in the simulation are also schematically shown in Fig.1. The standard $k-\epsilon$ mixture turbulence model is chosen for modeling turbulence in multiphase flows. The material properties and operating conditions are listed in Table 2. The simulation of air jet impingement on clogging arch of powder material in hopper has been carried out with FLUENT 14.0 software, using a small time step of 0.001 s with approximately 30 iterations [21].

### TABLE 2. Material Properties and Operating Conditions

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Particle average diameter, $d$ (mm)</td>
<td>0.05</td>
<td>Static compacted bed height, $H_b$ (mm)</td>
<td>250</td>
</tr>
<tr>
<td>Particle density, $\rho$ (kg/m$^3$)</td>
<td>520</td>
<td>Nozzle diameter, $d_n$ (mm)</td>
<td>3</td>
</tr>
<tr>
<td>Particle restitution coefficient, $e$</td>
<td>0.9</td>
<td>Jet velocity, $u_j$ (m/s)</td>
<td>125,150,175,200</td>
</tr>
<tr>
<td>Gas density, $\rho_g$ (kg/m$^3$)</td>
<td>1.2</td>
<td>Jet injection angle, $\theta_i$</td>
<td>40°, 20°, 0°</td>
</tr>
<tr>
<td>Gas viscosity, $\mu_g$ (Pa·s)</td>
<td>1.789×10$^{-5}$</td>
<td>Time step (s), Number of steps</td>
<td>0.001,1000</td>
</tr>
</tbody>
</table>
RESULTS AND DISCUSSION

Comparison of Simulation with Experimental Result

An experiment of air jet impingement on clogging powder material is conducted in a hopper, as shown in Fig. 2. The experimental material is flour with a particle diameter of 0.05 mm and density of 520 kg/m$^3$. The nozzle inclination angle, $\theta_i$, is 40°. The powder material is compacted artificially after shutting down the outlet at the bottom of hopper and the static bed height is about 250 mm. The working pressure of air compressor is adjusted to be 0.3 MPa and the opening time of solenoid valve is adjusted to be 1 s by using time relay, which means the injection time of air jet is 1 s. In this experiment, only one air jet is injected into hopper and its flow rate, $Q$, obtained by rotameter is 40.76 L/min. The velocity of air jet calculated by Eq. (6) is 96.15 m/s.

$$u_j = Q / (\pi d_J^2 / 4)$$  \hspace{1cm} (6)

Fig. 3 shows the packing state of powder material before and after air jet impingement. Jamming occurs above the outlet of hopper since powder material is so firmly cohered by artificial compacting and vibration, forming stable force chain inside the powder. Due to the great impact force caused by air jet exhausted from convergent nozzle, the force chain inside the powder is broken down and the agglomerations of powder shatter. An impingement zone has been formed along the penetration direction of air jet as shown in Fig. 3. It’s noteworthy that the powder material in the periphery of the impingement region still remains stationary and bonding state, not affected by the high-velocity air jet.

FIGURE 2. Schematic Diagram of Experimental Setup:
Fig. 4 shows instantaneous volume fraction and velocity vectors of particles at the $x$-$z$ plane. In this case, the operating conditions of simulation are the same as experimental measurement. The velocity of air jet is 96.15 m/s and the jet injection angle is $40^\circ$. The air jet injection time of 1 s is so short that the outlet at the below of hopper is assumed to be closed. As shown, the air jet impinges on the powder material in the hopper and the momentum is transferred to the solid particles from the gas phase. Due to the large drag force caused by violent impinging of air jet, the particles are uplifted, leading to the formation of impingement zone above the nozzle position. With the increase of air jet injection time, the impingement region expands along the jet penetration direction and the jet is a torch-like coherent void that bends upward. At the end of air jet penetration, there are two zones forming in the hopper, one is the core-annulus zone and the other is peripheral zone. The local solid volume fraction is relatively low in the core-annulus zone.

From the simulation we see that the velocities of particles in the periphery of the impingement zone, where the powder material remains static as shown in Fig. 3, are vanishingly low. In contrast, the particles in the core of impingement zone have a high velocity and move upward driven by air jet. Hence, the powder agglomerations in the impingement region are broken down. If the outlet at the below of hopper is opened, the powder material above and below the nozzle position would flow down owing to the gravity and the jamming above the outlet will be eliminated. However, the impingement region of one air jet is not large enough to smash all the powder agglomerations in the hopper. The simulation is in good agreement with the experiment result shown in Fig. 3. Obviously, the velocity of air jet, jet injection angle and nozzle numbers play significant role in breaking the arch formed by the clogging powder material in the hopper. These design factors influence the effect of arch breaking and energy consumption.
Effect of Air Jet Velocity

Since the hopper and powder material is not transparent, it is impossible to observe the process of air jet impinging on the powder. Hence, the CFD simulations are implemented in the present work, which can give detailed information of gas-particle flows. From the experimental result shown in Fig. 3, it is realized that only one air jet is unable to break down all the agglomerations of powder material in the hopper. Thus two air jets are imposed in the simulations using the 3D meshed geometry shown in Fig.1 and operation parameters listed in Table 2. The two air jets are located at the same height of 60 mm and opposite to each other.

Fig.5 shows the instantaneous volume fraction of gas phase at four air jet velocities with the same jet injection angle of 40°. It is found that the arch breaking process can be divided into two stages. One is the two air jets penetrate the powder material without disturbance before they meet with each other. Meanwhile, the particles move along the jet penetration direction under the drag force exerted by the air jet of high velocity. The other is the two air jets slam into each other with the formation of strong turbulence and then flow upward. The violent collision provides sufficient large impinging force for breaking down the powder agglomerations.

![Figure 5](image-url)

**FIGURE 5.** The Instantaneous Gas Volume Fraction at Four Air Jet Velocities ($\theta_i=40°$)

From the simulation we can see that the gas volume fraction in the region near the collision point of two air jets is relatively high in the hopper, forming an impingement zone. It is noteworthy that there are two gas penetration directions after the collision of two air jets in this case. One is the gas penetrates upward. The other penetration direction is perpendicular to the $x$-$z$ plane where the two air jets are located. In order to assess the performance of air jets impinging on the clogging powder, the gas penetration length, $L_p$, defined in Fig.5, is calculated as the distance of the lowest gas volume fraction along the second gas penetration direction at a certain bed height. Fig.6 shows that the penetration length increases as the air jets velocities increase, leading to a larger gas penetration region. The gas penetration region is constantly expanding until the injections of air jets stop. The gas penetration can result in the expansion of agglomeration, which makes powder becoming fluffy and easier to flow. Consequently, the time required for arch breaking is reduced with the increase of air jets velocities and penetration length. However, the increase in air jets velocities increases the gas consumption. This means more energy is required for arch breaking. Therefore, an optimum operating condition is always preferred for the better effect of arch breaking with less energy consumption.
Effect of Jet Injection Angle

Fig. 8 shows the instantaneous gas volume fraction at different jet injection angles with the same air jet velocity of 200 m/s. Obviously, the distance between the jets collision point and the bed bottom decreases as the jet injection angle decreases. Meanwhile, greater turbulence is formed after the jets collision at the bed bottom with the decrease in jet injection angle. From simulation we can see that at the jet injection angle of 40°, there is scarcely any gas penetrating downward at the time of 1 s. At the jet injection angle of 20°, gas penetrates toward the bed bottom a certain depth. At the jet injection angle of 0°, the powder material is completely penetrated by gas. From Fig. 7 we can see that as the jet injecting angle decreases, the maximum gas penetration length in the hopper increases. This indicates that the penetration region increases with the decrease in the jet injection angle. Consequently, after the powder agglomerations at bed bottom fully broken down and flowing through the outlet, the upper part of powder subsequently flows down under the action of gravity, resulting in a better effect of arch breaking.

FIGURE 6. The Profile of Gas Penetration Length at Different Bed Heights (θi=40°)

FIGURE 7. The Profile of Gas Penetration Length at Different Jet Injection Angles

FIGURE 8. The Instantaneous Gas Volume Fraction at Three Jet Injection Angles (uj=200 m/s)
CONCLUSION

A two-fluid model with the kinetic theory of granular flow (KTGF) has been implemented to simulate the air jet penetration for arch breaking in the hopper. A simple arch breaking experiment using one air jet is conducted and compared with the simulation result. It is realized that only one air jet can form a relatively large impingement zone along the jet penetration direction but not break down all the powder agglomerations in the hopper. Then two air jets are imposed in the CFD simulation and the jets penetration process is analyzed. The simulation shows that there are three gas penetration directions after the jets collision: up, down and perpendicular to the plane where the two air jets are located. The velocities and injection angles both have significant influence on the arch breaking in the hopper. The gas penetration length, Lp, increases as the jets velocities increase or jet injection angle decreases, leading to a larger gas penetration region and a better effect of arch breaking.

ACKNOWLEDGMENTS

This work is supported by the technical development project entrusted by enterprise (D8162990).

REFERENCES

Oral Presentation | Oil hydraulics

[2C01-06] H6 (HST, Mobile Applications)
Chair:Xiangdong Kong(Yanshan University), Hideki Yanada(Toyohashi University of Technology)
Thu. Oct 26, 2017 9:00 AM - 10:36 AM Room C (ACROS Fukuoka)

[2C01] POSITION CONTROL OF VALVELESS HYDRAULIC CLUTCH ACTUATOR
  *Chao Zhang¹, Bingzhao Gao¹, Xingjun Hu¹, Yulong Lei¹, Hong Chen¹ (1. Jilin University, China)
  9:00 AM - 9:16 AM

[2C03] DESIGN OF A POWER REGENERATIVE HYDROSTATIC WIND TURBINE TEST PLATFORM
  Biswaranjan Mohanty¹, Feng Wang², *Kim A Stelson¹ (1. University of Minnesota, 2. Zhejiang University)
  9:32 AM - 9:48 AM

[2C04] WAVE POWER CONVERTER PENDULOR WITH HYBRID H.S.T.
  *TOMIJI WATABE¹, Prasanna GUNAWARDANE², Hiroki MATSUMOTO³ (1. Director of T-Wave Consultant JAPAN (Inventor of Wave power converter Pendulor), 2. Senior Lecturer of Mechanical Eng. of Univ. of PERADENIYA SRILANKA (Reasercher of the Pendulor), 3. Lecturer of Mechanical Eng. of MURORAN I. T. JAPAN (Researcher on the wave propagation))
  9:48 AM - 10:04 AM

[2C05] DISC BRAKE WITH HYDROMECHANICALLY CONTROLLED BRAKE TORQUE FOR RAILWAY APPLICATIONS
  *Matthias Petry¹, Ahmed Zaki¹, Hubertus Murrenhoff¹ (1. Institute for Fluid Power Drives and Controls (IFAS), RWTH Aachen University)
  10:04 AM - 10:20 AM

[2C06] RESEARCH ON THE EFFECTS OF DOUBLE ARC OIL GROOVE PARAMETERS ON TORQUE CHARACTERISTICS IN HYDRO-VISCOSOUS DRIVE
  YUANYUAN DENG¹, Zisheng LIAN², *Hongwei CUI² (1. College of Mechanical Engineering, Taiyuan University of Technology, 2. Shanxi Key Laboratory of Fully Mechanized Coal Mining Equipment, Taiyuan University of Technology)
  10:20 AM - 10:36 AM
POSITION CONTROL OF VALVELESS HYDRAULIC CLUTCH ACTUATOR

Chao Zhang*, Bingzhao Gao*, Xingjun Hu*, Yulong Lei*, Hong Chen*

*The State Key Laboratory of Automotive Simulation and Control, Jilin University
Changchun, 130025, PR China
(E-mail: zchao@mails.jlu.edu.cn)

Abstract. The automatic clutch system is an important part of automated manual transmission (AMT). It plays a vital role in the fuel economy, comfort and drivability of the vehicle. However, automatic clutch displacement control has always been a challenge. This paper presents a new type of hydraulic actuator without valve. It uses motor speed control to achieve system pressure and flow control with the advantage of low cost and high efficiency. Next, the nonlinear feedforward feedback control scheme with gain-scheduling proportional-integral-derivative (PID) is adopted for the proposed model, due to the system nonlinearity to achieve the precise position control of the clutch. Finally, the designed controller is verified by simulation and experiment. The results show that the proposed actuator and controller is a good attempt to solve the clutch position control.

Keywords: Position control, clutch actuator, valveless hydraulic system.

1. INTRODUCTION

Automatic transmissions have a lot of applications in automobiles. Its performance directly affects the vehicle's drivability, comfort, safety and fuel economy. Automatic clutch system plays a crucial role in automated manual transmission (AMT) [1], which is widely used in pure electric vehicles and hybrid electric vehicles to implement the mode switching and gear shifting [2,3]. The performance of actuator has a significant influence on clutch system, and consequently impacts shifting quality of transmission. An automatic clutch is usually actuated by electro-hydraulic, electro-pneumatic, or electro-mechanical system. Electro-hydraulic has so many advantages that it is widely used in vehicles, including high power/mass ratio, fast response, easy installation and high stiffness.

The design procedure of automatic clutch system mainly focuses on creative innovation of mechanical structure and control strategy. However, there are many nonlinearities of the system, such as nonlinearity of the throw-out force and uncertainty in wear. To solve the nonlinear control problem of clutch actuator, many control strategies have been investigated, including predictive control [4,5], sliding-mode control [6], neuron proportional-integral-derivative (PID) [7], output-tracking-error-constrained robust position control [8] and triple-step nonlinear method [9]. References [9] points out a triple-step nonlinear method, which consists of three steps. The control algorithm has a good effect in trajectory tracking control.

The electro-hydraulic is widely used on automobiles, which usually contains pump, tank, and high accuracy valves [5]. Electro-hydraulic clutch actuator usually achieves the change of clutch displacement by the way of controlling the solenoid valves.

This paper is related to the position control of a valveless hydraulic clutch actuator which adopts a DC motor, external gear pump and hydraulic cylinder. This hydraulic system does not use solenoid valves and accumulators, thus reduce costs and improve system efficiency. The clutch displacement adjustment is achieved by driving the external gear pump by the motor. Although the system structure is simplified, the nonlinear relationship between pressure, displacement and speed of pump brings precise control to the difficulty. In this paper, a gain-scheduling proportional-integral-derivative (PID) controller will be adopted. The rest of this paper is organized as follows. The second part is the structure and model of main components. Others can be regarded as ideal rigid body. The third part is controller design based on the model, and the simulation and test results are given in the fourth section. Finally, the fifth part concludes the paper.

2. MODELS OF THE HYDRAULIC CLUTCH ACTUATOR

2.1 Structure and Working Process of Clutch System

As shown in Fig. 1, the valveless hydraulic clutch actuator adopts a DC motor as the power source, an external gear pump as pressure source and hydraulic cylinder; there is no other solenoid valve. When the motor receives control signal and rotates in one direction or in the opposite direction, the external gear pump moves hydraulic
fluid between hydraulic cylinder and tank. Piston of hydraulic cylinder and diaphragm moves, and then, the clutch is disengaged or engaged. When clutch reaches the desire position, DC motor stops to rotate or rotates slowly and holds on the pressure. The flow rate through the pump and pressure differential can be computed from the motor velocity at any time. Precise position control of the clutch can be achieved. Because the system is without valves, very high efficiency and low cost can be achieved.

![Schematic graph of clutch actuator.](image)

**FIGURE 1.** Schematic graph of clutch actuator.

### 2.2 Mathematical model of system

In this section, the sample mathematical model based on physical principles for hydraulic clutch actuator is presented. Other parts are not mentioned in consideration of simplicity.

**A. DC Motor Modeling**

The brush DC motor is considered here, which is driven by an H-bridge circuits made up of four MOSFETs. Pulse width modulation (PWM) control is used to modulate the motor voltage. The PWM frequency is set as 10 kHz, so that the current could be controlled without noticeable. According to physical principles, the equation of the armature circuit and the DC motor is:

\[
L_a I_a + I_a R = v_{bat} - k_v \dot{\theta}_m
\]

\[
k_t I_a = T + T_{mf} + J_m \ddot{\theta}_m
\]

where \(v_{bat}\) is the battery voltage, \(R\) is the resistance of the armature circuit, \(L_a\) is the armature inductance, \(I_a\) is the armature current, \(u\) is the PWM duty ratio, \(k_v\) is the back electromotive force coefficient, \(\theta_m\) is the motor rotational angle, \(T\) and \(T_{mf}\) are the output torque and friction torque of the motor, respectively, \(k_t\) is the torque coefficient, and \(J_m\) is the inertia.

**B. External Gear Pump Modeling**

The motor shaft is connected to the shaft of pump. The type of pump is external gear pump. It can move fluids by mechanical action. When the motor rotates, the external gear pump moves hydraulic fluid between tank and hydraulic cylinder. The input port is connected to tank, and the output port is connected to hydraulic cylinder. When the pump rotated backwards, the input and output exchange. The dynamic equation is given as

\[
q_v = V \omega \eta_{vol}
\]

\[
P_{hydr} = q_v \Delta P
\]

\[
P_{meca} = T \omega \eta_m
\]

With the equality of the hydraulic power and mechanical power, the torque is:

\[
T = \frac{\Delta P V \eta_{vol}}{\eta_m}
\]
where \( q_v \) is the volumetric flow rate, \( V \) is the instantaneous pump displacement, \( \omega \) is the shaft speed, \( \eta_{vol} \) and \( \eta_m \) are volumetric efficiency and hydraulic-mechanical efficiency, \( P_{hydr} \) and \( P_{meca} \) are the hydraulic power and mechanical power, \( T \) is the pump demand torque which equals the motor output torque.

### C. Hydraulic Cylinder Modeling

When the external gear pump forces the hydraulic fluid into the hydraulic cylinder, the pressure of hydraulic fluid acts on piston of hydraulic cylinder, and then, the piston pushes diaphragm spring of clutch. The hydraulic cylinder model can be described as:

\[
V \theta_m \eta_{vol} = xA \tag{7}
\]

\[
P A - F_f = m \ddot{x} \tag{8}
\]

\[
F_f(x, \dot{x}) = \begin{cases} f + F_l & \text{if } \dot{x} = 0 \text{ and } |f| < f_s \\ f_s \text{sgn} (\dot{x}) + k_2 \dot{x} + F_l & \text{otherwise} \end{cases} \tag{9}
\]

Where \( A \) is the piston area, \( x \) is the clutch displacement, \( m \) is the piston mass, \( f \) is the sum of resistance force and return force, \( F_l \) is the return force of the diaphragm spring.

<table>
<thead>
<tr>
<th>Table 1. Parameters of the actuator system</th>
</tr>
</thead>
<tbody>
<tr>
<td>Symbol</td>
</tr>
<tr>
<td>( L_a )</td>
</tr>
<tr>
<td>( v_{bat} )</td>
</tr>
<tr>
<td>( k_v )</td>
</tr>
<tr>
<td>( k_t )</td>
</tr>
<tr>
<td>( T_{mf} )</td>
</tr>
<tr>
<td>( J_m )</td>
</tr>
<tr>
<td>( V )</td>
</tr>
<tr>
<td>( R )</td>
</tr>
<tr>
<td>( A )</td>
</tr>
</tbody>
</table>

### 3. CONTROLLER DESIGN

Define the clutch displacement \( x \) as \( x_1 \), and clutch speed \( \dot{x} \) as \( x_2 \). The state-space form of the valveless hydraulic clutch actuator can be written as:

\[
\begin{align*}
\dot{x}_1 &= x_2 \\
\dot{x}_2 &= a_2 F_f(x_1, x_2) + b_2 x_2 + c_2 u
\end{align*} \tag{10}
\]

where

\[
a_2 = -\frac{T_{mf} \eta_{vol} \eta_m A V + V^2 \eta_{vol}^2}{V^2 \eta_{vol}^2 m + J_m A^2 \eta_m} \tag{11a}
\]

\[
b_2 = -\frac{K_t K_v \eta_m A^2}{(V^2 \eta_{vol}^2 m + J_m A^2 \eta_m) R} \tag{11b}
\]

\[
c_2 = \frac{K_t \eta_{vol} \eta_m AV_{bat}}{(V^2 \eta_{vol}^2 m + J_m A^2 \eta_m) R} \tag{11c}
\]

The property of diaphragm spring, friction among movement, and efficiency of pump are nonlinear functions from experiments tests and theoretical analysis. These parameters have a significant influence on the precise control of clutch. To overcome the influence of nonlinear functions, the controller contains feedforward and feedback. The feedback control law is the gain-scheduling proportional-integral-derivative (PID) position feedback to ensure the stability of the system. The nonlinear functions in the feedforward control law are expressed in the form of look-up tables, and the two parts constitute the final controller.

The control law is designed as a feedback-feedforward controller, which can be described as
\[ u = u_f + u_b \]  
(12)

\[ u_f = \frac{\left[ T_{mf} \eta_m A + V \eta_{volf}(x_1, x_2) \right] R}{K_i \eta_m AV \alpha} \]  
(13)

\[ u_b = K_p + K_i \int_0^t e(t) \, dt + K_d \frac{de(t)}{dt} \]  
(14)

where \( u_f \) is feedforward control law, \( u_b \) is gain-scheduling PID.

4. SIMULATION AND EXPERIMENT RESULTS

The clutch actuator is used for micro-electric vehicle, which contains a 2-speed transmission. A dry clutch is adopted to compensate the traction loss during gear shifting [10]. Clutch separate travel is 5mm, and the separation force of clutch is 500–600N.

4.1 Simulation results

Based on the equations above, the simulation model of valveless hydraulic clutch actuator is implemented in the software of AMESim. The results of step response are given to investigate the performance of system. The solid line representing \( y^* \) is the desire clutch position, and dash line representing \( y \) is the actual position. The tracking error and settle time reflects the system performance in fig 2. It is shown that the results meet the requirements of design.

4.2 Experiment results

The test system is made up of an actuator unit installed with a position sensor, a clutch, a motor driver, and a MicroAutoBox of dSPACE. The nominal power of motor is 100 W (at speed of 1500 rpm). The PWM frequency of the motor driver (H-bridge circuit) is set as 10 kHz, and the system sampling time is set as 1 ms. Fig 3 is the test bench, Fig 4 is the experimental results. The step responses are shown in fig 4. From fig 4, it can be seen that experimental test results are similar with simulation. The settle time of control is no more than 1s, and the steady-state error of control is less than 0.2mm, which satisfies the clutch operation requirements.
5. CONCLUSIONS

In this paper, a novel electro-hydraulic actuator for clutch displacement control is proposed. A feedback-feedforward controller is designed. The controller takes load force, friction force, and pump efficiency into account which makes it avoid the impact of nonlinear functions. The parameters of model are based on experimental tests and theoretical analysis. From simulations and experimental tests, the mechanical structure and controller can achieve precise clutch control well. The advantage of the scheme is cost reduction and high efficiency, which makes it easy to be adopted in micro-electric vehicle.

ACKNOWLEDGMENTS

The work was supported by the National Natural Science Foundation of China (61520106008, 61522307), the Key Research Project of Science and Technology Department of Jilin Province (20150204056GX).
REFERENCES

DESIGN OF A POWER REGENERATIVE HYDROSTATIC WIND TURBINE TEST PLATFORM

Biswaranjan Mohanty*, Feng Wang** and Kim A Stelson *

*Center for Compact and Efficient Fluid Power
Department of Mechanical Engineering,
University of Minnesota, Minneapolis, MN 55455, US
(E-mail: mohan035@umn.edu, kstelson@umn.edu)

**State Key Laboratory of Fluid Power and Mechatronic Systems
Department of Mechanical Engineering, Zhejiang University
38 Zheda Road, Hangzhou, Zhejiang 310027, China
(Email:wang2148@umn.edu)

Abstract. The demand for community wind turbines is increasing to fulfill local requirements and make the grid more stable. A turbine with a hydrostatic transmission is more reliable and cost effective than a conventional gearbox turbine making it an attractive alternative for community wind turbines. A power regenerative test platform has been built at the University of Minnesota to understand the performance of a hydrostatic transmission in a wind turbine. The design of the test bed is described in detail in this paper. The testbed emulates the rotor torque including the effects of the blade dynamics and pitch in a hardware-in-the-loop configuration. This test platform provides a powerful tool to investigate the performance of new components, controllers, fluids and energy storage methods on the hydrostatic wind turbine transmission.

Keywords: Wind Turbine, Renewable Energy, Hydrostatic Transmission

INTRODUCTION

Wind energy is the fastest growing green and clean energy source of electricity. The total installed capacity has reached 487 GW by the end of 2016, which is 4.7% of the world’s electricity demand [1]. Most wind energy comes from large wind farms. Large wind farms are far away from the point of use, increasing transmission cost. In contrast, small and midsize turbines can fulfill local demand and make the distributed grid more reliable and stable. These distributed wind turbines can be installed in homes, farms, businesses and public facilities such as schools or hospitals to reduce consumer electricity bills.

A conventional turbine uses a multi stage fixed ratio gear box to transmit power from the low speed generator. An expensive power converter is used to convert the generator frequency to the grid frequency. Studies conducted by the National Renewable Energy Laboratory (NREL) among others document failure frequency and downtime for wind turbines [2][3]. The studies show that electrical systems fail frequently with shorter downtimes, but gearboxes and generators fail less frequently with longer downtimes. The failure of gearboxes and generators is due to unsteady wind, causing impact loading which reduces the life of the components. Failures not only decrease the annual energy production of the turbine, but also increase the maintenance cost.

The reliability of a wind turbine can be improved by removing the gearbox. But the downside of the direct drive is it requires permanent magnets, which increases turbine cost and weight. The other way to increase the reliability is by replacing the gear box with a hydrostatic transmission (HST). HSTs are simple, light and cost effective and have high power density. The slight compressibility of the hydraulic fluids in an HST reduces the shock loading on mechanical components increasing their life. An HST is one type of continuous variable transmission, commonly used in off road and construction equipment. HST can be coupled with an accumulator to store energy [4][5].

Wind energy research and development efforts have primarily benefitted large wind turbines. NREL has dynamometer research facilities to perform steady-state research validation to determine turbine power curves. NREL employs "model-in-the-loop" techniques to emulate rotor, tower, pitch, and yaw systems with 225 kW, 2.5MW and 5 MW dynamometers to test gearbox turbines. The Institute for Fluid Power Drives and Controls in RWTH Aachen University developed a 1 MW hydrostatic wind test platform, designed to replace the commonly used gearbox and power converter [6][7]. To optimize the transmission efficiency, multiple pumps and motors
are used to enable a switching strategy at different wind power inputs. Although a hydrostatic transmission consisting of conventional pumps and motors has lower efficiency than a gearbox, the overall system efficiency is still competitive since the need for a power converter is eliminated.

For the community wind turbine a HST with single pump and motor will be the cost effective and reliable. It is important to understand the performance of the HST under various loading conditions. To measure the performance, a power regenerative hydrostatic wind turbine has been developed at the University of the Minnesota. The design of the testbed is described in this paper.

WIND TURBINE PERFORMANCE

In a wind turbine, airfoil shaped blades capture kinetic energy from the wind and transform it into rotational energy of the rotor. The rotor power is proportional to the power coefficient \( C_p \) and the cube of the wind speed \( \nu \) as shown in Eq. (1).

\[
P_{rot} = C_p(\lambda, \beta)P_{wind} = C_p(\lambda, \beta)0.5\rho\nu^3
\]

where \( \rho \) is the air density and \( A \) is the swept area of the rotor. According to Betz’s law, the maximum aerodynamic efficiency of a wind turbine rotor is 59% [8]. The power coefficient is a function of the tip speed ratio (TSR) and pitch angle \( \beta \), as shown in Eq. (1). The TSR \( \lambda \) is the ratio of the blade tip speed to the wind speed, where \( R \) is the rotor radius as shown in Eq. (2).

In a wind turbine the rotor interacts with time-varying wind. As a first step, the loading condition of a 90 kW, horizontal-axis and three-bladed variable speed turbine with variable pitch is analyzed. The power coefficient curve of the wind turbine at optimum pitch angle is shown in figure 1(a). The maximum \( C_p \) of the rotor is 0.46 at the optimal TSR of 7.

\[
\lambda = \frac{\omega R}{\nu}
\]

FIGURE 1. (a) Power coefficient curve of a three bladed variable speed turbine. (b) Power curve of a 90 kW wind turbine.

FIGURE 2. (a) Rotor Power curve (b) Rotor torque and power at rated speed (12 m/sec)
The output power of the turbine and available wind power is shown in figure 1(b). The turbine output power includes the inefficiency of the drive train. At wind speeds below 3m/sec (cut-in speed), the power available from the wind is less than the turbine losses. At wind speeds above 24m/sec (cut-out speed), the turbine is shut down to avoid damage. In the region from the cut-in speed to the rated speed (12m/sec) the turbine is operated at optimal TSR to maximize power. In the region above the rated speed, the power is limited to the rated power by changing the blade pitch.

The rotor power as function of wind speed and rotor speed is plotted in figure 2(a). It can be seen that maximum power at each wind speed is related to an optimal rotational speed. The torque and power at the rated speed is shown in figure 2(b). The rotor power is maximum (108 kW) at the optimal speed of 96 rpm but the torque is a maximum (13.5 kNm) at 67 rpm. This effect has to be taken into consideration during sizing of the hydrostatic transmission.

HYDROSTATIC TRANSMISSION WIND TURBINE

A hydrostatic transmission consists of a hydraulic pump driving a hydraulic motor. For a continuously variable transmission, at least one unit must have variable displacement. The hydraulic circuit of the transmission neglecting charge pump and cooling is shown in figure (3). The rotor drives the fixed displacement pump and the variable displacement motor drives the generator. This choice takes advantage of commercially available hydraulic components, control simplicity, transmission efficiency and cost, and is therefore the most practical solution. The closed circuit HST is chosen, eliminating the need for a bulky reservoir, making the transmission more compact.

The output pressure (P) of the pump is directly proportional to the applied torque (T), as shown in Eq. 3. Where $D_p$ is the displacement of the pump in cc/rev.

$$ T = \frac{PD_p}{2\pi} $$

(3)

Based on the rotor torque shown in figure 2(b) and the maximum pressure of the pump (350 bar), a 2512 cc/rev Hägglunds radial piston pump was selected. The large displacement pump is suitable for low speed and high torque application.

The HST decouples the generator speed from the rotor speed, allowing the generator to run at synchronous speed (1800 rpm) with time varying wind speeds. This also eliminates expensive power converters. The hydraulic motor converts the hydraulic flow (Q) to match the synchronous speed of the generator ($\omega_g$).

$$ Q = \omega_r D_p = \omega_g x D_m $$

(4)

As shown in Eq. (4), the ratio of rotor speed ($\omega_r$) to generator speed is proportional to the displacement ratio. $D_m$ is the full displacement of the motor and $x$ is the fraction of displacement. A variable displacement 135cc/rev Linde axial piston motor was selected to operate the motor at full displacement at optimal rotor speed.

In the HST, there are fluid losses through the components. To make up the case drain and leakage losses a charge pump is added to the circuit. Assuming a 75% volumetric efficiency of the HST, a 36cc/rev fixed displacement gear pump was used in the charge circuit. The mechanical losses of the transmission heat up the hydraulic fluid. A cooling system is added to the HST to draw heat from the fluid. Assuming a 70 percent
mechanical efficiency of the HST, 30 kW of heat needs to be removed for a 100 kW system. A shell and tube type, water cooling 40 kW of heat exchanger was added.

POWER REGENERATIVE TEST PLATFORM

The power regenerative research platform consists of two closed loop hydrostatic circuits as shown in figure 4 (a). The block in dark gray is the hydrostatic transmission under investigation as described above. The block in light gray is the hydrostatic drive (HSD), to simulate the rotor driven by time-varying wind. The output power of the HSD is fed to the pump of the HST through the rotor. The rotor is supported by two spherical roller bearings and the motor and pump are mounted on the splines of the shaft. The assembly is shown in figure 4 (b). The rotation of the hydraulic pump and the motor are restricted by the torque arm.

FIGURE 4. (a) Schematic of Power Regenerative Test Platform (b) Rotor Assembly

The HSD consists of an axial piston variable displacement pump (180 cc/rev) and a large fixed displacement radial piston motor (2512 cc/rev). The pump has an integrated internal charge pump to make up for the volumetric losses in the HSD circuit.

FIGURE 5. Power Regenerative Test Platform

The inertia of the rotor has a considerable effect on the dynamic behavior of the turbine. It is not practically possible to install a large flywheel on the test platform. As a result, the rotor on the test platform has significantly lower inertia than the rotor in the real turbine. To simulate the real dynamics of the rotor of a turbine, it is necessary to consider effect of large blade inertia in the control of the research platform. This can
be done by introducing a modified input torque ($\tau_d$) of the low speed shaft ($J_s$) to compensate for the effect of the large inertia. The modified torque of the rotor is given by

$$\tau_d = \tau_r - (J_r - J_s)\dot{\omega}_r$$

(5)

Where $\tau_r$ is the rotor torque of the actual turbine, $J_r$ and $J_s$ are the moment of inertia of actual rotor and testbed rotor. The modified aerodynamic torque ($\tau_d$) is simulated by controlling the line pressure in the HSD, by changing the displacement of the variable motor. The tracking of the torque/pressure will be performed by a feedback controller.

Instead of dissipating the turbine output power, the power is fed to the HSD along with electric power, allowing power regeneration. Assuming 80% overall efficiency of HST and HSD each, a total of 55 kW of power losses in the circuit is compensated by the electric motor. A three phase, four pole, squirrel cage induction motor with synchronous speed of 1800 rpm (120f/p) is mounted on the high speed shaft, between the pump of the HSD circuit and the motor of the HST circuit. The Variable Frequency Drive (VFD) of the electric motor is set to control the high speed shaft at constant synchronous speed. This enables the test stand to operate with less electric power and eliminates the use of resistors as the load. The power regenerative test platform is shown in figure 5. The HST and HSD loops of the test platform have independent hydraulic circuits so that fluid in each circuit can be varied. Each circuit is equipped with a suction and a return line filters to clean the oil and an independent oil cooler to control the oil temperature.

Sensors and Data Acquisition

The test platform is equipped with 27 sensors to monitor the system performance and three analog inputs to control the displacement of the HST motor, the displacement of the HSD pump and the speed of the electric motor. The pump and motor displacement are controlled by adjusting the swash plate angle with a solenoid valve requiring 225-600 mA for the HST motor and 280-740 mA for the HSD pump. The DAQ cannot provide the high currents required for the swashplate solenoid, so to precisely control the swash plate angle an electronic valve control (EVC) is used. The EVC can be used in simple proportional single coil or dual-coil applications even for more complex closed-loop pressure/speed control. The EVC is configured by its own graphical user interface for two independent single coil operation with PWM a frequency of 100 Hz for each coil. The power to the EVC is supplied independently to eradicate noise of 100Hz and its harmonic frequency in other sensors.

The speed of the electric motor is controlled by the VFD by varying the input frequency and voltage. The VFD chops the signal at 4 kHz for desired frequency and voltage. Input command of 0-10 VDC is supplied from the DAQ to control the electric motor speed from 0-1800 RPM.

The rotor shaft speed is measured with a speed encoder mounted at the end of the shaft. Rotational speed sensing unit, Hägglunds SPDC, is a digital incremental encoder using magnetic sensing. The sensor generates two square wave signals with 90° phase shift for detection of speed and direction of rotation. For connection, the F/A Converter converts a single pulse train to a 4-20 mA output signal. The output is based on an internal reference frequency selectable in 16 steps and connected to the DAQ board.

The torque ($\tau_r$) is calculated from the force (F) measured on the load cell with torque arm of distance (d) 0.6m. The size of load cell is calculated by the equation given below.

$$\tau_r = Fd = \frac{PD_p}{2n\eta_{mp}}$$

(6)

The system is designed for 350 bar pressure (P). Assuming 90% mechanical efficiency ($\eta_{mp}$) of the fixed displacement motor (2512 cc), the maximum force on the load cell is 26 kN. A load cell of a capacity of 33 kN is installed to measure the force. The 3mV/V load cell output is amplified with an inline amplifier to 0-10 VDC.

The Activa Sensor Array module consisting of pressure, temperature and flow sensors is mounted in line with the high pressure and low pressure lines of the HSD and HST as well as the return lines of the HST to monitor the performance of each component. This compact unit only requires one hydraulic line break. The module has a turbine type flow sensor with an analog output of 0-5 VDC. The pressure and temperature sensors generate 4-20 mA analog signal outputs.
In the high speed shaft, two rotary torque sensors were installed to measure the torque and speed. One is mounted between the electric motor and the HSD pump and the other is mounted between the HST motor and the electric motor. The sensitivity of the torque sensor is 4 mV/volt, which is prone to noise. Thus, this signal requires a transducer inline amplifier. The 4 kHz chopping frequency of the VFD make the torque signal distorted due to aliasing. The signal was analyzed by using an oscilloscope. The Fast Fourier Transform (FFT) spectrum of the signal is shown in figure 7(a). A fourth order active, multiple feedback, low pass Butterworth filter was designed with cutoff frequency of 100 Hz. The transfer function of the filter is shown in Eq. 7 and implemented by two cascade op-amp circuits. The performance of the filter was compared with the Bode plot of the transfer function as shown in figure 7(b). The torque sensor is integrated with 60 toothed gear and a magnetic pick up sensor to measure the shaft speed.

\[
TF = \left(\frac{3.948 \times 10^5}{s^2 + 1161s + 3.948 \times 10^5}\right) \times \left(\frac{3.948 \times 10^5}{s^2 + 480.9s + 3.948 \times 10^5}\right)
\]  

(7)

Analog voltage and current signals from all sensors are collected by two terminal boards. The 4-20 mA analog current signal is converted to 0-10 VDC by using a 500 ohm resistor. The noise from environmental electromagnetic interference was avoided by using shielded and twisted pair cables. To avoid ground loops a single ground was created for all sensors and supply line. There is a common supply of 24 volts for all sensors.

The signals from the sensors are sampled by the DAQ, which converts the analog signal to digital numerical values that can easily be read by a computer. A NI-6259 DAQ, from National Instrument is used for the testbed. It is a high-speed multifunction M Series DAQ board designed for PCI slot with 32 analog inputs and 4 digital outputs at 16 bit resolution. Each analog input is capable of capturing two types of signals: voltage drops \((\pm 5, \pm 10V)\) and currents \((0-20mA)\). The NI 6259 is connected to the PCI slot which is located in the target computer. Then two cables from the NI 6259 divide the channels and connect to the terminal boards. The two terminal boards collect all the signals from the sensors. In a DAQ, a multiplexer switches from one channel to the next to read the signal. High source impedance on a scanned channel causes its settling time to increase and hence reads the previous channel data. This is called the ghosting effect. To avoid ghosting and crosstalk the multiplexing frequency was decreased to 100 kHz.
Matlab xPC target is used to execute a Simulink model on the target computer for fast prototyping, hardware-in-the-loop (HIL), and other real-time testing applications. xPC Target is a real-time software environment from MathWorks that can manipulate several separate workstations at one time. It also transfers data back to the host machine while reading inputs and issuing commands to the data acquisition boards. The xPC target allow us to employ model-in-the-loop techniques to emulate rotor torque by considering blade aerodynamics and blade pitch. It also allow us to implement a real time control strategy on the testbed.

CONCLUSION

The design of a power regenerative hydrostatic test bed is described in this paper. This test bed allow us to test the performance of the hydrostatic transmission at different wind conditions. Test bed is equipped with sensors to monitor the system behavior. The testbed can emulate rotor torque of 14 kNm which is equivalent to a 100 kW rotor power turbine. This test platform provides a powerful tool to investigate the performance of the new components, controllers, fluids and energy storage method on the hydrostatic wind turbine transmission.

ACKNOWLEDGMENTS

This project is funded by the National Science Foundation under grant #1634396. We also thank Eaton, Linde, Danfoss, Bosch Rexroth, Flo-tech and ExxonMobil for donating the components for the test bed. We are thankful to Ching-Sung Wang, a visiting scholar from National Taiwan University, for working on the data acquisition and control system for the test platform.

REFERENCES

WAVE POWER CONVERTER PENDULOR WITH HYBRID H.S.T.

Tomiji WATABE* and S.D.G.S.P. GUNAWARDANE **

* T-Wave Consultant
5-23-3, Misono, Noboribetsu, 059-0036, Japan
(E-mail: t1watabe@rg8.so-net.ne.jp)

** Dept. of Mechanical Engineering, Faculty of Engineering, University of Peradeniya
Peradeniya, SriLanka
(E-mail: sdgs@gsp@pdn.ac.lk)

Abstract. This Pendulor: wave energy converter has been invented to be robust towards storms. The key is Hybrid H.S.T. for generator driving with changing the speed and torque of the piston pump which can work easier under over 360° free rotation of the Pendulum motion (Primary Energy Absorber). The idea has brought non-shock operation in the moving type wave energy conversion. It shall also give us a future possibility to present all the people with Cheap Electricity generated from the Ocean by this technology.

Keywords: Ocean wave power, Conversion, Pendulor, H.S.T., Hybrid, Large power plant, Pelamis, Oyster.

INTRODUCTION

Muroran Institute of Technology Japan developed the Wave Energy converter Pendulor 39 years ago. The research advanced well at the beginning but faced a fundamental problem on the survivability towards storms. Since the original Pendulor depends on moving body type energy conversion, the system must have a stopper for the pendulum to be kept the stroke within a limit, in any time of operation (1)(2). The stopper should be a shockless at the stopping of the massive Pendulum. If we cannot solve this matter, the dream to make electricity from the Ocean Waves would be impossible. This new Pendulor has changed the system completely safe with no stopper (3). The pendulum can rotate over 360° both directions freely by applying with the new Hybrid H.S.T. for generator driving. Plural number of piston pumps driven by one speed up gear, deliver much flow of oil by all together. The system and the pumps have been studied for the practical wave energy conversion with the Pendulor. This system shall be able to develop a large Pendulor in the future for peaceful world. (Three Patents Pending)

BIRTH OF THE OCEAN WAVES AND USE OF THEM

Blowing wind on the ocean excites the sea water at the surface so that there appears periodical moving of the water (ripple); birth of the waves. The growth of the sea waves depends on the wind power, lasting the blowing time and along to a place follows to the wind pass way from beginning and lasting to the final place. It tells that the waves are phenomenon altered from the wind energy, and it propagates from the birth place, as it to be travelling waves. They are classified themselves within near sized waves together while as travelling through such a long distance. Therefore, utilization of the wave energy; -applied with the antenna principle, it can be done without difficulties. (Muroran I.T. took the principle from the beginning stage). (4) For development of the large Pendulor, driving it with energy rich waves was studied from a basic viewpoint. The reason is that, the Pendulor must do resonant operation with the incident waves and the generator load condition must be adjusted in coincide with the impedance of the Pendulor device for the optimal driving. Therefore, the Pendulor study cannot do anything without understanding on the incident waves for optimization of the wave power conversion. FIGURE 1 (5)(6) is the first floating type Pendulor which has been studied in Korea after Muroran I.T. closed the study in 2000. This device has applied new ideas; (1) Floater to be stable by active damping with the wave forces, (2) Optimization control of wave energy conversion, (3) Giant Rotary vane pump direct coupled with the pendulum, (4) Oil seal fits to the large pump. Japan created the ideas and Korea challenged developing of the plant by applying the technologies (7).

WAVE POWER DENSITY IN THE WORLD

FIGURE 2 shows the wave energy distribution in the world, investigated by US Dept. of ENERGY by kW/m. The Department estimated of the world wave energy potential; That is at 2 or 3 million MW. In favorable location, the wave energy can average 40 MW/km (= 40kW/m of coastline) (8). Comparing with the power density of Japan and the world, Japanese one is only 1/3 of the world. Nevertheless, the giant level typhoon...
frequently attacks Japan lands with over 10 times stronger power. The Pendulor must survive towards them to accept not such the strongest machine but a clever and inexpensive one. The machine lets the dangerous waves go back to the sea as it does no energy conversion.

WAVE POWER FOR LARGE PENDULOR

Considering the magnitude of the wave energy potential on the EARTH, we must pay attention on the big energy converter driven by high energy density waves. Here introduces a study on the waves which grown by several wind conditions. Since the glowing wave height and its period can be calculated when the wind speed and the fetch length are given, applying with the data shown in the chart (in FIGURE 3). The authors tried
some investigations to find preferable waves which are good for driving the large Pendulor. FIGURE 3 and 4 show the wave height: $H_{1/3}$ (m) and its period: $T_{1/3}$ at given wind speed (m/s), blowing duration (hour) and distance (Fetch length, km). In case of 10 hour wind blow with several wind speed, glowing wave height estimated are shown in FIGURE 4 & 5. The wave height: $H_{1/3}$ increases as wind speed being higher. For the wave period: $T_{1/3}$, it becomes longer (by slower wave motion) excited with the higher speed wind. It means the wave motion changes slower as the blowing wind to be stronger. Therefore, when a large Pendulor is driven by energy rich waves, the richer the energy density, the operation condition of the Pendulor becomes the slower speed and the higher torque operation. This result tells us an important direction to challenge to the wave energy utilization with a large converter, the system must overcome the slower speed and the greater torque load operation. The authors decided to attack to this study with new ideas: shown below.

**NEW IDEAS FOR THE LARGE PENDULOR:**
(1) Survival to the Storm by no-stopper $\pm 360^\circ$ free rotation, (2) Invention of, Hybrid H.S.T. for Generator driving, & (3) piston pump can work at the condition, well

**EXPERIENCE ON SYSTEM SURVIVABILITY**

Since the Wave energy conversion depends on the reciprocal wave motion, the action parts by waves must be limited even when the device encounters storms. The wave power of storm would be over 10 times greater level than the normal, the shock by impingement to the stopper becomes often offers a damage situation to the system. During sea operation of a Pendulor, we experienced three times of accidental breaking of the flapper legs by

---

**TABLE 1** The Wind-Waves grown by Several wind speeds

<table>
<thead>
<tr>
<th>Case No.</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Terms</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wave height $H_{1/3}$ (m)</td>
<td>3.0</td>
<td>4.5</td>
<td>5.4</td>
<td>6.6</td>
</tr>
<tr>
<td>Wave period $T_{1/3}$ (s)</td>
<td>7.0</td>
<td>9.0</td>
<td>9.5</td>
<td>10.5</td>
</tr>
<tr>
<td>Wind speed, (m/s)</td>
<td>15.5</td>
<td>20</td>
<td>22.5</td>
<td>25.0</td>
</tr>
<tr>
<td>Duration (H)</td>
<td>10</td>
<td>10</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>Minimum fetch length, (km)</td>
<td>40</td>
<td>52</td>
<td>58</td>
<td>65</td>
</tr>
</tbody>
</table>

**FIGURE 4** Period change by growth of Wave

**FIGURE 5** Wave Characteristics(9) Relationship between blowing wind on the sea and glowing waves
impingement between the stopper and the legs. The happening gives us a hint to avoid the trouble; - by not to improve the legs stronger but to change the condition safer; - by no use of the stopper. We selected the Hybrid H.S.T. system for the Pendulor which requires no stopper as shown in FIGURE 6.

In FIGURE 6, a single pendulum drives four sets of geared piston pumps of the H.S.T. which drives a Generator. Since the gear is to be speed up use, it drives the four pumps with increased speed with divided power of 1/4 each, so, the displacement of the pump/ unit becomes to the value shown in EQUATION (1).

\[ D_p = D_0 \times \frac{n_p}{n_g} \ldots \text{(1)} \]

Here, \( D_p \): displacement of the pump when it is driven directly. In a case of 4 sets of pumps with teeth number of the pinion: \( n_p = 7 \) and teeth number of the gear: \( n_g = 70 \). \( D_0 \) takes the value shown by EQUATION (2), that is the case of FIGURE 6.

\[ D_p = D_0 \times 1/4 \times 7/70 = 0.025 D_0 \ldots \text{(2)} \]

This result means that the system of FIGURE 6 can reduce the pump capacity required drastically, comparing with the former system of one set pump. The Hybrid HST of Fig. 6 has no stopper because of the system permits over 360° rotation of the pendulum in either direction. The pendulum motion amplitude: \( \theta_a \) is shown by Equation (3) as a function of the incident wave height: \( H \).

\[ \frac{\theta_a}{Z_0} = \frac{k_0 Z_0}{4Y_0 \sinh k_0 h} \times \frac{H}{2} \ldots \text{(3)} \]

\[ Z_0 = \sinh 2k_0 h + 2k_0 h \ldots \text{(4)} \]

\[ Y_0 = k_0 \sinh k_0 h + \cosh k_0 h - 1 \ldots \text{(5)} \]

Here, \( k_0 \): wave number, \( H \): wave height and \( h \): water depth

\( l \): distance between the center of pendulum shaft and the water surface.

Piston pump of FIGURE 6 is non-rotating type. Its feature is much simpler than the rotary type piston pump and being strong towards the cavitation and dynamic load because of a tiny moment of inertia. The pump can be inherently to fit to the reciprocal rotation as well.

**IMPROVEMENT OF THE SURVIVABILITY OF THE PENDULOR**

The cause of damage on the wave energy converter observed was by shock loads most, not only on the Pendulor but also the Pelamis\(^{11}\) FIGURE 7 and, the Oyster\(^{12}\) FIGURE 8. The shock load happened both places with the stoppers and the mechanical power- transmission where small gap existed between the parts faced with two of them which cannot become within a one part. The gaps make shock by impingement. In the case of impingement at the stopper, there likely concerns with big energy on it. Therefore, some components of the stopper are damaged. For instance, the tightening bolts of the stopper to be loosened at beginning then the all bolts were broken away to the final stage. The incident lets us make the mind, to improve the power take off
components to be improved for being reliable much more. The decision is, to exchange the design principle with no use of the stopper. This idea has been realized by the Hybrid HST developed shown in (FIGURE 6).

**DESIGN OF THE HYBRID H.S.T.**

FIGURE 9 is a trial design with 16 pumps to make a set of geared pumps for the Hybrid HST. This idea is for a MW class study on the large Pendulor. With 16 pumps and given the speed ratio: \( z_1/z_2 = (120/9) = 13.33 \). It seems one of the balanced features from the engineering and the economic standpoint. The case shown in FIGURE 9, the pump speed is increased in 13.33 times higher than the case of no gear driving, 16 pumps of the displacement \( D_p \) is shown by Equation (6).

\[
D_p = D_O \times \frac{1}{16} \times \frac{6}{120} = D_O \times \frac{1}{320} \quad \cdots \quad (6)
\]

Therefore, the case of Fig. 9, the 16 pumps have the 320 times small displacement each is enough for the pumps of the hybrid HST. Despite of such as 16 pumps drive even in parallel, each the load is divided into each the pump exactly by an error absorption effect to the teeth pitch error of the pumps. For design on the 7 teeth pinion of the Hybrid HST, which have no under cutting by the interference on the pinion of 7 teeth. That is an allowable limit condition with the sharpened teeth tips.

For the gear lubrication of the high tooth load/ slow speed operation, attentions written below are useful. (1) The teeth are hardened by nitriding process, polished by lapping to make 0.5 \( \mu \)m flatness. The error of tooth profile and pitch are not important in case of slow rotation (under 100rpm). (2) For the teeth strength, especially to contact pressure, we should pay attention carefully (oil film thickness at the contact point etc.). (3) For the gear design, the contact point to be pure rolling as perfect as possible. This kind of care makes steady lubrication by Elasto-Hydro-Dynamic Lubrication (EHL). This Hybrid H.S.T. idea can be used also for wind turbine. Because the system drives high speed generator with very slow speed turbine. (11)
PISTON PUMP

FIGURE 10 is the pump designed for the Hybrid HST for study purpose of the large Pendulor. It belongs to non-rotating axial piston pump group having five pistons in a pump considering the abnormal slow speed operation. The pump can be used as a variable type or fixed one when the suction check valves are kept in open or in close. In case of the large Pendulor, (and FIGURE 9 is the Hybrid pump for it,) the displacement $D_p$ required for the hybrid piston pump is shown by the equation (6)

STUDY ON THE PISTON AND THE SHOE

The large Pendulor has an energy loss problem because of the HST which is oil leak at the slow speed operation. Axial piston machine uses swash plate to convert reciprocal motion to rotary motion. A conventional hydro-static bearing has been used for the swash plate. This component does make the efficiency drop of the HST. The author experienced an observation of the swash plate machine which happened metal contact whenever starting. The friction by the metal contact accompanied with a fair amount of oil leak which made the condition worse, too. FIGURE 11 is a piston and a shoe of the axial piston pump/motor invented to overcome the energy loss by the metal contact at slow motion. The principle is applied both in the piston and on the shoe as shown in FIGURE 11. On the surface of the static bearings (piston and shoe), there are some thrust control pools of which pressure is controlled by the outer load, irrespective of running speed. The oil pressure at a place is the function of flow-in and flow-out oil volume, the external load controls the pressure of the control pools, and the pool pressure adjusts the lateral clearance of the bearing pads non-uniformly. So, it produces the bearing action change; no metal contact. (Patent Pending). The thrust pools locate on the piston

FIGURE 10 Piston Pump for Hybrid H.S.T

FIGURE 11 Piston and Shoe with self-balancer (Patent Pending)
surface, too. The pool pressure is controlled when the load balance approves the lateral load and the reaction relationship caused by the oil pressure. FIGURE 11 shows that the piston clearance is controlled to move the piston center. Therefore, when the pressure is bigger, the lateral force becomes the bigger. The principle: (1) The piston configurations are located right to left; ball formed joint, clown formed seal, pools located on the taper formed guide. The piston can rotate within the lateral gap by the tapered guide which affects pressure change of the pools. (2) When a normal vector pushes the shoe at the sliding center shown in FIGURE 11, one force vector component of up direction, drives the piston by anti-clock wise moment. Therefore, on the taper part of piston, upper side clearance becomes larger and lower side one becomes minimum. (3) Then, this change makes the pressure of upper side pool the higher and lower side pool the smaller. The pressure difference between up and down side pushes the piston down direction which acts force to the piston with the opposite direction of the load. If the oil pressure cancels the piston load, the minimum friction loss would be possible applied with this technology. (Patent pending)

CONCLUSIONS

The idea of Pendulor with Hybrid HST can be concluded as below. (1) The new idea has been shown here that the Pendulor can be survival against the storm. It is realized by applying with Hybrid HST assembled into the Pendulor system. (2) This idea contributes to the conversion system not only being survival in the storm but also open a new possibility to develop MW class large Pendulor by Elasto-hydrodynamic lubrication (EHL) technology for the high torque and slow speed operation. (3) The axial piston pump of small moment of inertia, can improve the Pendulor efficiency at high torque low speed operation. This idea on the Hybrid HST is suitable for the large wind turbine application, too. (4) The converter on Moving Body type, “Pelamis and Oyster”, the cause of failure was studied to overcome the problems in the future.

ACKNOWLEDGMENT

Ms. C. Nagaishi donated 1.5 million yen for the study and Saga University Japan supported the research.

REFERENCES

1. T. WATABE, Utilization of the ocean wave energy, Fuji Print Press, 2008.
DISC BRAKE WITH HYDROMECHANICALLY CONTROLLED BRAKE TORQUE FOR RAILWAY APPLICATIONS

Matthias PETRY*, Ahmed ZAKI*, Hubertus MURRENHOFF*

*Institute for Fluid Power Drives and Controls (IFAS), RWTH Aachen University
Steinbachstraße 53, 52074 Aachen, Germany
(E-Mail: Matthias.Petry@ifas.rwth-aachen.de)

Abstract. Hydraulic disc brake systems are widely spread in commuter train applications. Due to their operation with an open loop control system, they are not able to deal with the influences of varying conditions surrounding the frictional contact zone between the brake pads and the disc. This results in a variable friction coefficient and affects the system performance. To detect the variation a closed loop control system with a feedback signal needs to be implemented in existing brake solutions to maintain high reliability and operating permit. This paper examines the design features of an existing system and finds a suitable mechanical feedback signal, whose eligibility is experimentally verified. Subsequently the geometrical calculations of the system are made and the control system is simulated. Based on the simulation results a preliminary design of the system is presented.

Keywords: Brake system; brake torque control; closed loop system; railway application.

INTRODUCTION

Railway vehicles are an efficient, reliable and eco-friendly mass transportation solution, which still provide a vast potential for further development despite their long existence. The brake system is an essential part of a functional railway vehicle. For the implementation of various braking functions, such as general operation, emergency, holding or parking, both passive spring-applied actuators and actively acting brakes are used. The adaption of the contact force between brake pads and disc or wheel can be affected both gradually and continuously. Especially in case of brake operations with wheel slide protection, a continuously variable control of the contact force offers the possibility of an improved brake performance. However, all conventional disk brake systems have the common inability to record the influence of varying conditions in the frictional contact zone between brake pads and disc. Thus, the brake performance of these systems is dependent on a changing frictional coefficient due to external disturbance variables. Additionally, the prevailing operating conditions of commuter trains are characterised by harsh weather conditions, severe vibrations, strong mechanical loads and a high number of actuation cycles. Figure 1 (a) illustrates a conventional hydraulic brake system which is affected by a large number of various disturbances.

The brake cycle of a consisting system begins with the signal from the brake arm shifted by the train driver. The displacement $s$ is transferred into the electrical signal $i$ to the brake pressure valve. According to the signal the valve increases or decreases the hydraulic brake pressure $p_B$ in the brake cylinder. Inside the brake cylinder, the normal force $F_N$ is generated and applied on the brake pads and subsequently the brake disc. Throughout those processes the brake signal is disturbed due to frictional losses and wear. The normal force and the friction coefficient between the pads and the disc induces the frictional force $F_R$ on the wheels, resulting in reducing the angular velocity of the wheel $\omega$ as depicted in figure 1 (b). Due to the contact between the slipping wheel and
the rail a brake force $F_R$ affects the vehicle. In the contact zones wetness, wear, temperature, vehicle velocity, and the surface roughness of the wheel cause disturbances. The driver perceives the deceleration $a$ of the vehicle as a feedback from the open loop and builds a man-made closed loop brake system.

One option to compensate the introduced disturbances and to control the actual brake torque is to use the self-energised hydraulic brake (SEHB) which was developed at the Institute for Fluid Power Drives and Controls (IFAS) [1][2]. It does not require an external power supply and provides a reduced installation space. Nonetheless, the primary structure of the SEHB differs considerably from a conventional hydraulic brake system, which accompanies the market introduction by various uncertainties and difficulties. At the same time, narrow specifications request brake performances with more stringent criteria. While the brake system has to make the optimal use out of the frictional contact, the compliance with minimum brake distances for comfort reasons present challenges for the design and operation of modern brake systems. An innovative option is the integration of a hydromechanical brake torque control system into an established hydraulic brake design for commuter trains. For this purpose, in the first section of this paper the structure of a selected brake system is analysed for its ability to detect the real frictional force $F_R$ with high reliability. A suitable mechanical feedback signal is found to build a closed loop system, which is developed and simulated in the following sections. The last section gives an overview of the preliminary design.

**ANALYSIS OF AN EXISTING BRAKE SYSTEM**

The selected brake system consists of a brake cylinder, a caliper, brake pads, and a supporting structure. **Figure 2 (a)** shows the integration of the brake into the bogie which includes the hydraulic supply and control elements. The brake is supported by two bearings in the bogie. The first one at the brake enclosure leads to a rotatable connection of the brake while the second one builds the supporting structure with a lever arm and fixes the brake position. During a brake process a frictional force $F_R$ is generated and directed through the supporting structure towards the bogie resulting in a supporting force $F_S$.

![Figure 2](image)

**FIGURE 2.** (a) Working concept; (b) Block diagram of the actual open loop system.

The open loop system is presented in **Figure 2 (b)** as a block diagram. The brake pressure $p$ is applied on the piston of the brake cylinder resulting in the hydraulic force $F_N$ as shown in equation (1).

$$F_N = A_{Piston} \cdot p - d_x \cdot \dot{x}_{Piston}$$ (1)

Hereby the hydraulic force $F_{Hyd}$ is diminished by the frictional force of the brake cylinder depending on the cylinder speed $x$ and its friction coefficient $d_x$. Multiplied by the number of brake pads and the friction coefficient $\mu$, the frictional force $F_R$ is achieved. The supporting force $F_S$ can be calculated with a factor $\alpha$ that depends on geometric relationships like the suspension $\Delta y$ of the brake disc axle towards the bogie as described in equation (2).

$$\alpha = \frac{F_S}{F_R} = f(\Delta y, ...)$$ (2)

With the supporting force $F_S$ as representative value for the real friction force the development of a closed loop control system is possible. However, bringing a control mechanism into the supporting structure will lead to a length variation $\Delta x$ of the lever arm, which in turn affects the factor $\alpha$. To investigate the influence of possible
deflections the mechanical structure of the brake system is implemented into the geometrical processing software Geogebra. **Figure 3** shows the brake model in its end positions on the brake disc due to maximal deflection values.

**FIGURE 3.** Force and position reaction due to suspension travel $\Delta y$ (1),(2) and lever arm length $\Delta x$ (3),(4)

The suspension travel $\Delta y$ between the geometrical center point of the brake pads and the brake disc variates between -12 mm (1) and 20 mm (2) while the length $\Delta x$ of the supporting structure will be changed between -10 mm (3) and 10 mm (4). Obviously, a change of the suspension travel $\Delta y$ leads to a rotating friction force $F_R$ and thus to a varying lever transmission between $F_R$ and the supporting force $F_S$. The change of the lever arm length $\Delta x$ also directly influences the transmission ratio due to shifting the brake pad’s position on the brake disc. **Figure 4** illustrates the results for the factor $\alpha$ calculated by Geogebra.

**FIGURE 4.** Force ratio $\alpha$ depending on suspension travel $\Delta y$ and supporting structure length $\Delta x$

The factor $\alpha$ changes linearly for both displacements and shows only small deviations of 5% to 7.5%. In real operation, only small deflections are expected. Therefore, it is assumed that the supporting force $F_S$ is nearly proportional to the frictional force $F_R$ and can serve as a feedback signal and indicator for the real brake force of the system. Nonetheless, possible effects will be considered in simulation by varying displacement values. The quality of the supporting force $F_S$ is experimentally verified using a brake test rig shown in **Figure 5 (a)**.
The brake disc and an additional inertia mass are driven electrically. The supporting structure with shortened lever arm is connected to the test rig frame with a force sensor. Figure 5(b) displays the results, which show the ratio between the measured supporting force $F_S$ and the brake torque $M_{Br}$ respectively the ratio of the normal force $F_N$ and $M_{Br}$ next to the calculated velocity. While the ratio $F_S/M_{Br}$ is nearly constant during brake, the ratio $F_N/M_{Br}$ changes crucially between and during the brake intervals. Additionally, the measured supporting force occurs in a technically useful span with approximately 10% of the normal force’s value. Hence, the supporting force $F_S$ can indicate the real brake torque $M_{Br}$ and the supporting structure can be used for a hydromechanical control system.

CLOSED LOOP SYSTEM CONCEPT AND SIMULATION

After verifying the eligibility of the feedback signal $F_S$, a principle concept with hydromechanical elements is defined. Figure 6 shows the simplified working concept of a closed loop brake system.

![Diagram of closed loop concept](image)

The control loop is closed using a 3/3 proportional hydraulic valve balancing between a demanded supporting force $F_{S,\text{Set}}$ and the current supporting force $F_S$. The brake force demand is provided by the original hydraulic control system of the vehicle. The current brake force is transmitted using the supporting structure with the factor $\alpha$. In case of over braking the supporting force exceeds the set brake force. Thus, the valve connects the brake cylinder with the tank resulting in reduced brake force until the equilibrium is reached again. In case of under braking the set brake force is higher than the supporting force and the valve connects the brake cylinder...
with the supply pressure. A mathematical model of the system is built to determine significant parameters for the system design. First, the equilibrium of forces around the valve shaft is considered in equation (3)

\[ m_{\text{Valve}} \cdot \ddot{y} = \sum F = F_{\text{S,Set}} - F_S - c_f \cdot y - d_{\text{friction}} \cdot \dot{y} \]  

(3)

where \( m_{\text{Valve}} \) represents the mass of the valve spool and \( d_{\text{friction}} \) a friction parameter depending on the spool velocity. The linearized equation of the hydraulic flow in a valve is introduced in equation (4) [3] and applies for both the inlet and the outlet flow.

\[ Q = \frac{\partial Q}{\partial y} \cdot p - \frac{\partial Q}{\partial p} \cdot \dot{y} = V_{Qy} \cdot y - V_{qp} \cdot p \]  

(4)

The flow factors \( V_{Qy} \) and \( V_{qp} \) express the dependence on valve opening \( y \) respectively the change of pressure \( p \). The leakage flow is determined using the leakage pressure factor \( K_{\text{lecp}} \) as follows in equation (5).

\[ Q_{\text{lec}} = K_{\text{lecp}} \cdot p \]  

(5)

Subsequently equation (6) gives the pressure in the system with \( C_H \) as the hydraulic capacity.

\[ \dot{p} = \frac{1}{C_H} \cdot \sum Q = \frac{1}{C_H} \cdot (Q_{\text{in}} - Q_{\text{out}} - Q_{\text{lec}}) \]  

(6)

Figure 7 visualizes the hydromechanical control system in a block diagram. The system is divided into four subsystems: The pilot valve, the mechanical modelling of the main valve, the hydraulic modelling of the main valve and the brake mechanics. The control loop is closed by returning the feedback signal \( F_S \) to the valve forces sum point.

Equation (7) states the transfer function of the system after linearization.

\[ G(s) = \frac{F_S}{F_{\text{Set}}} = \frac{(V_{Qy1} - V_{Qyp}) \cdot V_{\text{Piston}} \cdot 2 \mu \cdot \alpha}{A \cdot s^3 + B \cdot s^2 + C \cdot s + D} \]  

(7)

with

\[ A = m_{\text{Valve}} \cdot C_H \]  

(8)

\[ B = m_{\text{Valve}} \cdot (K_{\text{lecp}} \cdot V_{Qp1} + V_{Qp2}) + d \cdot C_H \]  

(9)

\[ C = d \cdot (K_{\text{lecp}} \cdot V_{Qp1} + V_{Qp2}) + c_f \cdot C_H \]  

(10)

\[ D = c_f \cdot (K_{\text{lecp}} \cdot V_{Qp1} + V_{Qp2}) + (V_{Qy1} - V_{Qyp}) \cdot A_{\text{Piston}} \cdot 2 \mu \cdot \alpha \]  

(11)

Using Matlab/Simulink, a parameter set is found which leads to stable system behaviour. Nonlinearities such as the volume flow’s square root dependence on the pressure difference and spool way limitations are considered in a simulation model in the one-dimensional simulation software DSHplus. Figure 8 illustrates the implementation of the brake system.
The mechanical part of the brake is implemented by using two cylinder models for both the main valve and the brake cylinder. The two chambers of the main valve piston are connected with the set pressure affected by a force demand, and the tank pressure. The supporting force signal $F_S$ is applied on the main valve piston so that the resulting hydraulic force equilibrium balances it. The output displacement $y$ of the main valve piston is connected with both control edges of the valve that represents the hydraulic part of the valve. Both control edges are connected to the same brake piston chamber and to the supply pressure and tank pressure on the other side respectively. When the main valve moves in positive direction control edge 1 opens to increase the braking pressure $p$ while control edge 2 closes. With a negative displacement $y$ the system reacts inversely and the braking pressure decreases. A throttle between braking pressure and tank leads to more damping. The brake piston is mechanically coupled with a spring-damper element to represent material properties of the pads and the disc [4]. Variations of the friction coefficient $\mu$ between brake pads and disc and the force factor $\alpha$ are set by additional signal generators.

**SIMULATION RESULTS**

Figure 9 depicts the simulation results of step responses due to switching force demands (a) and error response provoked by a decreasing friction coefficient $\mu$ (b).

The supply pressure $p_{\text{Supply}}$ of 100 bar corresponds to the maximum system pressure of the original brake. According to the force demand $F_{S\text{Set}}$ the valve opens and the pressure $p$ increases. After short settling time the friction force $F_R$ builds up and the current supporting force $F_S$ follows the demand $F_{S\text{Set}}$ precisely. At a higher pressure level the valve opens more to compensate the increasing pressure depending leakage flow. While in
this simulation the friction coefficient remains constant, for the investigation of the error behaviour the conditions between brake pads and disc are continuously changing. Figure 9 (b) shows the results achieved with a constantly reduced friction coefficient \( \mu \) to introduce a disturbance to the system. The system compensates the loss of the frictional force \( F_R \) directly by opening the main valve and increasing the braking pressure \( p \). As a result, the frictional force \( F_R \) and the supporting force \( F_S \) remain constant during the whole brake time. Hence, the system behaves as expected and fulfils its purpose as well. Figure 10 illustrates the results of the dynamic analysis of the system for various levels of the pressure \( p_{Set} \).

Using the Bode diagram, the dependence of the dynamic response on the input values can be observed. Sweeps of increasing frequencies at three distinct pressures \( p_{Set} \) (10, 50 and 100 bar) are performed. The supply pressure \( p_{Supply} \) is 100 bar again. Due to the nonlinearities of the hydromechanical system the dynamic response and the resonance frequency change between 20 and 80 Hz depending on \( p_{Set} \). While the dynamics of the system decrease as expected with rising set pressure, they are also reduced at very low pressure demands. Main reason for this behaviour is the flow factor \( V_{Qy} \) which depends on the pressure difference. Measures to improve the dynamics for this operation area would be a smaller valve opening or the reduction of the supply pressure.

**PRELIMINARY DESIGN**

An innovative design is created, that integrates the sensing tool of the feedback signal \( F_S \) and the control mechanisms of the brake pressure into the supporting structure. One of the aspects that needs to be considered while designing, is the direction of the supporting force \( F_S \) which changes with the direction of the vehicle wheels’ rotation from forwards to backwards. Therefore, the capture mechanism has to be able to indicate the supporting force \( F_S \) for both directions. It is also essential that, independent of the direction of the supporting force, an increase in the supporting force \( F_S \) is opposed with a decreased opening \( y \) of the valve, connecting the brake chamber to the tank and closing the supply pressure. For this function the relative motion between the valve sleeve and the valve spool must be in the same direction independently of the supporting force direction. Figure 11 schematically shows the concept of the bidirectional supporting force capture mechanism for increasing supporting force \( F_S \).

Figure 12 presents the main components of the valve integrated into the supporting structure in completely extended position for a pulling supporting force \( F_S \).
The valve mainly consists of four parts:
1. The valve spool with control grooves.
2. The valve sleeve that forms the metering edges as variable orifices with the valve spool. Additionally, it contains the ports for the four needed hydraulic connectors.
3. The supporting arm which is connected to the brake caliper.
4. The valve housing which is coupled with the vehicle bogie.

In pulling direction the supporting force $F_S$ moves the supporting arm and the valve spool to the right side. Due to the set pressure $p_{Set}$ and the area $A_{Valve}$, valve spool and sleeve are moving relative to a position where the metering edges between the supply, braking, and tank pressure achieve an equilibrium with the current supporting force $F_S$. In pushing direction the supporting arm moves left up to the valve sleeve and pushes it further - relative to the valve spool - until an equilibrium with the hydraulic force due to the pressure $p_{Set}$ and the area $A_{Valve}$ is reached again.

**CONCLUSION AND OUTLOOK**

Based on a conventional hydraulic brake system for commuter trains an advanced brake with hydromechanical brake torque control is presented. The force transmitted through the brakes supporting structure is identified and experimentally verified as representative value for the current brake torque. Simulation results show the functionality and good dynamics of the concept which is elaborated in a preliminary design.

Future research lays the focus on completion of the design process and building prototypes for an experimental verification of the concept. After gaining experience of this innovative brake system on a test rig a commuter train will be equipped with prototypes for extended field testing.

**ACKNOWLEDGMENTS**

This work was funded by the German Research Foundation (DFG) in the scope of the project “Schienenfahrzeugbremse mit hydraulisch-mechanisch geregeltem Bremsmoment” (MU 1225/40-1). The authors would like to thank DFG and Hanning & Kahl GmbH & Co KG for their support.

**REFERENCES**

RESEARCH ON THE EFFECTS OF DOUBLE ARC OIL GROOVE PARAMETERS ON TORQUE CHARACTERISTICS IN HYDRO-VISCOS DRIV

Yuanyuan DENG*, Zisheng LIAN**, Hongwei CUI**

* College of Mechanical Engineering
Taiyuan University of Technology
Taiyuan, Shanxi, 030024 China
(E-mail: yuanyuan.de@hotmail.com)

** Shanxi Key Laboratory of Fully Mechanized Coal Mining Equipment
Taiyuan University of Technology
Taiyuan, Shanxi, 030024 China

Abstract. To analyze effect of double arc oil groove parameters on shear torque characteristics, a numerical simulation flow field model of hydro-viscous drive(HVD) in full oil film shear stage was built, and flow field characteristics of velocity distribution, pressure distribution and shear torque were obtained. The parameter analysis platform of oil grooves was integrated with the parameterized design of oil grooves(CAD), numerical simulation of flow field(CFX) and design of experimental(DOE). The effects of double arc oil groove parameters, including the width of groove, the number of grooves, the eccentricity of double arc oil grooves, the diameter of inner eccentric and the depth of oil groove on the shear torque of oil film in HVD were analyzed by using orthogonal arrays. The relationship between the oil groove parameters and shear torque of oil film were analyzed.

Key words: Hydro-viscous drive(HVD), Double arc oil groove, Shear torque of oil film, Effect of oil groove parameters.

1. INTRODUCTION

HVD utilizes oil film to transmit power, and it is commonly used in mechanical equipment like fans, pumps, belt conveyors and scraper conveyors. It is the third fluid power transmission technique after hydraulic and hydrodynamic transmission. The double arc oil grooves are widely used in engineering industry, and the friction pairs constituting of friction plate(FP) and separator disk(SD) is key component. Therefore, accurate analysis of flow field characteristics and oil groove parameters is key step to predict torque characteristics and design oil groove parameters.

Most researches about the relation between shear torque and grooves focused on the wet clutch\[1-3\], although the HVD has some similarity with wet clutch, more researches need to be done. Summarizing the previous work, the influence of effective area ratio on the shear torque was well studied. However, the present studies about grooves mostly concentrate on circumferential grooves\[2\], radial grooves\[3,4\] and the inclined radial grooves\[5-7\]. However, the double arc grooves are more common in the engineering. So the effect of the double arc grooves needs to be studied further.

2. PARAMETERIZED ANALYZING PLATFORM

The previous researches about the influence of the double arc grooves parameters on the output torque are not sufficient, because the simulations about groove parameters are ponderous and repetitious. Therefore, an iSIGHT platform utilizing the multidisciplinary optimization was built to analyze the influences of groove parameters including the width $d$, the number $n$, the eccentricity $l$, the inner diameter of eccentric $d_0$ and the depth of oil groove $h_g$ on the torque characteristics. The platform integrated CAD and CAE by geometric module and finite module. And the effects of parameters on the output torque of HVD were analyzed in the parameter analyzing module. By utilizing the trail files, the script files, the batch commands and the parameterized command stream, the simulations were automatically driven. The platform is shown as Fig.1.
The trail files were used in the parametric modeling, and it reliably guaranteed the transition of oil groove parameters. Parametric modeling of geometric model is completed by Pro/E trail file, and the seamless connection between Pro/E and ANASYS is achieved by interface module. When the geometric parameters were altered, the family and its attachment were unchanged. By refreshing the geometric model, the new one was saved and exported as “.tin files”.

Computational domain, boundary conditions and initial conditions were set, then the parallel computing was running by calling the ANASYS solver, after the results were analyzed, the result files were exported.

The effects of double arc oil groove parameters on the output torque of HVD were studied. The oil groove parameters analysis platform was built by iSIGHT software, integrating oil groove parameterized design, flow field numerical simulation, design of experiment.

3. SIMULATION AND RESULTS

The oil film between the FP and SD was analyzed, and a schematic diagram of oil film is shown as FIG 2. One period of the film disk is calculated, since the film is N-fold periods. And the corresponding parameters are shown as Table.1.

<table>
<thead>
<tr>
<th>Items</th>
<th>Parameters</th>
<th>Items</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>inner radius of the FP R₀/mm</td>
<td>512</td>
<td>density of oil ρ[Kg·m⁻³]</td>
<td>886</td>
</tr>
<tr>
<td>outer radius of the FP R₁/mm</td>
<td>664</td>
<td>specific heat capacity of oil c[JKg⁻¹K⁻¹]</td>
<td>2093.5</td>
</tr>
<tr>
<td>thickness of oil film in the non-grooved area h₀/mm</td>
<td>0.3</td>
<td>dynamic viscos of oil η(Pa·s)</td>
<td>0.1848</td>
</tr>
<tr>
<td>thickness of oil film in the grooved area h₁/mm</td>
<td>-</td>
<td>input flow rate of HVD q/(L·min⁻¹)</td>
<td>0.0013376</td>
</tr>
</tbody>
</table>

FIG 1. iSIGHT parameters analyzing platform

The trail files were used in the parametric modeling, and it reliably guaranteed the transition of oil groove parameters. Parametric modeling of geometric model is completed by Pro/E trail file, and the seamless connection between Pro/E and ANASYS is achieved by interface module. When the geometric parameters were altered, the family and its attachment were unchanged. By refreshing the geometric model, the new one was saved and exported as “.tin files”.

Computational domain, boundary conditions and initial conditions were set, then the parallel computing was running by calling the ANASYS solver, after the results were analyzed, the result files were exported.

The effects of double arc oil groove parameters on the output torque of HVD were studied. The oil groove parameters analysis platform was built by iSIGHT software, integrating oil groove parameterized design, flow field numerical simulation, design of experiment.

3. SIMULATION AND RESULTS

The oil film between the FP and SD was analyzed, and a schematic diagram of oil film is shown as FIG 2. One period of the film disk is calculated, since the film is N-fold periods. And the corresponding parameters are shown as Table.1.

<table>
<thead>
<tr>
<th>Items</th>
<th>Parameters</th>
<th>Items</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>inner radius of the FP R₀/mm</td>
<td>512</td>
<td>density of oil ρ[Kg·m⁻³]</td>
<td>886</td>
</tr>
<tr>
<td>outer radius of the FP R₁/mm</td>
<td>664</td>
<td>specific heat capacity of oil c[JKg⁻¹K⁻¹]</td>
<td>2093.5</td>
</tr>
<tr>
<td>thickness of oil film in the non-grooved area h₀/mm</td>
<td>0.3</td>
<td>dynamic viscos of oil η(Pa·s)</td>
<td>0.1848</td>
</tr>
<tr>
<td>thickness of oil film in the grooved area h₁/mm</td>
<td>-</td>
<td>input flow rate of HVD q/(L·min⁻¹)</td>
<td>0.0013376</td>
</tr>
</tbody>
</table>

FIG 2. Schematic sketch of oil film
The rotational speed of SD is constant and relatively small. The input flow rate is constant and more than the flow rate when the aeration occurs in different simulations. Therefore, the aeration doesn't happen, and the friction pairs work in the full oil-film stage. The effective area ratio changes with variation of the double arc groove parameters including the width of oil grooves $d$, the number of oil grooves $n$, the eccentricity $l$, the inner diameter of eccentric $d_0$ and the depth of oil grooves $h_g$.

### 3.1 Number of oil grooves

When the width of oil groove is 1.38mm, the eccentricity is 388mm, and the inner diameter of eccentric is 809mm and the depth of oil groove is 0.5mm. The relation between the shear torque of oil film and the number of oil grooves and the relation between the effective area ratio and the number of oil grooves are shown as FIG 3. The figure shows the increase of the number of oil groove causes the decrease of the effective area ratio. With the increase of the number of oil grooves, the shear torque of oil film shear torque declines. The trend of the effective area ratio, shear torque of oil film are same, when the width of oil groove varies.

### 3.1.1 Width of oil groove

When the number of oil groove is 360, the eccentricity is 388mm, and the inner diameter of eccentric is 809mm and the depth of oil groove is 0.5mm. The relation between the shear torque of oil film and the width of oil groove and the relation between the effective area ratio and the width of oil groove are shown as FIG 4.
The figure shows the increase of the width of oil groove causes the decrease of the effective area ratio. With the increase of the width of oil groove, the shear torque of oil film shear torque declines. The trend of the effective area ratio, shear torque of oil film are same, when the width of oil grooves varies.

3.1.3 Eccentricity

When the number of oil grooves is 360, the width of oil groove is 1.38mm, and the eccentricity is 388mm and the depth of oil groove is 0.5mm. The relation between the shear torque of oil film and the width of oil groove and the relation between the effective area ratio and the eccentricity are shown as FIG 5. When the eccentricity increases from 385mm to 391mm, the effective area ratio increases with the eccentricity, whereas the shear torque of oil film decreases gradually. This phenomenon happens since the eccentricity has great effect on not only on the effective area ratio but also on the velocity distribution. When the eccentricity increase from 391mm to 403mm, the effective area ratio curve and shear torque of oil film curve are both increase first then decrease. It is because of complex effects of eccentricity on the effective area ratio. The eccentricity too large or too small both can cause the arrangement of grooves near the inner diameter of FP too narrow, even there will be no grooves any more. Therefore, the certain value of the eccentricity exists, and let the effective area ratio be maximum. The trend of the effective area ratio, shear torque of oil film are almost same, when the eccentricity varies.

![FIG 5. Relation between effective area ratio, shear torque and the eccentricity](image)

3.1.4 Inner diameter of eccentric

When the number of oil grooves is 360, the width of oil groove is 1.38mm, and the eccentricity is 388mm, and the depth of oil groove is 0.5mm. The relation between the shear torque of oil film and the inner diameter of oil grooves and the relation between the effective area ratio and the inner diameter of grooves are shown as FIG 6. The figure shows the increase of the inner diameter of oil grooves causes the decrease of the effective area ratio. With the increase of the inner diameter of oil groove, the shear torque of oil film declines. The trend of the effective area ratio, shear torque of oil film are same, when he inner diameter of oil grooves varies.

![FIG 6. Relation between effective area ratio, shear torque and the inner diameter of eccentric](image)
3.1.5 Depth of oil groove

When the number of oil grooves is 360, the width of oil groove is 1.38mm, and the eccentricity is 388mm, and the inner diameter of eccentric is 809mm. The relation between the shear torque of oil film and oil groove depth and the relation between the effective area ratio and oil groove depth are shown as FIG 7. The figure shows during the increase of oil groove depth, the effective area ratio keeps constant. While the depth of oil grooves increasing, the shear torque of oil film grows first then declines. It is because when the effective area keeps constant, the oil groove depth is another direct reason causing the change of shear torque of oil film. When the depth is less than 0.57mm, the increasing of it causes the change of velocity distribution, and then the shear torque of oil film grows. When the oil groove depth grows from 0.57mm to 0.63mm, the effect of oil groove depth on the shearing torque causing by bottom surface of the oil groove is more significant than the velocity distribution, so the shearing torque decreases.

![FIG 7. Relation between effective area ratio, shear torque and depth of oil groove](image)

3.2 The effects of oil groove parameters on shocking torque

When the FP keeps stationary and the SD rotates at the angular speed \( \omega \), the speed stream lines between the FP and SD are shown as FIG 8 and FIG 9. The figures show that the highest velocity of oil appears along the grooves whose direction is with the rotation direction of SD. Whereas oil moving along the grooves whose direction is against the rotation direction of SD gets its lowest value. The oil movement is different in the vertical direction. When \( 0 < z < h_g \), the oil is sheared by the drive of SD. The circumferential velocity is more than the radial velocity, the shear torque generated by oil film on the non-grooved area accounts for a large proportion in the output torque of HVD. When \( h_g \leq z \leq h_g + h_h \), the oil moves along the grooves by the influence of the pressure difference between inner and the outer radius of the FP, the flow inertia and the centrifugal force As a result, the radial velocity is more than the circumferential velocity. And the significant pressure difference exists in conjunctional zones between the grooved area and the non-grooved area, since the FP is stationary and the oil stroke the sidewalls of grooves. The groove sidewalls are subjected to the force caused by the pressure difference, and the grooved area of FP is subjected to the quite smaller shear stress by the oil movement along the grooves. The torque is generated by the two different kinds of mechanism.

![FIG 8. Stream lines of single period of oil film](image)
3.2.1 Number of oil grooves

When the width of oil groove is 1.38mm, the eccentricity is 388mm, and the inner diameter of eccentric is 809mm and the depth of oil groove is 0.5mm. The relation between the shocking torque of oil film and the number of oil grooves is shown as in FIG 10. With the increase of the number of oil grooves, the shocking torque of oil film shear torque grows. When the input of oil is constant and the number of oil groove increase, so the effect of oil shocking on the sidewalls of oil grooves enhanced.

3.2.2 Width of oil groove

When the number of oil groove is 360, the eccentricity is 388mm, and the inner diameter of eccentric is 809mm and the depth of oil groove is 0.5mm. The relation between the shocking torque and the width of oil groove is shown as FIG 11. The figure shows the shocking torque increases with the increase of the width of oil groove. It is because the velocity distribution changes with the oil grooves arrangement, and the effect of oil shocking enhances.
3.2.3 Eccentricity

When the number of oil groove is 360, the width of oil groove is 1.38mm, and the inner diameter of eccentric is 809mm and the depth of oil groove is 0.5mm. The relation between the shocking torque of oil film and the width of oil groove is shown as FIG 12. When the eccentricity increases, the shocking torque has an increase trend but gets fluctuations when the eccentricity is between 391mm to 401mm. It is because the eccentricity has complex effect on velocity distribution, therefore the shocking torque varies.

![FIG 12. Relation between eccentricity and the shocking torque](image)

3.2.4 Inner diameter of eccentric

When the number of oil grooves is 360, the width of oil groove is 1.38mm, and the eccentricity is 388mm, and the depth of oil groove is 0.5mm. The relation between the shocking torque of oil film and the inner diameter of oil grooves is shown as Fig 13. With the increase of the inner diameter of oil groove, the shear torque of oil film has a decreasing trend. With the variance of velocity distribution of the oil film, the shocking torque changes as the figure shows.

![FIG 13. Relation between inner diameter of eccentric and the shocking torque](image)

3.2.5 Depth of oil groove depth

When the number of oil grooves is 360, the width of oil groove is 1.38mm, and the eccentricity is 388mm and the inner diameter of eccentric is 809mm. The relation between the shocking torque of oil film and the oil groove depth is shown as FIG 14. With the increase of the oil groove depth, the shocking torque of oil film decreased. The velocity distribution of the oil film has a significant effect on the shocking torque.
3.3 The effects of oil groove parameters on output torque

The output torque of HVD in the oil film shearing stage consists of the shear torque and the shocking torque. Affected by dynamic pressure of flow field on the sidewall of oil grooves, the shocking torque produced by side walls of oil grooves is accounts for nearly 1/3 of the output torque of HVD. And the shearing torque accounts for nearly 2/3. Influences of width of oil grooves, the number of oil grooves, the eccentricity, the inner diameter of eccentric and the depth of oil grooves on the output torque were analyzed.

The geometric module and finite element module were packaged by simcode in iSIGHT. The effects of double arc oil groove parameters were analyzed by the design of equipment. The DOE was carried down by orthogonal arrays. The sensitivity of each parameter, main effects, and interaction effect were analyzed.

Pareto graph illustrates the effect of each oil groove parameter on the output torque. The blue bars represent the positive effects, they express that with the parameters grows, the response increases. While the red represent the negative effect, they express that with the parameters grows, the response decreases. As the FIG 15 shows, the positive response of the eccentricity, 14.37%, is most obvious than others. The interaction between the diameter of inner eccentric and eccentricity, accounting for 8.81%, has a quite strong positive effect on output torque. However, the interaction between the diameter of inner eccentric and the number of oil grooves has the most significantly negative effect, -12.69%.

In the FIG 16, the gradient of oil groove parameters is consistent with the effect in the pareto graph. The bigger the gradient is, the more obvious the effect of parameter on the response is. As figure shows, effect of the eccentricity and the depth are linear. With the effect of quadratic term, the effects of the diameter of inner eccentric, the number of oil grooves and the depth of oil groove on the output torque are parabolic.
The output torque consists of shear torque and the shocking torque. The shear torque of oil film increases with the width of oil grooves, the number of grooves and the diameter of the inner eccentric, and decreases with the growth of eccentricity. When oil groove depth grows, the shear torque gets fluctuated. The shocking torque increases with the width of oil grooves, the number of grooves and the eccentricity, and decreases with diameter of the inner eccentric and oil groove depth.

The eccentricity and diameter of the inner eccentric are sensitive parameters to the output torque.

The platform is integrated with CAD, CFX and DOE by utilizing trail files, script files, batch commands and parameterized command flow to set up parameters automatically and improve simulation efficiency. Effects of oil groove parameters on the shear torque are analyzed accurately, and a new method and theoretical foundation is provided for design of complex oil grooves of friction pairs of HVD.

ACKNOWLEDGMENTS

The authors would like to express their gratitude to colleagues at Shanxi Key Laboratory of Fully Mechanized Coal Ming Equipment for their contributions to this work. Thanks also to the Shanxi Coal-based Key Scientific & Technological Projects(MJ2014-06).

REFERENCES

**Oral Presentation | Water hydraulics**

[2C07] PERFORMANCE ANALYSIS OF LARGE FLOW SAFETY VALVE FOR POWERED SUPPORT  
*YongChang Guo\(^1,2\), ZiSheng Lian\(^1,2\), HongBing Yuan\(^1,2\), YaoYao Liao\(^1,2\) (1. College of Mechanical Engineering, Taiyuan University of Technology, 2. Shanxi Key Laboratory of Fully Mechanized Coal Mining Equipment)  
1:40 PM - 1:56 PM

[2C08] EXPERIMENTAL RESULT FOR ENERGY-SAVING TECHNOLOGY IN WATER HYDRAULIC MOTOR SYSTEM  
*Pha N. Pham\(^1\), Kazuhisa Ito\(^2\), Ryo Yagisawa\(^2\), Shigeru Ikeo\(^3\) (1. National Institute of Patent and Technology Exploitation, 2. Shibaura Institute of Technology, 3. Sophia University)  
1:56 PM - 2:12 PM

[2C09] DESIGN AND EXPERIMENTAL RESULTS OF THE WATER HYDRAULIC DRIVE SYSTEM FOR NEUTRON BEAM SHUTTER PROTOTYPE AT CSNS  
*Lixin Song\(^1\), Bing Xu\(^1\), Junhui Zhang\(^1\) (1. State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University)  
2:12 PM - 2:28 PM

[2C10] STUDY ON ACTIVE CHARGE ACCUMULATOR FOR AQUA DRIVE SYSTEM (Effective Parameters on Boosting Performance)  
*Satoru Takahashi\(^1\), Kazuhisa Maeda\(^2\), Futoshi Yoshida\(^3\), Shoichiro Iio\(^1\), Ato Kitagawa\(^4\) (1. Shinshu University, 2. TOYOTA AUTO BODY, 3. KYB Corporation, 4. Tokyo Institute of Technology)  
2:28 PM - 2:44 PM

[2C11] A NEW TYPE OF SPHERICAL MICRO PUMP  
*Hao Pang\(^1\), Yinshui Liu\(^1\), Luyi Wang\(^2\), Zhuang Niu\(^2\) (1. Huazhong University of Science and Technology, 2. Hust-Wuxi Research Institute)  
2:44 PM - 3:00 PM
PERFORMANCE ANALYSIS OF LARGE FLOW SAFETY VALVE FOR POWERED SUPPORT

Yongchang Guo***, Zisheng Lian***, Hongbing Yuan***, Yaoyao Liao***

* College of Mechanical Engineering, Taiyuan University of Technology
No.79, West Yingze Avenue, Wanbailin District, Taiyuan, Shanxi, 030024, P. R. China
(E-mail: gyc37@126.com)
** Shanxi Key Laboratory of Fully Mechanized Coal Mining Equipment
No.79, West Yingze Avenue, Wanbailin District, Taiyuan, Shanxi, 030024, P. R. China

Abstract. Safety valve is an important protection component of the powered support. Based on the voussoir-beam structure mechanical model of coal mining face overburden strata; The load on powered support is the sum of static load and dynamic load; The combined loading test bed is set up (loading of accumulator simulates static load, loading of gas explosion simulates dynamic load). Experimental conditions are as follows: the pressure of LPG-air mixtures is 1.6 MPa, the set pressure of the test safety valve is 45 MPa, the setting filling fluid pressure of accumulator is 31.5 MPa; Experimental results showed that: the pressure overshoot of the test safety valve is 4.9882 MPa, is about 11% of the set pressure, the pressure stabilization time of the test safety valve is 0.017 s. The flow field of safety valve is investigated by ANSYS FLUENT, pictures of safety valve overflow are obtained by the high speed camera, verifying the safety valve flow field simulation results.

Keywords: CFD, Flow field analysis, Safety valve, Powered support, Impact test

1. INTRODUCTION

Powered support is the safety equipment of fully mechanized coal mining face, safety valve installed in the leg is an important component of powered support, it is used to limit the actual working resistance of powered support and make the actual working resistance not exceed the allowable value, position of safety valve on powered support is shown in Fig 1. In the process of coal mining, when the face roof is broken and impacts powered support, the leg of powered support undertakes enormous load consisting of static load and dynamic load, when the work resistance of leg exceeds the set pressure of safety valve, the valve will open, a large part of impact energy delivered to powered support from roof will be dissipated, so it is necessary to know the work performance of the safety valve, especially dynamic characteristics. Andrzej Pytlik presents process characteristics of hydraulic legs, a powered roof support and an individual roof support that are equipped with pressure relief valves and additional safety valves protecting the legs against dynamic loads caused by mining tremors [1]. Operation of hydraulic prop and safety valve under dynamic load is analyzed by Yu.M.Lekontsev [2] and V. I. Klishin [3], the principle of experimental devices that they used is free fall hammer impact column. Junliang Chang establish the impact test-bed of high-flow water medium relief valve based on rapid loading of accumulator and detect the dynamic performance of high-flow water medium safety valve [4].

FIGURE 1. Powered support
With the development of computer technology, computational fluid dynamics (CFD) is increasingly being used to hydraulic valves for product development experiments. The flow process inside a pressure regulating valve is investigated by Himadri Chattopadhyay using a CFD approach [5].

This paper presents a safety valve impact test rig based on combined loading of accumulator and gas explosion, through the impact test of the test safety valve, pressure time curves are obtained, and dynamic characteristics of the safety valve are analyzed. Overflow of safety valve are studied by high speed camera and CFD approach. A typical high flow safety valve for powered support is shown in Fig 2.

![Valve spool is in original position](image1)

(a) Valve spool is in original position

![Valve spool is fully opened](image2)

(b) Valve spool is fully opened

**FIGURE 2.** 1000 L/min Safety valve

**2. EXPERIMENTAL SET-UP**

![Structure of voussoir beam](image3)

**FIGURE 3.** Structure of voussoir beam

Based on the voussoir-beam structure mechanical model of coal mining face overburden strata [6,7], the roof strata load in hydraulic support was classified by Guofa wang [8], the load on the support is the sum of static load and dynamic load. Structure of voussoir beam in coal mining face is shown in Fig 3.

The safety valve combined loading impact test bed is set up (loading of accumulator simulates static load, loading of gas explosion simulates dynamic load). The sketch of the safety valve test rig based on loading of gas explosion and accumulator is shown in Fig 4. The experimental device consists of two parts of the explosion cylinder (6) that located in the upper part and the hydraulic cylinder (7), in which A chamber is the explosion chamber, compressed air is supplied from air compressor (4) through pneumatic stop valve (3), combustible gas is supplied from gas tank (5) through pneumatic stop valve (3), according to explosion characteristics of liquefied petroleum gas (LPG) [9], and LPG is also easy to get, so this study uses LPG as a combustible gas, the gas in the chamber A is ignited by the ignition device (2), B chamber is directly connected with the accumulator group (16), in order to ensure the progress of experiment, the accumulator group that is composed of ten 100 L accumulators is selected for supply the emulsion to the B chamber in this paper, C chamber is the working chamber, connected with the test safety valve. A, B, C chamber are connected with pressure sensor, the pressure changes in A, B, C chamber are obtained during the experiment. The high speed camera is used to get the overflow photos of the test safety valve.

Preparations before the start of experiment: A chamber is washed by compressed air from air compressor (4) approximately 5 minutes, the accumulator group (16) is filled with emulsion from an emulsion pump (10) to 31.5 MPa, the piston of hydraulic cylinder (7) is moved to the top, the pressure of C chamber is 31.5 MPa. Then all valves are closed.

FIGURE 4. Sketch of the safety valve test rig

The working flow of experiment is shown in Fig 5. LPG-air mixtures were prepared by partial pressure method, wait 5 minutes before the A chamber is ignited.

3. RESULTS AND DISCUSSIONS

The pressure of LPG-air mixtures before explosion is 1.6 MPa, the LPG concentration is 6.4%. Pressure time curves throughout experiment are shown in Fig 6, the test safety valve is connected with C chamber, so the pressure of C chamber is can be considered the pressure of the test safety valve (the set pressure of the test safety valve is 45 MPa).
3.1 Dynamic Characteristics

3.1.1 Pressure Overshoot

The pressure overshoot is the difference between the peak pressure and the set pressure of the test safety valve, the peak pressure of the test safety valve is 49.9882 MPa, see Fig 6, so the pressure overshoot of the test safety valve is 4.9882 MPa, the pressure overshoot is about 11% of the set pressure.

3.1.2 Pressure Stabilization Time

The pressure stabilization time is the time that the safety valve first reaches the set pressure change to the pressure stabilized. the pressure stabilization time is 0.017 s, see Fig 6, (the first time of the safety valve reaches the set pressure:11.318581 s; the time of the pressure stabilized:11.335671 s).

3.2 Flow field analysis

![3D velocity contour of the safety valve](image1)
![Picture of safety valve overflow](image2)

**FIGURE 7.** 3D velocity contour of the safety valve (the displacement of valve spool is 5.35 mm)

**FIGURE 8.** 2D velocity contour and picture of the safety valve overflow (the displacement of valve spool is 5.35 mm)
According to Fig 6, when t=11.345902 s, the pressure of the safety valve is 45.4 MPa, corresponding the displacement of safety valve spool is 5.35 mm. At this time, the flow field of safety valve is investigated by computational fluid dynamics (CFD) software ANSYS FIUENT, the distribution of velocity at this position of the safety valve flow channel is obtained. Simultaneously, the picture of safety valve overflow is obtained by the high speed camera, to verify the safety valve flow field simulation result. The 3D velocity contour of safety valve flow field (the displacement of valve spool is 5.35 mm) is shown in Fig 7. The 2D velocity contour of safety valve flow field and the picture of safety valve overflow is taken by high speed camera are shown in Fig 8.

4. CONCLUSIONS

(1) Considering the load on powered support is the sum of static load and dynamic load, a safety valve test rig based on combined loading of gas explosion and accumulator is set up (loading of accumulator simulates static load, loading of gas explosion simulates dynamic load); The experimental and theoretical results verified that: The safety valve characteristics can be tested by this rig.

(2) The pressure of LPG-air mixtures before explosion is 1.6 MPa, the set pressure of the test safety valve is 45 MPa, the pressure overshoot of the safety valve is 4.9882 MPa, is about 11% of the set pressure; The pressure stabilization time of the safety valve is 0.017 s.

ACKNOWLEDGMENTS

This work was supported by Shanxi Province Coal-based Key Scientific and Technological Project (No. MJ2014-07).

REFERENCES

EXPERIMENTAL RESULT FOR ENERGY-SAVING TECHNOLOGY IN WATER HYDRAULIC MOTOR SYSTEM

Pha N. PHAM*, Kazuhisa ITO**, Ryo YAGISAWA***, Shigeru IKEO****

*National Institute of Patent and Technology Exploitation, Ministry of Science and Technology
39 Tran Hung Dao Str., Hanoi, Vietnam
(E-mail: phamngocpha@gmail.com)
**Department of Machinery and Control Systems, Shibaura Institute of Technology
**Graduate School of Mechanical Engineering, Shibaura Institute of Technology
307 Fukasaku, Minuma-ku, Saitama, 337-8570 Japan
***Professor Emeritus, Sophia University
7-1 Kioicho, Chiyoda-ku, Tokyo, 102-8554 Japan

Abstract. Nowadays, water hydraulics still faces with some main difficulties for widening application. First, the initial cost of water hydraulic components is normally more expensive than oil hydraulic ones; this property can be compensated by using very cheap pressure medium (water), much reducing insurance and disposal fees if the utilized period is long enough. The second challenge is that the control performances of water hydraulic systems are effected by nonlinearity; strong friction and considerable leakage than oil hydraulics. This can be overcome by using advanced control methods or by improving the performance of water hydraulic devices. Very important challenge for not only water hydraulics but also oil hydraulics is low energy efficiency of hydraulic systems, from 6% - 40% for oil hydraulics depending on applications and even lower in water hydraulics. In this study, a novel water hydraulic motor system is introduced. The main purpose is to reduce supply pressure to be nearly equal to load pressure. This leads to the energy loss can be reduced drastically. The experimental results showed that proposed system can raise the energy efficiency 41.2% and the control response was acceptable.

Keywords: Water hydraulics, Energy-saving, Control performance

INTRODUCTION

Recently, the application of water hydraulics has received considerable attention because of its inherent merits such as environmental friendliness, hygiene, washability, high safety against fire hazard, lower running cost, and availability of tap water. Owing to the outstanding advantages, water hydraulics, which uses pure tap water as the pressure medium has been applied in many fields, especially food [1,2], beverages, semiconductors, medicine processing, steel and glass production, ocean exploration, underwater robotics [3], nuclear power generation [4], underwater gait-training orthosis [5], wave/wind power generation systems, mining machinery, ocean development machinery (e.g., underwater drilling machinery) [6], etc.

On the other hand, energy efficiencies of fluid power systems are low. It varies between 6% – 40% depending on the application; the average efficiency is only around 21% [7,8] for oil hydraulics. The energy efficiencies of aqua drive systems are much lower than that of oil hydraulic ones owing to the larger friction and higher amount of leakage in water hydraulic devices, which in turn are due to the low viscosity of pure tap water. In water hydraulic systems, only fixed displacement pumps have been used until now because a water hydraulic variable displacement pump requires advanced technology, which will increase the manufacturing cost. As a result, load-sensing technology, which requires a variable displacement pump, cannot be applied in water hydraulic systems. The energy losses due to the surplus pressure and flow rate degrade the energy efficiency of hydraulic systems.

This study is aimed at designing a novel water hydraulic transmission technique to improve the efficiency of motor systems. The proposed system uses two two-way, two-position flow control valves for controlling the pressure at the input port of the hydraulic motor by releasing small amounts of surplus flow rate to a reservoir through these valves. The surplus pressure in such systems can be minimized and made equal to the hose pressure drop. The paper [9] shows the simulation results of such novel system. This paper discusses on the experimental results. The experimental results showed that the proposed transmission system can improve the energy efficiency of conventional system to more than 41% and the control performance is acceptable, it is not much different from the response of the conventional water hydraulic motor system.
FIGURE 1. Schematic diagram of conventional water hydraulic motor system.

FIGURE 2. Schematic diagram of proposed water hydraulic motor system.

SYSTEM STRUCTURES

Figures 1 and 2 show the schematic diagrams of the conventional water hydraulic servo motor system and proposed water hydraulic motor system, respectively. Conventional servo motor system consists of the main elements as follows. A fixed displacement pump connected to an electric motor. High pressure port of the pump is connected to a relief valve for setting the working value of the supply pressure. A servo valve is connected to a hydraulic motor for controlling the rotational velocity of flywheel to track a given reference. Proposed motor system includes following elements. A fixed displacement pump connected to an electric motor and the high-pressure port of the pump is connected to a relief valve. These devices used in this system are same as in the conventional system. However, the relief valve in the proposed system is only used for safety purpose. Two solenoid On/Off valves are assembled to two paths from the hydraulic pump to cylinder for changing the working direction of the flywheel. Two fluid control valves $SV_1$ and $SV_2$ are for controlling the rotational velocity of the flywheel by controlling the working pressure $P_{m1}$ and $P_{m2}$.

Based on the reference phases as shown in Fig. 4, the operation of the proposed system can be explained as follows. In the first cycle corresponding to the clockwise direction of the flywheel, the valve $VS_2$ is closed.
the acceleration and constant velocity phases, the valve On/Off₁ is opened, the flow control valve SV₂ is opened fully and the control valve SV₁ is used for controlling the velocity of the flywheel to track the reference. In the deceleration phase, the valve SV₁ and the valve On/Off₁ are closed, the valve SV₂ is used for controlling the velocity of the flywheel. Meanwhile, the suction fluid supplied to the motor flows through the check valve CV₁. In the second cycle, corresponding to the anticlockwise direction of the flywheel, the operation of the system can be described in the same manner as above, with the only difference that the valve On/Off₂ is opened in the acceleration and constant phases, and the valve On/Off₁ is closed and the functions of the valves SV₁ and SV₂ are reversed simultaneously [9].

CONTROL SYSTEM DESIGN

The proposed system (shown in Fig. 2) is designed to improve energy efficiency. The hydraulic transmission consists of two valve On/Off₁ and On/Off₂ used for changing the rotational direction (clockwise or anticlockwise cycles abbreviated as CW and ACW, respectively) of the flywheel FW. Two flow control valves SV₁ and SV₂ are used for controlling the flywheel to track the reference. The control algorithm for these flow control valves are modified depending on whether the direction of rotation of the flywheel FW is CW or ACW and whether the system is in the acceleration, constant, deceleration, or idle phase.

FIGURE 3. Schematic diagram of controller for proposed system.

Fig. 3 shows the schematics diagram of the controller for the proposed system. There are two independent control circuits for the On/Off valves and flow control valves. The On/Off valves mode switch and SV₁ and SV₂ mode switch are based on the reference signal for choosing the working states for each valve. The detail is shown in Fig. 4.

The clockwise motion of the flywheel FW is considered first. The valve On/Off₂ was always closed. During the acceleration and constant phases, the valve On/Off₁ was opened because the hydraulic motor M required high pressure fluid in the chamber A. The velocity of the hydraulic motor M in these phases was controlled by the flow control valve SV₁. Note that in Fig. 4, PID represents the proportional-integral-derivative controller.
During these phases, the fluid control valve $S_V_2$ was opened fully for reducing the meter-out pressure to nearly the atmospheric pressure. During the deceleration and idle phases, the valve On/Off$^1$ was closed because the hydraulic motor $M$ did not require the supply of energy. The fluid control valve $S_V_1$ was closed as well. The velocity of the flywheel was controlled via the meter-out orifice – the fluid control valve $S_V_2$. The hydraulic motor $M$ sucked water from the reservoir via the check valve $C_V_1$. During the anticlockwise rotation of the flywheel, the control logic for these valves were the same as that during clockwise rotation with the only difference that the functions of the valves On/Off$^1$ and On/Off$^2$ and flow control valves $S_V_1$ and $S_V_2$ were reversed simultaneously. The detail can be observed in Fig. 4. The controller for conventional system is PID controller and the gains were tuned by trial and error method. The gains of the PID controllers for controlling two flow control valves $S_V_1$ and $S_V_2$ were also tuned by trial and error method.

**EXPERIMENTAL RESULT**

The simulation results were shown in the paper [9]. This study analyzes and discusses on the experimental results ie. velocity response, energy performance. The main contribution of this study is to reduce the energy loss and raise the energy efficiency of the water hydraulic transmission. In this study, the encoder, which measures the velocity of the flywheel, can measure one rotational direction only; hence, only one cycle corresponding to one direction will be introduced. The full working cycles with clockwise and anticlockwise directions will be carried out in near future.

**Velocity response**

Figures 5 and 6 shows the experimental control responses and errors of conventional servo motor system and proposed system, respectively. It is easy to realize that in the constant phase, the control performances of both systems are almost the same. In the acceleration phase, the control performance of the proposed system shows its advantage due to faster response. In contrast, in the deceleration phase, the response of the conventional system is better because it used the supply pressure in this phase for braking the velocity of the flywheel. That means the control response quality in the proposed system is not much different from the conventional water hydraulic motor system. Thus, the proposed system can be used instead of the conventional one in application.
In the experimental rig, the energy consumption was measured directly on the electricity supply to the electric motor. Figure 7 shows energy consumption in conventional and proposed systems. The conventional system consumed 38.8Wh for one cycle while the proposed system reduced nearly half with the supply energy of 22.8Wh. The main difference is come from constant and deceleration phases while in the acceleration phase, the energy consumption of both systems are almost the same, only slightly reduce in proposed system. In deceleration phase, the proposed system did not need to supply energy, as a result, in this phase there is no energy supply to the proposed system. In contrast, conventional system still used the energy for braking the flywheel. In the constant phase, the proposed system used meter-in meter-out method. This method can minimize the energy loss via high pressure in the meter out. By this way, the energy consumption in the constant phase of the proposed system downs drastically.

Fig. 8 shows the supply pressures in conventional and proposed system, pressure in two chambers of the hydraulic motor in conventional system. It is easy to realize that the supply pressure in conventional system is much bigger than proposed one during constant and deceleration phases. The reason is mainly because of the meter-out pressure in the constant phase is too high and the braking pressure in the deceleration phase. In the acceleration phase, the supply pressure in the conventional system is only slightly higher than proposed system.
CONCLUSIONS

This study introduced the experimental results of a novel water hydraulic motor system in comparison with a conventional system. This paper is the succeeding results of the paper [9] which shown the simulation results of the proposed system. The main contribution of the proposed system is for raising energy efficiency. The experimental result showed that the proposed system can reduce energy consumption by 41.2%. The main improvement of energy consumption in the proposed system came from constant and deceleration phases while both systems consumed almost the same amount of energy during acceleration phase. The control performance of the proposed system was acceptable with slightly faster response during acceleration phase, slightly slower response during deceleration phase, and nearly similar during constant phase.

ACKNOWLEDGMENTS

The authors gratefully acknowledge the support of the Sumitomo Foundation.

REFERENCES

Design and experimental results of the water hydraulic drive system for neutron beam shutter prototype at CSNS

Lixin Song, Bing Xu*, Junhui Zhang

State Key laboratory of Fluid Power & Mechatronic Systems, Hangzhou 310027, People’s Republic of China
(E-mail: bxu@zju.edu.cn)

Abstract. The neutron beam lines shutter system which directly affects the effective availability of the neutron beam is a key component of China Spallation Neutron Source (CSNS). The marked feature of the neutron beam lines shutter system is to realize the high repetitive positioning accuracy requirement under heavy load condition, so the stability of low speed is an essential requirement for CSNS neutron beam lines shutter system. Considering the load impacts induced by the motion stop, two different speed control loops were proposed. Simulation models were established to quantify the cushion performance and then the simulation models were verified by experimental tests. Taking advantage of water hydraulic system, the neutron beam line shutter prototype drive system was successfully performed and the achieved terminal speed was less than 3mm/s. This type of water hydraulic drive system will be used in the CSNS neutron beam lines shutter system, which is under construction.

Keywords: Neutron beam line, Water hydraulic, Speed control loop, Heavy load.

1. INTRODUCTION

The China Spallation Neutron Source (CSNS) of which the total investment is RMB 2.2 billion yuan is the biggest major national science and technology infrastructure project in China during "the eleventh five-year plan" [1]. The neutron beam line shutter system is one important subsystem which consists of 20 shutter switches as shown in Fig 1. The structure of each shutter switch is similar but the movement of each shutter switch is independent and stochastic. The neutron guide which is in the shutter gate is the key equipment that controls the switch of the neutron beam line.

The schematic diagram of the water hydraulic drive system for one neutron beam lines switch is shown in Fig 2. By lifting the shutter gate whose weight is about 17 tons, the neutron beam lines switch drive system realizes the up and down of the neutron guide in one minute which is corresponding to the off and on of the neutron beam lines.

There are three neutron beam lines shutter systems at present which can be classified into mechanical drive and water hydraulic drive[2-4]. The mechanical drive system mainly consists of gear motor, worm gearing and pull rod in J-PARC [5-6]. The positive and negative motion of the gear motor is corresponding to the up and down of the shutter gate. The self-locking characteristics of the worm gearing can stop the sudden fall of the shutter gate when it’s out of control. The water hydraulic drive system mainly consists of pump station, cylinder, hydraulic manifold, safety catcher in SNS [7-9]. The water hydraulic drive system whose power source can be within no neutron radiation is what we prefer to use in the CSNS. The water hydraulic drive system can close the neutron guide in case of power failure which is more reliable than the mechanical drive system.
In this paper, the R&D activity of the water hydraulic drive system for the CSNS neutron beam lines shutter prototype will be presented, including the physical design, the optimization of the speed control circuit and the performance test results of the prototype.

2. WATER HYDRAULIC DRIVE SYSTEM DESIGN

2.1 System Function

By means of one water hydraulic cylinder, the drive system realizes the up and down of the neutron beam line shutter gate. The cylinder has smooth motion and its speed can be controlled.

The water hydraulic drive system consists of cylinder control subsystem, pump station subsystem and electronic control subsystem. The technical performance is shown in Table 1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>System pressure</td>
<td>14MPa</td>
</tr>
<tr>
<td>Work load</td>
<td>17t</td>
</tr>
<tr>
<td>Medium temperature</td>
<td>&lt;50 ºC</td>
</tr>
<tr>
<td>Hydraulic medium</td>
<td>deionized water</td>
</tr>
<tr>
<td>Rated power</td>
<td>10KW</td>
</tr>
<tr>
<td>Switching time</td>
<td>&lt;1min</td>
</tr>
</tbody>
</table>

The schematic diagram of water hydraulic driving system is shown in Fig 3.

1) Cylinder control subsystem

The entrance throttle speed controls circuit which controls the cylinder’s up speed consists of fixed displacement pump, throttle valve and relief valve. The cylinder whose down speed is controlled by the proportional valve is lowered by the load. The on and off of the 2/2way solenoid directional control valve which is decided by the travel switch are corresponding to the up and down of the cylinder.

2) Pump station subsystem

The pump station subsystem contains two separate electric motor pump unit. The pump is started under unloading condition. The check valve, 2/2way solenoid directional control valve, relief valve, pressure transducer and pressure gauge are integrated on the modular valve block. The bypass cycle circuit is used to cooling, filtering and ultraviolet disinfection of the deionized water.

3) Electronic control subsystem

By using of PLC, the electronic control subsystem realizes the system’s motion control and flow control, the real-time monitoring of the system temperature pressure and so on, the fault alarming and intelligent protection of the system. The data acquisition system which consists of industrial computer can acquire, process and display the cylinder’s motion condition and other key parameters.
2.2 The Design of the Speed Control Circuit

The cylinder rod port which is connected to the water tank is always under lower pressure. The flow control of the head port is what we need to realize the speed control of the cylinder. System with proportional valve can realize the cylinder speed switch smoothly. But when considering the selection of proportional valve in water hydraulic system, we have fewer options because the proportional valve piston structure is a special curve structure [10]. This type proportional valve general applies to the work condition in which the working medium only flow from the "P" port to the "A" port, so the cylinder up circuit and down circuit must be designed respectively. The circuit 1 in Fig 4 is what we designed to meet the design requirements.

The proportional valve in water hydraulic system requires high filtering accuracy which is not less than 10μm, so the blocking of proportional valve is a big problem. The throttle valve with higher reliability is what we prefer to replace proportional valve. The schematic diagram is shown as circuit2 in Fig.4. Since the up speed and the down speed are required to be approximately equal, the cylinder up circuit and down circuit are also designed respectively.

2.3 The Simulation Analysis of the Speed Control Circuit

The analyses and comparison the performance of the two speed control circuits were done by simulation. The simulation model is shown in Fig 5.
The simulation results of two speed control circuits are shown in Fig 6 and Fig 7. We can see that the cylinder is quick to respond to the control command and the speed soon reach to the preset speed. Contrasting the two curves in the red boxes, it’s easy to find that the deceleration process from 10mm/s to 5mm/s in the proportional valve circuit is smoother. The direct speed switch from 10mm/s to 5mm/s in the throttle valve circuit may produce system pressure impact which can damage the components. Therefore we analyzed the pressure in cylinder head port. The results are shown in Fig 8 and Fig 9.

There are no pressure peak and pressure impact in the proportional valve circuit. The pressure peak in the throttle valve circuit is not big enough to cause a great impact on the performance of the system. The circuit 2 is what we finally used in the neutron beam shutter prototype.
3. EXPERIMENTS

In order to confirm the CSNS neutron beam shutter design, one neutron beam line shutter prototype which was used to do the verification was built. Since the cylinder speed switch happened at the end of opening process of the neutron beam line shutter, the test results during the opening process are analyzed in detail.
FIGURE 11. Simulations and experimental results of the cylinder motion process
In Fig 11, the test results of the cylinder upward and downward motion process anastomose the simulation results. The reason why there are numerical and change rate errors between them are as follows:
(1) The system damping and friction were not analyzed in detail during the system simulation modeling since the main purpose of the simulation modeling is to realize the system function and acquire the system parameters. So the cylinder speed simulation results show enormous rigidity which has deviation with the real system.
(2) To avoiding the noise data the average filter is what we used to improve the accuracy. Thus, there were errors between the drawing curve and the real cylinder motion.

FIGURE 12. the cylinder displacement diagram
In Fig 12 and Fig 13, the cylinder speed is decelerated obviously from 10mm/s to 3mm/s near the fiftieth seconds. The system vibration and impact at the terminal velocity can meet the system requirement.
As shown in Fig 14, the cylinder pressure is about 8.2 MPa when the cylinder moves upward and the cylinder pressure is about 7.6 MPa when the cylinder moves downward. When the speed switches directly near the fiftieth seconds, there is no pressure peak.

4. CONCLUSIONS

Design of the water hydraulic drive system used for the CSNS neutron beam lines shutter prototype is described and two different speed control circuits are discussed by means of simulation. The experiment of the system was performed with satisfactory results. The test shows a good agreement between the experimental and the simulation results. It also confirms that the water hydraulic drive system is suitable for the CSNS neutron beam lines shutter system design.

REFERENCES

Study on Active Charge Accumulator for Aqua Drive System (Effective Parameters on Boosting Performance)

Satoru Takahashi* Kazuhisa Maeda** Futoshi Yoshida*** Shoichiro Iio* Ato Kitagawa****

* Shinshu University  
4-17-1 Wakasato, Nagano-shi, Nagano, 380-8533, Japan  
(E-mail:14t6029h@shinshu-u.ac.jp)  
** TOYOTA AUTO BODY.  
100 Kanayama, Kariya-shi, Aichi, 440-8666, Japan  
*** KYB Corporation  
1-12-1 Azamizodai, Minami-ku, Sagamihara-shi, Kanagawa, 252-0328, Japan  
**** Tokyo Institute of technology  
2-12-1, Ookayama, Meguro-ku, Tokyo, 152-8550, Japan

Abstract. Active Charge Accumulator (ACA) is one of the key solution for simplification of system, reduction of cost and energy consumption in water hydraulic system especially for pressure conversion process. This study is focused on pressure boosting process by using ACA. The pressure reducing performance of ACA had been revealed by the authors, however, the boosting process is not fully understood and the boosting performance is not clarified yet. It is significance for widespread of water hydraulics in the future to understand and reveal the pressure boosting of ACA. The purpose of this study is to investigate and reveal important parameters; accumulator volume, valve discharge coefficient, initial gas volume in the accumulator. As a result, influence of each parameters for the boosting performance of ACA can be clarified.

Keywords: Water Hydraulics, Aqua Drive System, ACA, Pressure Boosting

INTRODUCTION

The driving technology that uses tap water as working fluid is a new technology—the fourth technology after conventional oil-hydraulic, pneumatic, and electric driven technologies. Systems consisting of water-hydraulic devices are very clean, as well as safe and secure; they are expected to be used in the areas of food machinery, medical care and pharmaceuticals, beverages, and semiconductor manufacturing [1]. The active charge accumulator (ACA) is a pressure converter device in a water-hydraulic system that is able to boost and decompress pressure without using a pump and pressure-regulating valve. The previous studies by the authors showed characteristic of pressure reducing process with ACA [2][3]. In these studies, to evaluate the performance characteristics of an ACA during the decompression process in water-hydraulic systems, we checked the characteristics based on experiments and analyzed them based on a mathematical model. The experimental and analysis results indicated that: ACA system that uses water as the working fluid can generate a load flow by reducing a supply pressure of $10 \times 10^6$ Pa to $2 \times 10^6$ Pa. The pressures in the pressure-conversion and low-pressure sections were almost the same between the analysis and experimental results. Both the experiment and analysis revealed that change cycle in the pressure-conversion section increases with decrease in load flow rate. Decompression process of ACA was revealed, however, there is not enough knowledge on boosting process of ACA. So, this study is aimed to grasp important parameters for boosting performance of ACA.

EXPERIMENTAL METHOD

Figure 1 shows a testing water hydraulic circuit for boosting. Boosting process with ACA is controlled by open/shut four valves, $V_{L1}$, $V_{L2}$, $V_{L3}$, $V_{L4}$, shown in the figure. Detail of the process is mentioned in the previous study by Maeda et al. [4]. The authors use a two-staged ACA for this experiment. In the ACA, there are three chambers. An accumulator was connected to Chamber c. Boosting pressure level is theoretically defined by the pressure-receiving area ratio between the top and the bottom of a piston. For releasing mode, the piston moves downward and pressurized water is issued from Chamber a and it is supplied to the load. Note that the valve $V_{L5}$ downstream $V_{L4}$ was used as a load. At the same time, the water in Chamber b flows back to a reservoir after through $V_{L4}$ and $V_{l}$. The water flow rate from the Chamber b was adjusted by changing the valve aperture of $V_{l}$. Valve discharge coefficient, $C_{th}$, is defined by Eq. (1) as an orifice flow.
where, $Q, \rho, \Delta P$ is flow rate, water density, differential pressure between the valve, respectively. In this study, influence of pressure setting condition in Chamber c on the boosting characteristics is investigated. So the following two methods were selected to set the pressure in Chamber c:
1) initial pressure adjusted by change of filling gas volume,
2) initial pressure adjusted by change of supplying water volume.
Character of supplying pressure to a load (actuator) is one of the most important factor for the performance of ACA. The variation of output pressure, $P_a$, with piston stroke, $x_p$, for above mentioned two methods was described in this paper.

**MODELING AND SIMULATION**

Figure 2 shows a model and parameters for simulation for the ACA. Software of MATLAB / Simulink was used for simulation. The piston position is the bottom of the ACA, $x_p$=0mm, as an initial condition.
Flow rate into/from Chambers a and b is expressed by Eq. (2).

$$Q_i = \text{sign}(\Delta P)C_{bi}\sqrt{\frac{2\Delta P}{\rho}} \quad (i=sa, sb, bt, aL)$$

The pressures, $P_a$ and $P_b$, are defined by Eq. (3).

$$\dot{P}_j = \frac{K}{V_{j0}+\Delta V} \sum Q_i$$

Where, $V_{j0}$ ($j=a, b$) is the initial volume of Chambers a and b. $K$ is the bulk modulus of water, $\Delta V$ represents change of volume.

The pressure in Chamber c, $P_c$, is given by Eq. (4).

$$P_c = (P_{c0} + P_{atm})\left(\frac{V_{c0}}{V_{c0} + A_c x_p}\right)^n - P_{atm}$$

Where, $P_{c0}, P_{atm}, V_{c0}, A_c, n$ represents the initial pressure in Chamber c, atmospheric pressure, initial gas volume in the accumulator, pressure receiving area on the top of the piston, polytropic index (=1.37), respectively.

The momentum equation of the piston is defined by Eq. (5).

$$M\ddot{x}_p = P_a A_a + P_b A_b - P_c A_c - f$$
$M$ is mass of the piston. $A_a$ and $A_b$ are pressure-receiving area on the piston in Chambers a and b. $f$ is the friction force attracted between the piston and cylinder wall. The value of $f$ was measured experimentally.

**FIGURE 2. Simulation model of ACA.**

**RESULTS AND DISCUSS**

Figure 3 illustrates the time history of $x_p$, $P_a$ and $P_b$ by experiment for three discharge coefficients. This result is extracted a cycle of boosting process. The pressure of $P_a$ changes with $x_p$ as shown here. The piston speed, which is recognized from the gradient of Fig.(a), becomes large with increase of the discharge coefficient $C_{bt}$ of $V_t$. The maximum of $P_a$ around 6sec. becomes higher with larger of $C_{bt}$. The value of $P_a$ gradually decreases with time after 6sec. until changing to charge mode. The variation pattern of $P_b$ is almost opposite that of $P_a$. Especially for the smallest case of $C_{bt}=0.04$, ACA does not work well. It is revealed that the discharge coefficient is very important factor for boost pressure, in other words, boost pressure of ACA can be control easily by adjust the discharge coefficient.

**FIGURE 3.** Time histories of $x_p$, $P_a$ and $P_b$ under various discharge coefficients in Experiment.

Copyright (C) JFPS All Rights Reserved.

- 2C10 -
Figure 4 describes the simulation result of boost process of ACA. Each result in Fig.4 shows same parameters, $x_p$, $P_a$, $P_b$ shown in Fig.3. Three cycles of boosting process can be seen in Fig.4. The result for magnitude relation is same as experiment result. The simulation result shows good agreement with the experimental result.

Next the authors investigated the influence of initial gas volume, $V_{c0}$, in the accumulator on the boosting performance by simulation. Figure 5(c) is variation of $P_{\text{gas}}(=P_c+P_{\text{atm}})$ in the accumulator. It is easily recognized that the amplitude of pressure variations of $P_a$ and $P_{\text{gas}}$ becomes smaller for larger volume of an accumulator.

Figure 6 shows experimental result of the relationship between the initial pressure in Chamber c, $P_c$ and the boosted pressure $P_a$ when the piston moves from $x_p=0$ to $x_p=50 \times 10^{-3}$ or $100 \times 10^{-3}$m. The different colored plot in the figure is results for the different initial pressure adjustment methods as mentioned above. From this result, it is cleared that the boosted pressure becomes higher of approximately 30% by adjusting of method 2, that is adjusted the initial pressure by changing of supplied water volume in Chamber c. This is because of quite different of initial gas volume between each adjustment method. The pressure in Chamber c drastically changes with change of the piston position in case of small initial gas volume by the method 2. It is suggested that the maximum value of boosted pressure can be easily controlled by only adjust of supplying water volume in Chamber c.

**CONCLUSION**

This study is focused on the pressure boosting process with Active Charge Accumulator (ACA). It was aimed to reveal important parameters for the boosting performance of ACA. Experimental and theoretical approach was conducted. As a result, the calculation model of ACA shows similar tendency of pressure variation with experimental result. Initial condition of Chamber c and flow discharge coefficient from Chamber b to the reservoir are the important for the boosting performance.

**REFERENCES**

3. F. YOSHIDA, K. Ito, S. IIO, A. KITAGAWA, Experimental and Theoretical Analysis of Active Charge Accumulator for Water Hydraulics System, The Institute of Electrical and Electronics Engineers (IEEE), 2016, Online ISSN: 2169-3536.


FIGURE 5. Time histories of $x_p$, $P_a$ and $P_{gas}$ under various initial gas volumes condition.

FIGURE 6. Relationship between boosted pressure and various initial pressures in Chamber c.
Abstract. The application prospects of micro hydraulic pump are broad. With the advantages of small volume, light weight and high pressure, micro pump has been applied in working condition of small flow and high pressure such as walking robots, fire protection and food industry. This paper proposes a new type of spherical micro pump which adopt a special structure. The mathematical models of the spherical micro pump’s kinematics and flow pulsation are established and related simulations are conducted. The pressure-flow characteristic of the pump is obtained by experiment.

Keywords: Spherical Micro Pump, Flow Pulsation, Kinematics, Structure

I. INTRODUCTION

In recent years, more and more attention has been paid to the micro fluid systems [1]. Micro pump, as the core component of the micro fluid system, is the key to the research of micro fluid system. There introduces a variety of micro pumps in [2]. According to the working principle, micro pump is classified into positive-displacement pump and non-displacement pump [3]. Positive-displacement pump includes gear pump, vane pump and piston pump. Non-displacement pump is mainly centrifugal pump. Because of complex structure, it is difficult to achieve miniaturization for piston pump and vane pump. Gear pump has the advantage of simple structure, but its structure could not achieve high discharge pressure as a consequence of severe leakage and imbalanced radius force, which limits the application of gear pump. In the past 20 years, micro pump has been developed quickly. However, most of them just have slight improvement on the basis of traditional pump without any breakthrough in structure [4, 5]. This paper introduces a new kind of micro pump, spherical micro pump.

Spherical micro pump, which adopts a new structure of spatial spherical mechanism, is a new kind of positive-displacement pump [5]. Depending on the periodical change of closed chamber shaped by the sphere and shell, the spherical micro pump has high output pressure. The structure of spherical chamber is compact and easy to realize microminiaturization. What’s more, this micro pump has reliable seal and less leakage [6]. Nonetheless, the main disadvantage of the spherical micro pump is severe flow pulsation. In order to decrease the effect of flow pulsation, the paper studies the kinematic relation between the components of spherical micro pump firstly and then analyses the factors that have influence on flow pulsation.

The remainder of this paper consists of following sections. Section II introduces the structure of the spherical micro pump. Section III investigates flow pulsation and the kinematical relation between the main components. Section IV shows the pressure-flow characteristic of the spherical micro pump and difference between other micro pumps. Section V is the conclusion.

II. BRIEF INTRODUCTION OF STRUCTURE

As shown in figure 1, the spherical micro pump is composed of three main components, piston, rotary disc and main shaft. The piston and rotary disc are hinged by the pin 8 and move together, and the main shaft is connected with rotary disc by the cylinder of the rotary disc inserting into the hole of the main shaft. The working principle is as followed. the main shaft drives the rotary disc by friction torque . The movement of rotary disc is combined by two motions: one is the rotation around its own axis, the other is the movement around the axis of main shaft, and the surface swept by the axis of rotary disc is a conical surface which uses center of the spherical chamber to be vertex and 2α to be the cone angle [7]. So the movement of rotary disc in the whole process is equivalent to fixed point rotation around center of the spherical cavity. The piston only rotates around its own axis, whose angular velocity is equal to the angular velocity of rotary disc
moving around its own axis because they are hinged together by pin. When put into operation, the volume of two working chambers V1 and V2 composed of cylinder cover, cylinder block, rotary disc and piston changes periodically to realize the absorption and discharge of fluid. From the figure 1, we can know the structure of spherical micro pump is very simple, compact and easy to realize microminiaturization. As shown in figure 2, the size of the physical spherical micro pump is very small. The displacement of the pump shown in figure 2 is 1ml/r, maximum speed is 3000 r/min and rated pressure is 4 MPa.

The spherical micro pump has following advantages:

a. Since all sliding friction pairs requiring prevention fluid from leakage are face-to-face seal structure rather than linear seal, the seal of spherical micro pump is reliable and has less leakage;

b. Spherical micro pump has simple structure, few parts and realizes miniaturization easily;

c. High output pressure;

d. The pump can rotate forward and backward, and the performance parameters don’t change.

III. ANALYSIS OF KINEMATICS AND FLOW PULSATION

Kinematic Analysis

According to the previous description, the movement of rotary disc is combined by two parts, the rotation around the axis of main shaft and the movement around its own axis. Now assuming the angular velocity of main shaft is \( \omega \), the angular velocity of rotary disc around the axis of main shaft is also \( \omega \). But the angular velocity of rotary disc around its own axis can’t be gotten directly. In order to get the angular velocity, a coordinate system is established as shown in figure 3, which use the center of sphere as the coordinate origin, the axis of main shaft as coordinate axis \( z \) and axis of pin as coordinate axis \( y \). Coordinate axis \( x \) can be determined by the principle of right-hand rule. The direction of axis \( x, y, z \) are shown in figure 3. In order to understand conveniently, the coordinate system is simplified as figure 4 where two points \( c, d \) are the endpoints of pin’s axis and points \( a, b \) are located on the axis of rotary disc and piston respectively shown in figure 3. At this time, axis \( y \) coincides with the axis of pin.

After a certain period of time, main shaft rotates by an angle \( \theta \) and point \( a \) and \( c \) move to point \( a', c' \) as shown in figure 4. By the geometric relation, the vector \( \overrightarrow{oc} \) can be expressed as follow:

\[
\overrightarrow{oc} = (\cos \alpha \sin \varphi, R \cos \varphi, R \sin \varphi \sin \alpha)
\]  

Where \( R \) is the radius of sphere shown in figure 3, \( \varphi \) is the angle of pin rotating around axis \( ob \), \( \alpha \) determined by the structural parameters is the angle between axis \( z \) and axis of piston.

The vector \( \overrightarrow{oa} \) is written as:

\[
\overrightarrow{oa} = (l \sin \alpha \cos \theta, l \sin \alpha \sin \theta, -l \cos \alpha)
\]  

Where \( l \) is the length of \( \overrightarrow{oa} \), \( \theta \) is the angle of main shaft rotation.
In the whole movement, $oa$ is always perpendicular to $oc$, because the structure determines that the axis of rotary disc is always orthogonal to the axis of pin. Thus, the geometric relation is written as:

$$oa \cdot oc = 0$$

(3)

According to the three expressions (1), (2) and (3), the relation between $\theta$ and $\phi$ is calculated as follow:

$$\phi = \arctan \frac{1 - \cos \theta}{\sin \theta \cdot \cos \alpha}$$

(4)

So the angular velocity of rotary disc around its axis is:

$$\omega_\phi = \frac{d\phi}{dt} = \frac{\cos \alpha}{\cos^2 \alpha (1 + \cos \theta) + 1 - \cos \theta} \cdot \omega$$

(5)

Where $\omega$, $\omega_\phi$ are the angular velocity of main shaft and rotary disc around its axis, respectively.

The angular acceleration of rotary disc which rotates around its axis is shown as:

$$\varepsilon_\phi = \frac{d^2\phi}{dt^2} = \frac{-\cos \alpha \sin^2 \alpha \sin \theta}{[\cos^2 \alpha (1 + \cos \theta) + 1 - \cos \theta]^2} \cdot \omega^2$$

(6)

Where $\varepsilon_\phi$ is angular acceleration.

From the formulas (4), (5), (6), the simulation result is as follow. The simulation parameters are shown in Table 1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle $\alpha$</td>
<td>$\pi/18$</td>
<td>rad</td>
</tr>
<tr>
<td>Angular velocity $\omega$</td>
<td>$50\pi$</td>
<td>rad/s</td>
</tr>
<tr>
<td>Time $t$</td>
<td>0.2</td>
<td>s</td>
</tr>
<tr>
<td>Step size $\Delta t$</td>
<td>0.002</td>
<td></td>
</tr>
</tbody>
</table>

FIGURE 3. Coordinate $o$-xyz

According to the three expressions (1), (2) and (3), the relation between $\theta$ and $\phi$ is calculated as follow:

$$\phi = \arctan \frac{1 - \cos \theta}{\sin \theta \cdot \cos \alpha}$$

(4)

So the angular velocity of rotary disc around its axis is:

$$\omega_\phi = \frac{d\phi}{dt} = \frac{\cos \alpha}{\cos^2 \alpha (1 + \cos \theta) + 1 - \cos \theta} \cdot \omega$$

(5)

Where $\omega$, $\omega_\phi$ are the angular velocity of main shaft and rotary disc around its axis, respectively.

The angular acceleration of rotary disc which rotates around its axis is shown as:

$$\varepsilon_\phi = \frac{d^2\phi}{dt^2} = \frac{-\cos \alpha \sin^2 \alpha \sin \theta}{[\cos^2 \alpha (1 + \cos \theta) + 1 - \cos \theta]^2} \cdot \omega^2$$

(6)

Where $\varepsilon_\phi$ is angular acceleration.

From the formulas (4), (5), (6), the simulation result is as follow. The simulation parameters are shown in Table 1.

TABLE 1. Simulation parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle $\alpha$</td>
<td>$\pi/18$</td>
<td>rad</td>
</tr>
<tr>
<td>Angular velocity $\omega$</td>
<td>$50\pi$</td>
<td>rad/s</td>
</tr>
<tr>
<td>Time $t$</td>
<td>0.2</td>
<td>s</td>
</tr>
<tr>
<td>Step size $\Delta t$</td>
<td>0.002</td>
<td></td>
</tr>
</tbody>
</table>

FIGURE 5. The relation between $\phi$ and $\theta$
Figure 5 shows the relation between $\phi$ and $\theta$ with different $\alpha$, 10°, 20°, 30°. The simulation results show that the smaller $\alpha$ is, the stronger the linear relation between $\phi$ and $\theta$ is and $\phi : \theta$ is almost 1 : 2. In other words, when the main shaft rotates $2\pi$, rotary disc only rotates $\pi$. Form the figure 6 and 7, we can know the angular velocity changes cyclically and the cycle is 0.04s same as the main shaft rotation. Moreover, with the $\alpha$ increasing, the amplitude of angular velocity and angular acceleration also increases. Thus, the $\alpha$ should be chosen reasonably and can’t be too large, otherwise the vibration and noise of the pump will increase which will cause a bad performance for pump. Generally, $\alpha$ is on more than 15°. In addition, at 0.02s, 0.06s, 0.1s, 0.14s and 0.18s, the angular velocity is smallest and the angular acceleration is 0, which means that the pump has the problem of dead point. At these five moments, the axis of piston and rotary disc rotate to a same straight line, which is just the position where the dead point occurs.

Analysis of Flow Pulsation

According to the kinematic analysis, the relation of rotation angle $\phi$ and $\theta$ is 1:2, that is to say, when main shaft revolves around its axis once, the piston and rotary disc only rotates half a cycle. As for the working chamber, when the piston rotates one cycle, two working chambers $V_1$ and $V_2$ respectively absorb and discharge fluid one time. Thus, it’s equivalent that when the main shaft rotates one cycle, only one working spherical chamber absorbs fluid, another discharges fluid. So the displacement of spherical micro pump is equal to the volume of a single working chamber shown as follow:

$$V = \frac{\psi}{2\pi} (V_1 - V_2)$$

(7)

Where $V$ is the displacement of spherical micro pump. $\psi$ is the maximal angle between the inclined plane of piston and rotary disc that is shown in figure 3. $V_1$ is the volume of spherical cavity which is included by cylinder cover and cylinder block. $V_2$ is the volume of the pin. The equations of $\psi$, $V_1$ and $V_2$ can be written:

$$\psi = 4\alpha$$

(8)

$$V_1 = \frac{4\pi}{3} R^3$$

(9)
\[ V_z = \frac{4\pi}{3} R^3 - \frac{4\pi}{3} R^2 \sqrt{R^2 - r^2} + \frac{4\pi}{3} r^3 \sqrt{R^2 - r^2} \]  
(10)

Where \( R \) and \( r \) is radius of the spherical cavity and the pin respectively.

The average flow of the pump is described as Eq. 11.

\[ Q = Vn = \frac{8}{3} \alpha n (R^3 - r^3)^{\frac{2}{3}} \]  
(11)

In order to analyze the flow pulsation, it needs to get the instantaneous flow of the pump. According to the working principle of the spherical micro pump, we know that the volume changing of working chambers is caused by the change of angle between piston and rotary disc. It means that the instantaneous flow is related to the changing ratio of the angle between piston and rotary disc. Now, assume that the include angle of working chamber varies by \( \Delta \beta \) within \( \Delta t \), and the volume of pump discharging fluid can be written:

\[ \Delta V = \frac{\Delta \beta}{2\pi} (V_i - V_o) \]  
(12)

Where \( \beta \) is the angle between piston and rotary disc. When \( \Delta t \) is small enough, the instantaneous flow can be described:

\[ Q_i = \frac{\Delta V}{\Delta t} = \frac{1}{2\pi} \frac{\Delta \beta}{\Delta t} (V_i - V_o) = \frac{1}{2\pi} \frac{d \beta}{dt} (V_i - V_o) \]  
(13)

The relation of angle \( \beta \) changing with time \( t \) should be discussed so as to get the instantaneous flow. Establish a local coordinate \( o-x'y'z' \), which also use the center of spherical cavity as the coordinate origin, the axis of piston as the axis \( z' \), and the axis of pin as the axis \( y' \). The position relation between local coordinate \( o-x'y'z' \) and global coordinate \( o-xyz \) is shown in figure 8 where Point \( e \) is the point on the piston shown in figure 3. At the initial position, the angle between \( oe \) and \( x' \) is \( 2\alpha \). When the piston rotates, point \( e \) and \( c \) move around the axis \( z' \) in circle. As shown in figure 8, after a certain time \( t \), \( c \) and \( e \) rotate by an angle \( \varphi \) and move to point \( c' \) and \( e' \) respectively.

\[ \vec{oe} = (R \cos 2\alpha \cos \varphi, R \cos 2\alpha \sin \varphi, R \sin 2\alpha) \]  
(14)

\[ \vec{no} = (R \cos 2\alpha \cos \alpha \cos \varphi + R \sin 2\alpha \sin \alpha, R \cos 2\alpha \sin \varphi, R \sin 2\alpha \cos \alpha - R \cos 2\alpha \sin \alpha \cos \varphi) \]  
(15)

The normal vector of piston inclined plane is written:

\[ n = \vec{oc} \times \vec{oe} \]  
(16)

\[ = (R^2 \cos \varphi \sin 2\alpha \cos \alpha - R^2 \cos 2\alpha \sin \alpha, R^2 \sin 2\alpha \sin \varphi, -R^2 \cos 2\alpha \cos \alpha - R^2 \sin 2\alpha \sin \alpha \cos \varphi) \]
What’s more, because of the vector $\vec{oa}$ always perpendicular to the inclined plane of rotary disc, it can be as to the normal vector of rotary disc. Therefore, the angle $\beta$ can be expressed:

$$\cos \beta = \frac{\vec{oa} \cdot \vec{n}}{||\vec{oa}|| ||\vec{n}||} = \sin 2\alpha \sin \alpha \cos \alpha \cos \varphi (1 + \cos \theta) - \cos 2\alpha \sin^2 \alpha \cos \theta + \sin 2\alpha \sin \alpha \sin \theta \sin \varphi + \cos 2\alpha \cos^2 \alpha$$

(17)

The instantaneous flow of spherical micro pump is obtained by bringing Equ.17 into Equ.13. Since the equation is too complex, the specific expression isn’t given here. The instantaneous flow curve of the pump under different angle $\alpha$ can be obtained by introducing relevant parameters. The parameters are shown in Table 2.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle $\alpha$</td>
<td>$\pi/12$</td>
<td>rad</td>
</tr>
<tr>
<td></td>
<td>$\pi/9$</td>
<td>rad</td>
</tr>
<tr>
<td></td>
<td>$5\pi/36$</td>
<td>rad</td>
</tr>
<tr>
<td>Sphere radius $R$</td>
<td>12.5 mm</td>
<td></td>
</tr>
<tr>
<td>Pin radius $r$</td>
<td>5 mm</td>
<td></td>
</tr>
<tr>
<td>Angular velocity $\omega$</td>
<td>50$\pi$</td>
<td>rad/s</td>
</tr>
<tr>
<td>Time $t$</td>
<td>0.2 s</td>
<td></td>
</tr>
<tr>
<td>Step size $\Delta t$</td>
<td>0.002 s</td>
<td></td>
</tr>
</tbody>
</table>

Figure 9 shows the instantaneous flow of the spherical micro pump under different angle $\alpha$. The simulation results show that the instantaneous flow varies greatly from 0 to maximum value and then from maximum to 0, which is mainly caused by the structure of the pump because of the special structure determining the two working chambers to wrok at different periods. With the $\alpha$ increasing, the amplitude of the instantaneous flow also increase. It can be proved by the Equ.11.

Now we discuss other factors that affect flow pulsation. The parameter flow pulsation rate, which is used to estimate the degree of flow pulsation, is introduced. The expression is as follow:

$$\eta = \frac{Q_s - Q}{Q} = \frac{d\beta}{dt} \frac{1}{4\alpha n}$$

(18)

The flow pulsation rate has nothing to do with the radius of spherical cavity and pin, and only relates to the angle $\alpha$ and the angular velocity $\omega$. But $\omega$ is only related to the frequency and can’t have effect on the amplitude of flow pulsation rate. So this paper doesn’t discuss the influence of $\omega$. Setting $\alpha=15^\circ, 20^\circ, 25^\circ, 30^\circ, \omega=50\pi$, the simulation curves of flow pulsation rate is shown in figure 10.

From the figure 10, the flow pulsation rate is very large and even reaches to 100% at some time which may cause pressure shock to some extent and unstable movement of actuator when the pump is used in a hydraulic system. So it’s necessary to adopt some measures to decrease the pressure shock, for example, connecting an accumulator at the outlet of the pump. What’s more, with the angle $\alpha$ increasing, the flow pulsation rate decreases to some extent, but the change is very small. So, by changing the angle $\alpha$, the improvement the flow pulsation of the pump is not obvious.
To understand the spherical micro pump more fully, let’s see the performance of the pump. Through the experiment of the prototype, we get some datum. The experimental result shows that the micro pump can not only use mineral oil as working medium, but also water. The maximum pressure can become 24 Mpa when working medium is mineral oil and the pressure of water is up to 5Mpa. Pressure-flow characteristic curve is shown as follow with the experiment datum:

Compared to the existing micro water pumps on market, the spherical micro pump has a great advantage. Figure 13 are the pressure-flow characteristic curves of other kinds of micro water pumps, Italy fluid-0-tech, Nanjing Oukerui, Italy ULKA, and Shanghai suofu. From these curves, the following conclusions can be obtained:

**IV. PERFORMANCE EXPERIMENT**
Figure 13. Pressure-flow characteristic of other kind of micro pump

(a) The pressure of spherical micro pump is much higher than that of other micro pumps.
(b) The slope of spherical micro pump’s pressure-flow characteristic curve is smaller. So the Pressure-flow characteristic is better than other water micro pumps.
(c) At the same pressure, the volume efficiency of spherical micro pump is higher than others.
In a word, the spherical micro pump has a good performance.

V. CONCLUSION

(1) The spherical micro pump adopts a new kind of structure that is simple and completely different from conventional structure of hydraulic pumps. This spherical micro pump has less components to realize micro-miniaturization easily and has high output pressure and high volumetric efficiency comparing with other micro water pumps.
(2) The angle $\alpha$ has a great influence on angular velocity and acceleration of rotary disc. The bigger the angle $\alpha$ is, the greater the amplitude of the angular velocity and acceleration is, which will cause pump producing severe vibration and noise. Thus, $\alpha$ should be chosen reasonably when design the spherical micro pump. Generally, $\alpha$ is on more than 15°.
(3) The spherical micro pump has large flow pulsation because of its structure. What’s more, flow pulsation rate is only related to the parameter angle $\alpha$. But changing the angle $\alpha$ has a little influence on flow pulsation rate.

ACKNOWLEDGMENTS

The authors would like to thank the professor Yinshui Liu from the School of Mechanical Science and Engineering, Huazhong University of Science and Technology, for valuable technical suggestions. This work is supported by the Hust-Wuxi Research Institute.

REFERENCE

Oral Presentation | Oil hydraulics

[2C12-16] H9 (Hydraulic Valves 1)
Chair: Yinshui Liu (Huazhong University of Science and Technology), Yutaka Tanaka (Hosei University)
Thu. Oct 26, 2017 3:30 PM - 4:50 PM Room C (ACROS Fukuoka)

[2C12] ANALYSIS OF FLOW CONTROL VALVE IN HYDRAULIC SYSTEM USING PARTICLE EXCITATION
*Takahiro Ukida¹, Koichi Suzumori¹, Hiroyuki Nabae¹, Takefumi Kanda² (1. Tokyo Institute of Technology, 2. Okayama University)
3:30 PM - 3:46 PM

[2C13] COMPUTATIONAL ANALYSIS OF SOLENOID SPOOL VALVE CONSIDERED OF LEAKAGE FLOW
*Fumio Shimizu¹, Takahiro Tsukazaki¹, Takayuki Hori¹, Kazuhiro Tanaka¹, Tomohiro Yasuda², Masahito Watanabe² (1. Kyushu Institute of Technology, 2. Nidec Tosok Corporation)
3:46 PM - 4:02 PM

[2C14] A NOVEL PROPORTIONAL DIRECTIONAL VALVE WITH INDEPENDENTLY CONTROLLED PILOT STAGE
*Zhenyu Lu¹, Junhui Zhang¹, Bing Xu¹, Qi Su², Di Wang¹ (1. The State Key Lab of Fluid Power and Mechatronic Systems, Zhejiang University, 2. China Aerospace Science and Technology Corporation)
4:02 PM - 4:18 PM

[2C16] EXPERIMENT-BASED FLOW RATE INFERENTIAL MEASUREMENT METHOD OF HYDRAULIC VALVE
*Di Wang¹, Junhui Zhang¹, Bing Xu¹, Zhenyu Lu¹ (1. State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University)
4:34 PM - 4:50 PM
ANALYSIS OF FLOW CONTROL VALVE IN HYDRAULIC SYSTEM USING PARTICLE EXCITATION

Takahiro UKIDA*, Koichi SUZUMORI**, Hiroyuki NABAE**, Takefumi KANDA***

* Department of Mechanical and Aerospace Engineering, Graduate School of Science and Engineering, Tokyo Institute of Technology
2-12-1 Ookayama, Meguro-ku, Tokyo, 152-8552 Japan
(E-mail: ukida.t.ab@m.titech.ac.jp)

** School of Engineering, Tokyo Institute of Technology
2-12-1 Ookayama, Meguro-ku, Tokyo, 152-8552 Japan

*** Graduate School of Natural Science and Technology, Okayama University
3-1-1 Tsusima-naka, Kita-ku, Okayama, 700-8530 Japan

Abstract. In this paper, we attempted to clarify the driving condition of the flow control valve in hydraulics system using the particle excitation. A new analytical model was proposed to reveal the characteristics of the valve and the effect of the viscosity of the working fluid. It was examined the open state of the valve under several assumptions including the assumption of Reynolds equation, focusing on the movement of the particles near the orifices and the flow of the working fluid in the narrow flow path. The results of the analysis were compared with the experimental results. The analytical results indicated that the proposed model represents the open state of the valve in the low viscosity hydraulic oil.

Keywords: Proportional Control Valve, Hydraulics, Flow Control, Piezoelectric Element.

INTRODUCTION

Mobile robots driven by hydraulic actuators have the possibility to evolve into “tough robots” which are resistant to impacts from the external environment and can generate a large force [1–3]. Mobile robots comprise hydraulic actuators equipped with a large number of two-stage servo valves to control complex behaviors with multiple degrees of freedom. Compared to direct drive servo valves, two-stage servo valves are smaller and have high responsiveness [4], which makes them suitable for use in hydraulic actuator based mobile robots. However, two-stage servo valves have a complex structure and cost disadvantage.

In order to mitigate these disadvantages, we have developed a small, PZT driven flow control valve with a simple structure [5]. This valve was originally developed as a flow control valve for pneumatic systems [6, 7]. The orifices of the valve are sealed by the particles confined in the valve. They open and close because of the excitation of particles caused by the vibration of the orifice plate and fluid force; this results in a simple and compact structure. The open condition of the orifices is that the particles gain inertia above the pressing force due to differential pressure between the inlet of the valve and the outlet of the valve. However, when the viscous fluid is used as the working fluid, the effective flow does not occur unless the particles do not gain larger inertia than the above condition. Furthermore, the pressure generated by squeeze effect between the particles and the valve seat of the orifices prevents the movement of the particle. As a result, the flow path keeps narrow. This phenomenon also makes the working fluid hard to flow.

In this paper, we propose a new analytical model focusing on the flow between the particles and the valve seat of the orifices to reveal the characteristics of the valve and the effect of the viscosity of the working fluid. To evaluate the proposed model, the results of numerical simulation is compared with experimental results.
FIGURE 1. Photograph of the valve.

FIGURE 2. Working principle of the valve using particle excitation.

PRINCIPLE OF THE PARTICLE EXCITATION VALVE

Figure 1 and 2 show a photograph and a working principle of a particle-excitation flow control valve, respectively. The valve consists of the orifice plate, upon which multiple orifices have been arranged; particles, and piezoelectric elements, which are 0.2-mm-thick ring-shaped lead zirconate titanate (PZT). When this valve is not in operation, the particles which are carried onto the orifices by fluid force seal the orifices. The valve is the closed state at this time. When a sinusoidal voltage is applied to piezoelectric elements, the valve changes into the open state. The piezoelectric elements which the sinusoidal voltage is applied to at resonant frequency vibrate the orifice parts. The particles which gain the inertia by the vibration of the orifice parts separate from the orifices. This principle results in a simple and compact structure.

The condition that the particles separate from orifices is derived from the balance force acting on the particle [6].

\[ a_o > \frac{\pi r_o^2 P_s \pm mg}{m} \]  

(1)

Where \( a_o \) is the acceleration of the valve seat at the orifice part, \( P_s \) is supply pressure, \( r_o \) is the radius of the orifice, \( m \) is the mass of the particle, and \( g \) is the acceleration due to gravity.
When the valve is applied to hydraulics, for the valve to actually start operation, it is necessary that the acceleration at the orifices greatly exceeds the lower limit value of the acceleration obtained from Eq. (1). Figure 3 shows the acceleration of the orifice part at which the valve begin operations obtained from Eq. (1) and that obtained through experiments. Note that we defined the control valve stably opens, when the flow rate exceeded 5 ml/min. The experiment was conducted using silicone oil as the working fluid. The result indicates that the viscosity of the working fluid affects the driving of the valve. This fact shows necessity of an analytical model to clarify the effect of viscosity to the valve operation. This section discusses a modeling method for representing the phenomena under the valve operation.

Assumptions for modeling

Figure 4 shows the relationship between the flow rate and the applied voltage with the applied pressure of 0.5 MPa and the ability to flow per one orifice, when silicone oil with kinematic viscosity of 1 mm²/s is used as the working fluid. When the applied voltage was 120 Vp-p or more, not only was the flow rate of the valve higher than the maximum flow rate per one orifice, it was also saturated because all particles were sufficiently separated from all orifices. On the other hand, when the applied voltage was 100 Vp-p or less, the flow rate of the valve was less than the maximum flow rate per one orifice. The results predict that the particles are near the orifices and repeatedly opening and closing them, in sequence. Therefore, we examine the open state of the valve under the following assumptions, focusing on the movement of the particles near the orifices and the flow of the working fluid in the narrow flow path.
(i) When the flow path is narrow, the pressure of the chamber in the valve does not reduce.
(ii) The fluid between the particles and the valve seat of the orifices has the assumption of the Reynolds equation.
(iii) The relationship between the particles and the valve seat of the orifices are regarded as a parallel plate.
(iv) The particles separating from the orifices move on the vertical direction axis of the orifices.
(v) The valve seat moves on the vertical direction axis of the orifices.
(vi) The flow between the particles and the valve seat of the orifices is axisymmetric.
(vii) The collision between particles and the orifices does not affect the vibration of the orifice plate.

![FIGURE 5](image)

**FIGURE 5.** Relationship between vibrational velocity at the central part of the orifice plate and the applied voltage, where the silicone oil is used as the working fluid with kinematic viscosity value 1 mm²/s.

![FIGURE 6](image)

**FIGURE 6.** Pressure distribution around the particle.

**FIGURE 7.** Developed view of the flow path between the particle and the valve seat.

### Movement of the particles

The movement of the particle separated from the valve seat is obtained from initial velocity of the particle, the forces acting on the particle, and a coefficient of restitution between the particle and the valve seat. The initial velocity of the particle is obtained from the vibrational velocity of the orifice plate. As shown in figure 5, the relationship between the vibrational velocity of the orifice plate and the applied voltage is expressed by Eq. (2).

\[
\nu_0 = C_{vel} V \sin(\omega t + \phi)
\]  

(2)
Where \( v_o \) is the velocity of the orifice plate, \( C_{vel} \) is the Proportionality constant, \( V \) is the amplitude of the applied voltage to the PZT, \( \omega \) is the angular velocity, and \( \phi \) is the phase difference between the applied voltage and the velocity of the orifice plate.

The amplitude of the vibrational velocity changes by the position on the orifice plate because the orifice plate is vibrated like a clamped circular plate. Therefore, the initial velocity of the particles \( v_{init} \) is expressed by Eq. (3)

\[
v_{init} = C_{part} C_{vel} V
\]

Where \( C_{part} \) is the Proportionality constant.

The forces acting on the particle is obtained from the pressure distribution around the particle, the drag force due to the working fluid, and the gravity force. The pressure distribution around the particle is divided into three areas as shown in figure 6. The pressures in the area shown in figure 6 (a) and (c) are the supplied pressure and the atmospheric pressure from the assumption (i), respectively. The pressure distribution in the area shown in figure 6 (b), which is the narrow flow path is obtained from the assumption (ii). The area is developed as figure 7. From the assumption (iii), the height of the flow path is the average value of it. From the figure 7, the surface of the sphere \( h_p(r) \) and the average height of the flow path \( h_{ave} \) are expressed by Eq. (4) and Eq. (5).

\[
h_p(r) = r_p + z_h \sin \alpha - \sqrt{r_p^2 - (r - r_p - z_h \cos \alpha)^2}
\]

\[
h_{ave} = \frac{\int_{r_1}^{r_2} h_p(r) dr}{r_2 - r_1}
\]

Where \( r_p \) is the radius of the particle, \( z_h \) is the distance between the displacement of the particle based on the assumption (iv) and that of the valve seat based on the assumption (v), \( r_1 \) and \( r_2 \) are the distance to the lower edge and upper edge at the valve seat, and \( \alpha \) is the half apex of the valve seat, respectively. The flow in the area shown in figure 6 (b) is expressed by Navier-Stokes equations Eq. (6) in cylindrical coordinates with the assumption (ii) and (vi).

\[
\frac{\partial P(r)}{\partial r} = \mu \frac{\partial^2 v_r(r, h)}{\partial h^2}
\]

Where \( P(r) \) is the pressure between the particle and the valve seat, \( \mu \) is the dynamic viscosity of the working fluid, and \( v_r(r, h) \) is the flow velocity in the \( r \)-axis direction, respectively. The flow velocity \( v_r(r, h) \) is derived from Eq. (6) with the boundary conditions shown in Eq. (7). The flow velocity is expressed by Eq. (8).

\[
v_r(r, 0) = V_{r1}
\]

\[
v_r(r, h_{ave}) = V_{r2}
\]

\[
v_r(r, h) = \frac{1}{2 \mu} \frac{\partial P(r)}{\partial r} (h^2 - h_{ave} h) + \frac{h}{h_{ave}} (V_{r2} - V_{r1}) + V_{r1}
\]

Where \( V_{r1} \) and \( V_{r2} \) are the velocity of the valve seat and the particle in the \( r \)-axis direction, respectively. The continuity equation in the cylindrical coordinates is expressed by Eq. (9) with the assumption (vi).

\[
\frac{\partial v_r(r, h)}{\partial r} + \frac{v_r}{r} + \frac{\partial v_h(r, h)}{\partial h} = 0
\]

Where \( v_h(r, h) \) is the flow velocity in the \( h \)-axis direction. The pressure distribution \( P(r) \) is derived by integrating Eq. (9) over \( h \) from 0 to \( h_{ave} \) and substituting Eq. (8) for it. The pressure distribution is expressed by Eq. (10).
Where $V_{h1}$ and $V_{h2}$ are the velocity of the valve seat and the particle in the $h$-axis direction, and $C_1$ and $C_2$ are constants of integration, respectively. $C_1$ and $C_2$ are derived from the boundary conditions shown in Eq. (11). $C_1$ and $C_2$ are expressed by Eq. (12).

\[
P(r_1) = 0 \quad P(r_2) = P_s
\]

\[
C_1 = \frac{1}{\log \frac{r_2}{r_1}} (P_s - A(r^2_2 - r^2_1) - B(r_2 - r_1))
\]

\[
C_2 = -Ar^2_1 - Br_1 - \frac{\log r_1}{\log \frac{r_2}{r_1}} (P_s - A(r^2_2 - r^2_1) - B(r_2 - r_1))
\]

Where $P_s$ is a supply pressure.

The vertical force generated by pressure in the flow path $F_{path}$ is derived from $P(r)$ and the surface area of the particle in contact with flow path. $F_{path}$ is expressed by Eq. (13).

\[
F_{path} = \int_{r_1}^{r_2} P(r) \theta_c r dr
\]

Where \(\theta_c\) is the central angle in the developed view of the flow path. \(\theta_c\) is expressed by Eq. (14).

\[
\theta_c = 2\pi \sin \alpha
\]

From the above, the resultant force acting on the particle due to the pressure $F_p$ are expressed by Eq. (15).

\[
F_p = (0 - \pi r_o^2 P_s) + (F_{path} - \int_{r_1}^{r_2} P_s \theta_c r dr)
\]

The first term is the force due to the differential pressure between the pressure in the valve and the orifice, the second term is the force due to the differential pressure between the pressure in the valve and the flow path.

The drag force acting on the particle is used the Stokes drag shown as Eq. (16).

\[
F_D = 6\pi \mu r_p \nu_p
\]

Therefore, the acceleration of the particle $a_p$ is expressed by the Eq. (17).

\[
a_p = \frac{F_p + F_D \pm mg}{m}
\]

The collision between the particle and the valve seat is expressed by Eq. (18) from the assumption (vii).

\[
\nu_p' = \nu_o - e \nu_p + e \nu_o
\]

Where $e$ is the coefficient of the restitution, $\nu_p$ and $\nu_p'$ are the velocity of the particles before and after colliding with the orifice plate, $\nu_o$ is the velocity of the orifice plate before and after colliding with the particles.
The flow rate is derived from the average of the instant flow rate flowing through the area shown in figure 8. The instant flow rate \( Q(t) \) is expressed by Eq. (19).

\[
Q(t) = 2\pi r_o \int_0^{h_{ave}} v_f(r_f, h) dh
\]  

Therefore, the average flow rate \( Q_{ave} \) is expressed by Eq. (20).

\[
Q_{ave} = \frac{\int_0^T Q dt}{T}
\]  

Where \( T \) is the period of the integration.

**RESULTS OF THE NUMERICAL CALCULATION**

We calculated the average flow rate by the numerical calculation, when the silicone oil shown in Table 1 was used as the working fluid under the applied pressure of 0.1 MPa to 0.5 MPa. The input parameters are shown in Table 2. The values obtained through the previous experiment were used as the applied voltage and the driving frequency in the simulation. The calculated results are shown in Table 3. The experiment was conducted to obtain the applied voltage which is required for flow of 5 ml/min. Most results with the kinematic viscosity value 1 mm²/s show errors under 5% though 26.4% error occurred under 0.2 MPa. According to the increase of the kinematic viscosity, the results show large errors. It is considered that the increase of the error comes from the assumptions of (iii) in the previous section. Note that the calculated results were calculated by tripling the value obtained from Eq. (20) because there are three orifices arranged on the closest places from the center of the orifice plate.

**TABLE 1.** Condition used to calculate the average flow rate.

<table>
<thead>
<tr>
<th>No.</th>
<th>( \mu ) [×10⁻⁸ Pa s]</th>
<th>( \rho ) [kg/m³]</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1</td>
<td>818</td>
<td>818</td>
</tr>
<tr>
<td>#2</td>
<td>2610</td>
<td>869</td>
</tr>
<tr>
<td>#3</td>
<td>4480</td>
<td>897</td>
</tr>
</tbody>
</table>

**TABLE 2.** Model parameters for the numerical calculation.

<table>
<thead>
<tr>
<th>Item</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( r_o ) [mm]</td>
<td>0.4</td>
</tr>
<tr>
<td>( \alpha ) [rad]</td>
<td>( \pi / 4 )</td>
</tr>
<tr>
<td>( r_f ) [mm]</td>
<td>0.283</td>
</tr>
<tr>
<td>( r_f' ) [mm]</td>
<td>0.424</td>
</tr>
<tr>
<td>( m ) [mg]</td>
<td>2.11</td>
</tr>
<tr>
<td>( T ) [s]</td>
<td>1</td>
</tr>
<tr>
<td>( C_{rel} )</td>
<td>0.0074</td>
</tr>
<tr>
<td>( C_{mot} )</td>
<td>0.984</td>
</tr>
<tr>
<td>( e )</td>
<td>0.65</td>
</tr>
</tbody>
</table>
TABLE 3. Calculated flow rate and errors comparing its value and the experimental results.

<table>
<thead>
<tr>
<th>Kind of working fluid</th>
<th>$P_s$ [MPa]</th>
<th>$Q_{ave}$ [ml/min]</th>
<th>Error [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.1</td>
<td>4.94</td>
<td>1.2</td>
<td></td>
</tr>
<tr>
<td>0.2</td>
<td>3.68</td>
<td>26.4</td>
<td></td>
</tr>
<tr>
<td>0.3</td>
<td>4.89</td>
<td>2.2</td>
<td></td>
</tr>
<tr>
<td>0.4</td>
<td>4.93</td>
<td>1.4</td>
<td></td>
</tr>
<tr>
<td>0.5</td>
<td>5.23</td>
<td>4.6</td>
<td></td>
</tr>
<tr>
<td>#2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.1</td>
<td>24.2</td>
<td>384</td>
<td></td>
</tr>
<tr>
<td>0.2</td>
<td>23.3</td>
<td>366</td>
<td></td>
</tr>
<tr>
<td>0.3</td>
<td>14.5</td>
<td>190</td>
<td></td>
</tr>
<tr>
<td>0.4</td>
<td>10.3</td>
<td>106</td>
<td></td>
</tr>
<tr>
<td>0.5</td>
<td>9.59</td>
<td>91.8</td>
<td></td>
</tr>
<tr>
<td>#3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.1</td>
<td>115</td>
<td>2200</td>
<td></td>
</tr>
<tr>
<td>0.2</td>
<td>33.4</td>
<td>568</td>
<td></td>
</tr>
<tr>
<td>0.3</td>
<td>20.2</td>
<td>304</td>
<td></td>
</tr>
<tr>
<td>0.4</td>
<td>12.4</td>
<td>148</td>
<td></td>
</tr>
<tr>
<td>0.5</td>
<td>10.5</td>
<td>110</td>
<td></td>
</tr>
</tbody>
</table>

DISCUSSION AND CONCLUSION

In this paper, an analytical model considering the motion of the particle and the flow rate was proposed, when the particles are near the orifices. We compared the analytical results and the experimental results. The errors between the calculation results and the experimental results were maximum 26.4% and minimum 1.2%, when the silicone oil was used as the working fluid with the kinematic viscosity value 1 mm²/s. From the results, we verified the validity of the analytical model with the low viscosity fluid. However, the increase in the dynamic viscosity of the working fluid caused the larger error in the flow rate. This is due to the assumptions that the relationship between the particles and the valve seat of the orifices are regarded as a parallel plate. This hypothesis causes the difference of the pressure distribution, which gives larger effects in the results with the higher kinematic viscosity. Therefore, an improved model considering the spherical shaped flow path should be developed to obtain reasonable results under larger viscosity in future work.

ACKNOWLEDGMENTS

This research was funded by ImPACT Program of Council for Science, Technology and Innovation (Cabinet Office, Government of Japan).

REFERENCES

COMPUTATIONAL ANALYSIS OF SOLENOID SPOOL VALVE CONSIDERED OF LEAKAGE FLOW


*Department of Mechanical Information Science and Technology, Faculty of Computer Science and Systems Engineering, Kyushu Institute of Technology
680-4 Kawazu, Iizuka, Fukuoka, 820-8502 Japan (E-mail: shimizu@mse.kyutech.ac.jp)
** Design Department 1, R&D Division, Nidec Tosok Corporation
2-24-1 Sobudai, Zama, Kanagawa, 252-8570 Japan

Abstract. A solenoid spool valve is an indispensable component for controlling flow direction as well as flow rate in an oil-hydraulic system. Because little motion of the spool valve controls high pressure flow field in oil-hydraulic machines, high accuracy control of the spool valve is required. Final target of the present study is to clarify the oscillation mechanism of the spool valve and to propose the restraint measure of the oscillation. We supposed that leakage flow between a spool and a sleeve had a great influence on the valve control, and three dimensional simulations were carried out for the flow field inside the spool valve including the clearance space. In the present paper, two types of computational method were carried out for treatment of narrow space and a moving body, and the accuracy of the computational results was verified.

Keywords: Solenoid valve, Spool, Clearance, Leakage flow, CFD

INTRODUCTION

Oil-hydraulic system is widely used in construction and transportation machines. A solenoid spool valve is one of control devices which are key component of oil-hydraulic system, and it is an indispensable component for controlling flow direction as well as flow rate. Small size of a spool is located inside a valve body, the diameter is about 10 millimeters and the length is about 20 millimeters. The spool valve is operated under high-level pressure environment and the flow direction changes according to the spool motion. Because little motion of the spool valve controls high pressure flow field in oil-hydraulic machines, high accuracy control of the spool valve is required [1]. Sometimes oscillation of output pressure or flow rate occurs at spool valve operations, and it is possible that the output oscillation causes a serious damage. So we have to predict the fluid force acting on the spool accurately and to control the spool motion.

By recent progress of both computational research technique and computer performance, many computational studies were carried out to understand the flow characteristics inside the spool valve. Two or axisymmetric calculations [2-3] as well as three dimensional simulations [4-5] were carried out for many researchers. However, leakage flow between a spool and a valve body did not considered in any research because the clearance was very narrow and the influence of the leakage flow was ignored. To clarify instability oscillation of spool motion, we have to investigate pressure propagation and flow pattern at narrow clearance between the spool and the body.

Final target of the present study is to clarify the oscillation mechanism of the spool valve and to propose the restraint measure of the oscillation. We supposed that leakage flow between the spool and the body had a great influence on the valve control, and three dimensional simulations were carried out for the flow field inside the spool valve including the clearance space. In the present paper, two types of computational method were carried out for treatment of narrow space and a moving body, and the verification of the computational results was verified.

ANALYTICAL TARGET

A schematic figure of a solenoid spool valve treated in the present paper is shown in Fig. 1. There is a spool, a sleeve, a solenoid and a spring for main components of the spool valve. A spool was a cylindrical rod with several dent stages. The spool was enclosed by the valve body and the body is called a sleeve. The spool moved from side to side inside the sleeve, and it operated opening and closing of multiple flow passages at the same time. Flow direction of mechanical oil or regulation of flow rate was controlled by the spool. The spool motion was controlled using a solenoid and a spring on the both side of the spool.
Five kinds of flow ports were connected on the sleeve, and we called these ports, Drain 1, Outlet, Inlet, Feed Back, and Drain 2, from the left hand side. At first, the mechanical oil flowed from the Inlet port to the valve chamber. After that the oil was discharged from the chamber to the Outlet port. At the same time, oil also flowed from the chamber to the Feed Back area because the oil passage was connected from the valve chamber to the Feed Back port. This construction have the restriction effect of the valve oscillation. Surplus oil was returned to a storage tank through the Drain 1 and 2 ports. The spool had two diameters for rod thickness, large one was 9 millimeters and small one was 4 millimeters. The spool motion was controlled using an electrical solenoid on left hand side and a mechanical spring on right hand side. A uniform clearance between the spool and the sleeve was assumed and it was set at 17 micrometers.

In the present study, steady calculations were carried out to fix the spool position. Figure 2 shows the relationship between the spool position and the valve operation. Figure 2 (a) indicates the initial positon of the spool and the situation that the solenoid was turned off electricity. In this situation, the spool was pressed to the left hand side by the spring force and the Inlet port was closed and the Drain 1 port was opened. Because the Drain 1 port was connected to the atmospheric tank, the pressure inside the chamber and the Outlet port became zero in relative pressure. When the solenoid was turned on the electricity, the spool was moved to the right hand side and the Inlet port was opened. This situation was shown in Fig. 2(b) and the spool position was end point. Because the Inlet was opened, the pressure inside the valve chamber rose and it became equal to the inlet pressure. The spool displacement from the initial to the end positions was about 1.2 millimeters.

**FIGURE 1.** Schematic Figure of Solenoid Spool Valve

**FIGURE 2.** Relationship between Spool Position and Valve Operation

**COMPUTATIONAL METHOD**

In the present study, a commercial software ANSYS CFX 14.0 was used for three dimensional simulation [6]. For the governing equations, the continuity and the Navier-Stokes equations were used for three dimensional incompressible flow and they were discretized using a finite volume method. Computational region was five flow ports and inside area of a sleeve to the exclusion of a spool. Leakage flow were considered and the
clearance between the spool and the sleeve was set at 17 micrometers. In the present paper, steady calculation was carried out for several positions of the spool. The clearance region was very narrow compared with the valve space of the chamber. In future, we will challenge to calculate unsteady simulation with a moving spool, and we want to investigate the mechanism of the spool oscillation. We have to verify the computational technique which can treat both narrow clearance and a moving body simultaneously. So two kinds of method, immersed solid method and general grid interface (GGI) technique, were investigated in advance.

The immersed solid method uses multiple computational grids and permits overlapping of these grids to easily apply to complicated configuration. In the spool valve simulation, computational grids were prepared to a moving body and a stationary flow passage independently. The grid generation was very simple and the movement of the spool was easily considered. The computational result which was calculated using the immersed solid method was shown in Fig. 3. This result shows the velocity magnitude distributions when the spool was closed to the Inlet port. Oil flowed in the clearance between the spool and the sleeve, and oil flow through the spool body was also calculated. Because the spool had a solid body, unrealistic flow pattern was simulated. It seems that the interpolated process was caused for this reason. When the overlapped computational grids were used, data transfer between multiple grids was conducted. The interpolated process were carried out when the position of computational grid points did not agree with the overlapped region. Although we applied this immersed solid method to narrow clearance flow, flow field was created at the inside of the spool body. For this reason, we supposed that the clearance was very narrow compared with the grid interval and the accuracy of the interpolated process was wrong. So utilization of this method was gave up in the present study.

On the other hand, general grid interface (GGI) method is the technique which was previously set to a moving interface on the body. Flow passage and a moving body were set across the interface and a sliding interface is prepared to connect with the body and the flow passage. We prepared a cylindrical computational grid to solve the leakage flow and it was connected to the sliding interface. We can calculate the narrow leakage flow. Although flexibility of grid generation was restricted a little, high accuracy calculation was carried out in the GGI simulation. So we decided to use the GGI method in the present study.

PRELIMINARY ANALYSIS

At first, the computational accuracy using the GGI method for the narrow leakage flow was verified. We can create a fundamental leakage flow when simple configuration of a spool body is used. Preliminary analysis were carried out for two types of spool configuration. Schematic figure is shown in Fig. 4. First one has uniform diameter of the spool and the other one is dented at the center part. In the case of uniform cylinder, the flow passage between the spool and sleeve is equal the double cylindrical tube. The flow rate of the double cylindrical tube is theoretically evaluated. The eccentricity between the spool and the sleeve axis was considered, the following equation was obtained [7].
\[ \Delta Q \approx \frac{\pi d h^3}{12 \mu l} \Delta P \left(1 + 1.5 \varepsilon^2\right) \]  

Where \( \Delta Q \) is leakage flow rate [m\(^3\)/s], \( \mu \) is fluid viscosity [Pa*s], \( d \) is the spool diameter [m], \( h \) is the clearance height [m], \( l \) is the spool length [m], \( \Delta P \) is the pressure difference between inlet and outlet [Pa], \( \varepsilon \) is the eccentricity (=\( e/h \) [-]), \( e \) is the eccentric length between two axis [m]. From oil-hydraulic spool condition, \( d=9 \) [mm], \( \mu=0.0125 \) [Pa*s], \( h=20 \) [\( \mu \)m], \( l=20 \) [mm], \( \Delta P=0.55 \) [MPa] were given. We evaluated the leakage flow rate \( \Delta Q \) to changing of the eccentricity \( \varepsilon \), and the result was shown in Fig. 5. The horizontal axis indicates the eccentricity and the vertical axis indicates the output flow rate. The dash line indicates the theoretical data given by the Eq. (1). According to increase of the eccentricity, the flow rate also increases.

Computational results were also plotted in Fig. 5. Diamond marks indicate the result of the uniform cylinder tube, and square marks indicate the result of the single stage spool. In the case of uniform cylinder, the computational and the theoretical results were good agreement at each eccentricity. The computational calculation can be accurately simulated in narrow clearance flow. In the case of the single stage cylinder, the flow rate increased compared with the case of the uniform cylinder and it was about double mount of the flow rate. Both results shows the same tendency and the eccentricity was important factor for the leakage flow of the spool. Because the eccentricity between the spool and the sleeve axis was influenced to the flow rate of the outflow, we recognized that the eccentricity must be considered in future work.

\[ \text{FIGURE 4. Spool Body of Simple Configuration} \]

\[ \text{FIGURE 5. Flow Rate of Leakage Flow} \]

**COMPUTATIONAL RESULTS AND DISCUSSION**

External shape of the realistic configuration of the solenoid spool valve was shown in Fig. 6(a) and the whole computational grids were shown in Fig. 6(b). The computational region was five flow ports and sleeve inside. A spool was located the sleeve inside and the clearance between the spool and the sleeve was 17 micrometers. The total number of the grid points was about four million. The shear stress transport model was used for the turbulence model. On the Inlet port boundary, high-level pressure, such as 0.55 [MPa], was given. On the Outlet and Feed Back ports boundaries, the wall boundary condition was given. The atmospheric condition was given on the Drain 1 and 2 ports boundaries. The computational conditions were shown in Table 1.

Global view of the computational results were shown in Fig. 7. Pressure distributions of the sleeve inside with five ports were shown and three kinds of spool position were indicated. High pressure was given at the Inlet port, so the inlet area was drawn in red color. On the other hand, two Drain ports were drawn in blue color, because the drain port was connected to the atmosphere. When the spool was located at the initial position, the valve
chamber and the Outlet port had low pressure. According as the spool moved from left to right, the pressure increased inside the chamber. Because the Feedback room was connected to the valve chamber through the flow passage, so the Feedback port was same pressure to the Outlet port. It is confirmed that the pressure variation at each flow port could be calculated by the three dimensional simulation.

![Computational Domain and Grids](image)

**FIGURE 6.** Configuration of Computational Domain and Grids

### TABLE 1. Computational Conditions

<table>
<thead>
<tr>
<th>Setting conditions</th>
<th>Boundary conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid Density</td>
<td>Inlet 0.55 [MPa]</td>
</tr>
<tr>
<td>Fluid Viscosity</td>
<td>Outlet Wall</td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>Feedback Wall</td>
</tr>
<tr>
<td>Sleeve=Spool Clearance</td>
<td>Drain 1 0.0 [MPa]</td>
</tr>
<tr>
<td>No. of Grid Points</td>
<td>Drain 2 0.0 [MPa]</td>
</tr>
</tbody>
</table>

![Pressure Distributions](image)

**FIGURE 7.** Global View of Computational Results (Pressure Distributions)

The flow situation around the valve chamber was investigated in detail. Pressure color contours and velocity vectors plot near the chamber room were shown in Fig. 8. In Fig. 8(a), the Inlet port was closed and the Drain 1 port was opened. Because the Drain 1 port was connected to the atmospheric environment, the Drain 1 and the Outlet ports were zero as relative pressure. Although the Inlet port was closed, the clearance between the spool and the sleeve existed. So the leakage flow was formed and toward the valve chamber and the velocity vectors...
showed this situation. In Fig. 8(b), both the Inlet and the Drain 1 ports were closed. The pressure inside the chamber slightly rose, because the pressure propagated from the Inlet to the chamber region. Fig. 8(c) indicates that the Inlet port was slightly opened. As the Inlet port was opened, the pressure inside the valve chamber rapidly increased and the pressure level in the chamber was same at the inlet pressure. The inside velocity vectors shows that the oil was fast flowed from the inlet to the chamber. From these results, the pressure distribution and the flow velocity field rapidly change according to the spool position.

![Pressure and Velocity Diagrams](image)

**FIGURE 8.** Computational Results near Valve Chamber (Pressure Distributions and Velocity Vectors)

The relationship between the spool position and the outlet pressure was shown in Fig. 9 and both experimental and computational results were plotted. To compare with the computational results, an experimental measurement of the same configuration of spool valve were carried out. Diamond marks indicated the experimental results and square marks indicated the computational results. The operation of the Inlet and the
Drain 1 ports was indicated on the upper side of the figure. While the Drain 1 port was opened, the outlet pressure was almost zero in both results of the experiment and the computation. However, the outlet pressure changed for the closing of the Drain 1 port. The outlet pressure was almost zero in both results of the experiment and the computation. However, the outlet pressure changed for the closing of the Drain 1 port. The outlet pressure was gradually increased in the computational data. Although the Inlet port was closed in this moment, the leakage flow between the spool and the sleeve was created and the pressure propagation occurred. But in the experimental data, the pressure rise was slowly. This reason is as follows. In the experimental apparatus, the solenoid valve was enclosed by a manifold. So another clearance existed between the sleeve and the manifold. When the Drain 1 port was closed, it seems that the leakage oil flowed at the clearance between the sleeve and the manifold. So that the oil flow was not created from the Inlet port to the valve chamber, the pressure rise was slowly in the experimental data. Although the Inlet port was closed in this moment, the leakage flow between the spool and the sleeve was created and the pressure propagation occurred. But in the experimental data, the pressure rise was slowly. This reason is as follows. In the experimental apparatus, the solenoid valve was enclosed by a manifold. So another clearance existed between the sleeve and the manifold. When the Drain 1 port was closed, it seems that the leakage oil flowed at the clearance between the sleeve and the manifold. So that the oil flow was not created from the Inlet port to the valve chamber, the pressure rise was slowly in the experiment. When the Inlet port was opened, the pressure rapidly increased and the computational and the experimental results were almost same value. Because it is difficult to obtain the accurate experimental data using a small size spool, the completely correspondence with the experimental and the computational conditions was very hard. Since the computational results were qualitatively corresponded with the experimental results, the confidence of the computational results were confirmed.

![FIGURE 9. Relationship between Spool Displacement and Output Pressure](image-url)

**COCLUDING REMARKS**

Accuracy of the computational methods was verified to investigate the flow field inside a solenoid valve including the narrow clearance. It is clearly that the general grid interface technique was effective to calculate the leakage flow with moving body. And the computational method to solve the leakage flow inside a spool valve was constructed. The pressure propagation and leakage flow occurred near the valve chamber through the narrow clearance. We confirmed that these computational results were valid compared with the experimental data. By these consideration of the computational technique, we can construct the effective computational method. For the next step, we will investigate the fluid force acting on the spool body and the flow characteristics of the spool valve will be investigated.

**REFERENCES**

A NOVEL PROPORTIONAL DIRECTIONAL VALVE WITH INDEPENDENTLY CONTROLLED PILOT STAGE

Zhenyu Lu*, Junhui Zhang*, Bing Xu*, Qi Su**, Di Wang*

*The State Key Lab of Fluid Power and Mechatronic Systems, Zhejiang University
38 Zheda Road, Hangzhou, Zhejiang Province, 310058 P.R. China (E-mail: benzjh@zju.edu.cn)
**China Aerospace Science and Technology Corporation
Beijing 100048, P.R. China.

Abstract. The pilot operated directional valves are widely used in hydraulic systems. For the purpose to improve the performance of the traditional valve and not increase the manufacturing cost, this paper proposes a novel pilot valve that employs two independent valve spools instead of the traditional port coupled valve spool. Due to the new structure, the mass and the resistance force of the novel pilot valve are reduced, and its damping character is improved. According to comparative experiments between the novel valve and the traditional valve, the novel valve has impressive improvement of the entire valve performance that the -3dB-frequency of novel valve has improved more than 50% compared with the traditional valve.

Keywords: Directional valves; Pilot operated; Pilot valve; Independent valve spool;

INTRODUCTION

The pilot operated directional valves are widely used in hydraulic systems with a nominal flow rate of 100–1000 L/min [1]. It is composed of the pilot stage, the main stage and the pressure reducing valve as shown in FIGURE 1(a). The pilot stage of traditional valve is a four-way proportional directional control valve, which is used to control the pressure and direction of fluid flow to the main stage. The main stage is a pilot-operated four-way spool valve, which can be seen as a hydraulic cylinder controlled by the pilot control valve. The main spool with the LVDT position sensor is centered by a push-pull preload spring. The working principle has been described in the former paper [2]. We can conclude that the pilot stage is the key control component which the performance and stability of the entire valve system largely depend on.

The approaches which improve the performance of directional control valves by pilot stage design can be divided into two categories. The first category is to adopt the new kinds of electro-mechanical converters instead of the traditional proportional solenoid. The bi-directional solenoid can drive the valve spool realizing bi-directional movement. This kind of solenoid saves the installation space, because it replaces two traditional proportional solenoids installed symmetrically on both sides of the valve housing, such as the 4WRVE type of valve of Rexroth[3]. The voice coil motor has a higher response speed than the traditional solenoid[4], and it is used as the electro-mechanical converter for the D30FB series of valve of Parker. What’s more, new function materials such as the magnetic shape memory (MSM) materials[5], piezoelectric crystal[6][7][8], and giant magnetostrictive materials[9][10] that possess outstanding dynamics, whose frequency can reach several kHz, are applied into the design of new electro-mechanical converter. But it is always necessary to add a mechanical amplifier to overcome the stroke limitation of these new function materials, which enlarges the dimension of the entire valve. What’s more, the properties of these new materials are affected by temperature and nonlinear hysteresis.

The second method is to amend the valve structure of pilot stage. The flow channel and damping orifice can be optimized to improve the performance of hydraulic valves. Kenji Kawashima [11] used a slit structure instead of an orifice plate to reduce the noise and pressure fluctuations of the four-port nozzle-flapper-type servo valve. Bing Xu [12] utilizes a three-stage structure with two high-speed on/off solenoid valves as the pilot stage and two cartridge poppet valves as the second stage to overcome the fundamental tradeoff between valve response and flow capacity of a directional valve. Massimiliano Ruggeri [13-15] proposed a new patented roto- translating valve whose metering control precision is virtually quadratic in respect to the traditional valve spool position control. The D30FB series of valve of Parker employs a closed center type of pilot stage [4] that improves the compactness of valve.

For the purpose to improve the performance of the traditional valve and not increase the manufacturing cost, we propose a novel structure for the pilot valve of the two-stage directional flow control valve, as shown in FIGURE 1(b). The novel pilot valve employs two independent valve spools instead of the traditional valve spool whose control ports are coupled. The distribution of deadzones of the pilot valve is reconstructed. The mass and
the resistance force of the novel pilot valve are reduced. These changes can improve the performance of the entire valve.

![Diagram of Pilot Valve Stage and Main Valve Stage](image)

**FIGURE 1.** The layout of traditional valve (a) and the novel valve(b).

**THE STRUCTURE OF THE NOVEL VALVE**

The comparison of the traditional valve and novel valve

The control target of the proportional directional valve is to realize a stable, accurate and rapid tracking of a reference main spool position \( x_{\text{ref}}(t) \). Due to different reference positions, the control modes of the proportional directional valve can be divided into two kinds. The first is the single side control mode in which the reference main spool position changes during the right or left position of the main valve. The second is the both sides control mode in which the reference main spool position changes from right to left position of the main valve or from left to right.

![Diagram of Overlaps and Displacement](image)

**FIGURE 2.** (a) the structure of the traditional pilot valve; (b) schematic of traditional valve.

As shown in FIGURE 2(a), there are two kinds of overlaps in the pilot stage of traditional valve: the positive overlaps \( (L_{Ac}, L_{Bc}) \) and the negative overlaps \( (L_{Ad}, L_{Bd}) \). The positive overlaps is 1.1mm and the negative overlaps’ is 0.9mm. The schematic of the traditional valve is shown in FIGURE 2(b). In the schematic, the pressure reducing valve is simplified as a constant pressure oil source \( P_\text{ps} \) and the main stage of the valve is simplified as a hydraulic cylinder. Due to the relative position of the overlaps and the displacement of the pilot valve spool \( x_p \), the pilot stage of the traditional valve has five working states. They are left position \( (x_p < L_{Bc}) \), left intermediate position \( (L_{Bc} < x_p < L_{Bd}) \), center position \( (L_{Bd} < x_p < L_{Ad}) \), right intermediate position \( (L_{Ad} < x_p < L_{Ac}) \) and right position \( (x_p > L_{Ac}) \).

During different control modes of the proportional directional valve, the deadzones introduced by the overlaps that the pilot valve spool needs to move across are various. In the single side control mode, e.g. \( x_{\text{ref}}(t) \) stays in the left position, the pilot stage of the traditional valve needs to continuously switch the position between the right position and center position to keep tracking of \( x_{\text{ref}}(t) \). In this case, the pilot valve spool needs to move across the intermediate deadzone with length of \( (L_{Ac} - L_{Ad}) \). In the both sides control mode, when \( x_{\text{ref}}(t) \) changes direction, e.g. from left position \( (x_{\text{ref}}(t) < 0) \) to right position \( (x_{\text{ref}}(t) > 0) \), the pilot stage of the traditional valve needs to switch the position from right to left. In this case, the pilot valve spool needs to move across the
deadzone with length of $L_A$, first and then the deadzone with length of $L_B$. This causes the switching delay that cannot be avoided because of the coupled ports of the pilot stage of traditional valve.

![FIGURE 3](image)

Keeping the main structure of the valve unmodified to not increase the manufacturing cost, then cutting the traditional pilot valve spool into two parts (A and B) and reorganizing the ports distribution can lighten the influence of deadzones to the direction changing performance of the main valve. As shown in FIGURE 3(a), there are also two kinds of overlaps whose distribution are changed in the pilot stage of the novel valve: the positive overlaps ($L_{A}$, $L_{B}$) and the negative overlaps ($L_{A}$, $L_{B}$). Due to the relative position of the overlaps and the displacement of the pilot valve spool $x_{pA}$ or $x_{pB}$, the pilot stage of the traditional valve has three working states for both parts which are shown in FIGURE 3(b). They are pressure drain position ($0 < x_{pA} < L_{A}$, $0 < x_{pB} < L_{B}$), pressure holding position ($L_{A} < x_{pA} < L_{A}$, $L_{B} < x_{pB} < L_{B}$), and pressure supply position ($x_{pA} > L_{A}$, $x_{pB} > L_{B}$).

Due to the structure changes of the novel valve, the deadzone introduced by the overlaps that the novel pilot valve spool needs to move across is the same as the traditional valve during the signal side control mode, but different during the both sides control mode. In the both sides control mode, when $x_{m}(t)$ changes direction, e.g. from left position ($x_{m}(t) < 0$) to right position ($x_{m}(t) > 0$), right part of the pilot stage needs to change from activated state to inactivated state, and the left part needs to change from inactivated state to activated state. Due to the decoupled relationship of the two pilot valve spools, the deadzone that the novel pilot valve spool moves across is $L_{A}$ or $L_{B}$, which is shorter than deadzone of the traditional valve ($L_{A} + L_{B}$). This can decrease the switching delay time and improve the response speed of the entire valve in comparison with the traditional valve. Not only the deadzone is shortened for the novel valve, but also the damping effect of the pilot valve spool is decreased. We analyze the novel pilot valve spool A for example. The damping coefficient $B_A$ of the spool is composed of three sections. The first is the viscous damping effect induced by the surface of spool, the second is the viscous damping effect induced by the viscous friction force of the slender damping hole, and the third is the differential pressure force at the ends of the pilot valve spool induced by the movement of spool. The damping coefficients are described as below. The notations of the parameters are as shown in FIGURE 3(a).

The viscous friction caused by the surface of spool can be expressed by formula (1) [16-17].

$$F_{BA1} = \frac{\pi d_b \delta}{2} \sum A_P u = \frac{\pi d_b \mu}{\delta} \sum l_p V_p$$  \hspace{1cm} (1)

where $v_p$ is the speed of the pilot valve spool; $\mu$ is the dynamic viscosity of oil; $\delta$ is the tolerance clearance between pilot valve spool and valve housing; $d_b$ is the diameter of the pilot valve spool; $l_p$ and $A_P$ are the width and pressure difference of contact surface between the pilot valve spool and valve housing. For the contact surfaces between the pilot valve spool and valve housing are symmetrical, the first term of $F_{BA1}$ is zero, and the equivalent damping coefficient is rewritten as formula (2).

$$B_{A1} = \frac{\pi d_b \mu}{\delta} \sum l_p$$  \hspace{1cm} (2)

The viscous friction caused by the slender damping hole can be expressed by formula (3) [16-17].

$$F_{BA2} = -8\mu \pi l \left( \frac{d_{s}^2 - d_{p}^2}{d_{s}^2} \right) \frac{v_p}{d_{p}^2}$$  \hspace{1cm} (3)

where $l$ is the length of the pilot valve spool; $d_s$ is the diameter of slender hole of the pilot valve spool; $d_p$ is the diameter of pilot spool.

And the equivalent damping coefficient is expressed by formula (4).

$$B_{A2} = 8\mu \pi l \left( \frac{d_{s}^2 - d_{p}^2}{d_{s}^2} \right)$$  \hspace{1cm} (4)
The differential pressure of the ends of the pilot valve spool caused by the movement of spool can be expressed by formula (5) [16-17].

$$\Delta p = -\frac{128\mu \ell}{\pi d_b^2} q_c$$

(5)

where 

$$q_c = \frac{\pi (d_p^2 - d_b^2) v_p}{4}$$

is the flow rate through the slender hole of pilot valve spool.

Therefore, the differential pressure force at the ends of the pilot valve spool can be expressed by formula (6).

$$F_{\Delta p} = \Delta p \frac{\pi (d_p^2 - d_b^2)}{4} = -8\mu I l \frac{(d_p^2 - d_b^2)^2 v_p}{d_b^2}$$

(6)

And the equivalent damping coefficient is calculated by formula (7).

$$B_{\Delta p} = 8\mu I l \frac{(d_p^2 - d_b^2)^2}{d_b^2}$$

(7)

And the total equivalent damping coefficient is equal to the summation of $B_{\Delta p}$, $B_{\Delta 2}$, $B_{\Delta 3}$ as formula (8).

$$B_A = \pi \mu \left[ \frac{d_p}{\delta} \sum I_n + \frac{8l(d_p^2 - d_b^2)}{d_b^2} \left( 1 + \frac{(d_p^2 - d_b^2)}{d_b^2} \right) \right]$$

(8)

The total equivalent damping coefficients of the pilot valve spool $B$ can also be estimate by famous (8), as well as the total equivalent damping coefficients of the traditional pilot valve. From formula (8), we can see that the length of the pilot valve spool affects the damping coefficients significantly.

<table>
<thead>
<tr>
<th>Item</th>
<th>Value</th>
<th>Item</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_p$</td>
<td>15mm</td>
<td>$d_b$</td>
<td>2mm</td>
</tr>
<tr>
<td>$\delta$</td>
<td>10μm</td>
<td>$l$</td>
<td>25(50)mm</td>
</tr>
<tr>
<td>$\mu$</td>
<td>0.04 Pa·s</td>
<td>$\Sigma l_n$</td>
<td>4(8)mm</td>
</tr>
</tbody>
</table>

(Note: the value in the parentheses is of traditional pilot valve)

Substituting the values of the parameters in TABLE 1 to the formulas (2), (4), (7), (8) calculating the damping coefficients, we get the corresponding damping coefficients of the novel pilot valve and traditional pilot valve in the condition of 40℃ and 2mm slender hole diameter with the No.46 hydraulic oil, as shown in the TABLE 2. According to TABLE 2, the damping coefficient caused by differential pressure is the major section; the damping coefficient of the novel pilot valve is 79.07 Ns/m while it is the 158.14 Ns/m for the traditional valve. And along with the lighter mass of the novel valve spool which is 18g compared with 40g of traditional valve, the novel pilot valve may have a quicker response speed than the traditional valve.

### The detailed structure of the novel pilot valve and its advantages

The novel pilot valve spool is cut into two parts with exchanged distribution of the P port and T port compared with the traditional pilot valve spool, which is shown in FIGURE 4(b). Its housing is varied corresponding to the valve spool, and the centering springs are installed in the middle, as shown in FIGURE 4(d). The experimental measurement ports for pressure sensors in the housing are reserved, which can be deleted in the practical valve product. The assembly of the novel pilot valve is shown in FIGURE 5(a). The centering springs are fixed by the location pin and oriented by the guide sleeve. The manufactured novel pilot valve is shown in FIGURE 5(b).

---

Copyright (C) JFPS All Rights Reserved. - 2C14 -
In the view of manufacturing, the machining processes of both the housing and spool of the novel pilot valve are easier than the traditional pilot valve. The casting process of the valve housing is almost the same, but the length of the convex surfaces of the novel valve housing that needs to be ground at the same time is shorter than that of the traditional valve housing. The spool of the novel valve is shorter and its ports are without coupling relationship which must be considered in the traditional valve manufacturing processes.

In the view of dynamic characteristics, we can conclude that the novel pilot valve may have a quicker response speed, which can be attributed to three reasons.

1) The deadzone that the novel pilot valve spool moves across is shorter than deadzone of the traditional valve in the both sides control mode.
2) The mass driven by the solenoid of the novel pilot valve is only half of the traditional valve. The reduction of mass and dimension of the novel pilot valve spool decrease the inertial force, viscous force and friction.
3) The damping effect influenced by the slender damping hole is lightened due to the shorter length of the novel valve spool.

Therefore, the response of the novel pilot valve is quicker than the traditional pilot valve with the same driving condition of the solenoid. In the following sections, the dynamic characteristics of the novel valve are experimentally analyzed in comparison with the traditional valve.
EXPERIMENTAL CLOSED-LOOP POSITION CONTROL OF THE ENTIRE VALVES

The experimental closed-loop position control of the novel valve and the traditional valve which is Rexroth 4WRKE pilot operated proportional valve are carried out. The control strategies of the novel valve and the traditional valve are the same. The control strategy contains two closed-loops: the first closed-loop is the feedback controller of the solenoid current; the second closed-loop is the outer trajectory feedback controller of the position of the main valve spool with inverse compensation of center deadzone. The details of the control methods are described in the former papers [2, 18]. The valves used in the comparative experiments and the experimental facility are shown in FIGURE 6.

In the cases of 0~90% (single side control mode, SS) input step signal and -90%~90% (both sides control mode, BS) input step signal, the step response experiments are carried out. The curves and the statistical results are shown in FIGURE 7 and TABLE 3, where the response time value is rounded. It is shown that the novel valve has higher speed and less overshoot than the traditional valve. It is because the novel pilot valve spool has lighter mass and weaker damping effect, then it can reach the desired position more quickly than the traditional valve in the process of closed-loop control of pilot stage.

![FIGURE 6. The valves used in the comparative experiments and the experimental facility](image)

![FIGURE 7. The step response of the two kinds of proportional valve: (a) SS; (b)BS.](image)

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Novel valve</th>
<th>Traditional valve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delay time (0-90%, SS)</td>
<td>15ms</td>
<td>25ms</td>
</tr>
<tr>
<td>Rise time (0-90%, SS)</td>
<td>50ms</td>
<td>75ms</td>
</tr>
<tr>
<td>Overshoot (0-90%, SS)</td>
<td>2.22%</td>
<td>12.67%</td>
</tr>
<tr>
<td>Fall Time (0-90%, SS)</td>
<td>50ms</td>
<td>50ms</td>
</tr>
<tr>
<td>Delay time (±90%, BS)</td>
<td>5ms</td>
<td>5ms</td>
</tr>
<tr>
<td>Rise time (±90%, BS)</td>
<td>76ms</td>
<td>93ms</td>
</tr>
<tr>
<td>Overshoot (±90%, BS)</td>
<td>2.78%</td>
<td>5.56%</td>
</tr>
<tr>
<td>Fall Time (±90%, BS)</td>
<td>75ms</td>
<td>95ms</td>
</tr>
</tbody>
</table>

(Note: SS- single side control mode; BS- both sides control mode)
The result curves of the sinusoidal response experiments are shown in FIGURE 8. It is shown that the tracking ability of the novel valve is better than the traditional valve. The root mean square track error of novel valve is 2.6% and that of the traditional valve is 6.21% in the case of 0–90% input signal. And in the case of -90%~90% input signal, the root mean square tracking error of novel valve is 3.96% and that of the traditional valve is 7.48%. It is also because that the novel pilot valve spool has lighter mass and weaker damping effect.

What’s more, when the main valve spool changes the direction in the case of 0~90% input signal at the zero position and move across the zero position in the case of -90%~90% input signal, the response delay both exist. It is because of the center deadzone of the pilot valve. For the novel pilot valve has a shorter length of deadzone, $L_{Ac} + L_{Bc}$ for the traditional valve, $L_{Ac}$ or $L_{Bc}$ for the novel valve, that needs to move across, the delay time of the novel valve is less than the traditional valve.

Changing the frequency of the input signal, we can get the frequency response of the valves. The frequency responses of both valves are shown in FIGURE 9 and TABLE 4, where the frequency value is rounded. Due to the advantages of the novel pilot valve, the -3dB-frequency of novel valve improves more than 50% compared with the traditional valve.

For the purpose to improve the performance of the traditional valve and not increase the manufacturing cost, this paper proposes a novel structure for the pilot valve of the two-stage directional flow control valve. The structure of the novel pilot valve with decoupled ports makes the length of the deadzone that the novel pilot valve spool needs to move across become shorter than that of the traditional valve. What’s more, the mass and the damping
effect of the novel pilot valve are lightened, that improves the dynamic characteristic of the valve. The experimental closed-loop position control of the entire valves demonstrates that the novel valve has impressive performance improvement of the entire valve. The results of experiments show that the -3dB-frequency of novel valve improves more than 50% compared with the traditional valve.

In future, the work will be focused on the optimization of the pilot valve port and the control strategies of the pilot valve to improve the performance of the entire valve.

ACKNOWLEDGMENTS

The authors gratefully acknowledge the support of the National Natural Science Foundation of China [No. U1509204 and 51475462] and the National Basic Research Program of China (973 Program) [No. 2014CB046403].

REFERENCES

EXPERIMENT-BASED FLOW RATE INFERENTIAL MEASUREMENT METHOD OF HYDRAULIC VALVE

Di WANG*, Junhui ZHANG*, Bing XU*, Zhenyu LU*

* State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University
38 Zheda Road, Hangzhou, Zhejiang Province, 310058 P.R. China
(E-mail: benzjh@zju.edu.cn)

Abstract. In hydraulic system, the pressure and the flow rate are two main variables to be controlled. It is practical to measure the pressure for sensors’ small volume and low price. But the usage of flow sensors is limited by large volume and high cost. Therefore, attention is paid on the valve because the flow is mainly controlled by the valve orifices. When using the orifice flow equation in practice, the discharge coefficient is often assumed constant and the pressure differential is measured diversely. However, in different orifice shapes and flow states the discharge coefficient and pressure differential change much. In this paper, the near and far end pressure differentials for a commercial spool valve are measured and compared. The discharge coefficient is analyzed with pressure differential and the control signal. A function to evaluate the discharge coefficient is proposed so that the flow rate can be calculated.

Keywords: Flow rate, Inferential measurement method, Spool valve

INTRODUCTION

In the hydraulic systems, the main tasks of a spool valve are to allocate a defined flow and adjust the pressure. To precisely manage hydraulic power and improve valve intelligence, sensors are built into the valve by many companies. The AxisPro proportional valves of EATON embed four pressure sensors in all four ports and temperature sensor in T port at the opposite side of the connection surface [1]. The Rexroth 4WREQ type of valves, however, have three pressure sensors in P, A, B ports within a sensor plate at the connection surface [2]. And there are many other products that have embedded sensors to realize flexible control strategies and diagnoses on machine health and performance [3-4].

The pressure can be measured with the embedded sensors, but precise flow sensors are not installed widely for its large volume thus external help is required to measure the flow rate. The inferential measurement is a good way to know the flow rate with the help of pressure and displacement sensors but not flow sensors [5]. The often used equation of the volumetric flow rate through an orifice (Figure 1) is derived from Bernoulli’s equation as

\[ Q = C_d A \sqrt{\frac{2}{\rho} (P_1 - P_2)} \]  

where \( Q \) is the volumetric flow rate, \( C_d \) is the discharge coefficient, \( A \) is the orifice area, \( \rho \) is the fluid density, \( P_1 \) is the pressure at upstream and \( P_2 \) is the pressure at the vena contracta.

![Figure 1: Turbulent flow through an orifice and the pressure measurement points](image_url)

In equation (1), the discharge coefficient \( C_d \) and the pressure differential is closely related with the flow state and the valve structure. To calculate the flow rate accurately, many formulas are proposed for the discharge coefficient according to different flow states and orifice shapes [6-9]. These formulas give a variety of equations between the discharge coefficient and the Reynolds numbers. The equation for calculating the Reynolds number is shown in equation (2), where \( v \) is the average velocity of the flow, \( d \) is the hydraulic diameter, \( \eta \) is the dynamic viscosity, \( \mu \) is the kinematic viscosity, and \( S \) is the perimeter of the discharge orifice. The Reynolds number is related with the volume flow rate, which is related with the square root of the pressure differential.
Therefore, the discharge coefficient is related with the pressure differential and the relationship must be considered according to the above publications.

\[
Re = \frac{\rho vd}{\eta} = \frac{vd}{\mu} = \frac{\frac{Q}{A}}{\frac{\mu}{S}} = \frac{4Q}{\mu S}
\]  

(2)

As for the pressure differential, in the publication [9], Merritt derived the equation (1) for the orifices based on the pressure measurement points shown in Figure 1. Between the point 1 and the point 2, which is at the vena contracta, the flow is streamline or potential flow and the Bernoulli's equation can be used. The point 3 is chosen at the place where the kinetic energy of the jet is converted into an increase internal energy of the fluid by the turbulence and is used to get approximately equal pressure value of the point 2. Because the structure of the spool valves is quite complicated, the pressure measurement strictly according to the theory is not practical. And the ISO standard 4411 [10] is proposed to approximately measure the pressure differential. However, the installed positions of the mentioned sensors in paragraph 1 do not accord with the ISO4411:2008 [10], in which standard pressure-tapping plates are required to measure the pressure differential of the subplate-mounted valves. Also, the ISO standard changes the rules of measuring the pressure differential from measuring at the far end points to the near end points on the standard pressure-tapping plates in 1985 and 2008 versions [10-11]. The change of measurement positions of the pressure differential not only affects the measured pressure differential, but also affects the discharge coefficient. To use the equation (1) to calculate the flow rate in practice, the effect of the pressure measurement position change need to be analyzed first.

Because the pressure measurement is so important and it is not practical to obtain the theoretical pressure differential, experiments are conducted to find out the pressure differential difference at the far end and the near end of the valve ports. The effect of the pressure differential on the discharge coefficient is analyzed, and inferential measurement method is proposed to accurately calculate the flow rate based on the experiment results.

TEST RIG

Because of the complexity both in the orifice structure and the flow field, experiments are quite important to obtain the pressure distribution and its relation with the flow rate. A symmetrical subplate-mounted valve with electrical position feedback of the model 4WRKE16E200L from Rexroth is chosen as the tested valve. A standard pressure-tapping plate and a measurement block are designed according to ISO4411 [10] and ISO4401 [12] to measure pressure and temperature at different positions. The schematic of the standard pressure-tapping plate and the measurement block is shown in Figure 2. The measurement positions of the A-T port experiment are shown in Figure 2 as an example.

The standard pressure-tapping plate is installed at the connection surface of the valve and the measurement block is mounted under it. There are two sets of points designed for pressure measurement to evaluate the difference. The near end set is on the standard plate and is 15 mm in vertical direction from the connection surface of the valve according to ISO4411 [10]. The far end set is designed on the measurement block. The distance is chosen to be 11d1 according to the distance rules applied for testing the valves other than subplate-mounted valves and sandwich-mounted valves in ISO4411 [10]. The temperature measurement points are also
designed with the rule of at least 5\(d_1\) from the pressure measurement point in ISO4411 [10]. The \(d_1\) in the rules is the inside diameter of the tube, which is chosen to be 17.5mm according to ISO4401 [12].

The test rig is built based on the designed test devices and is shown in Figure 3. There are four pressure sensors, two temperature sensors and one flow rate sensor. The NI USB 6343 which has the resolution of 16bits and up to 500kS/s sampling rate is used as the data acquisition device. The valve is controlled with signal generator which is not shown in Figure 3. The main parameters of the sensors are listed in Table 1.

<table>
<thead>
<tr>
<th>Device</th>
<th>Range</th>
<th>Accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure sensor</td>
<td>0-350bar</td>
<td>±0.1%FS</td>
</tr>
<tr>
<td>Flow sensor</td>
<td>3-700L/min</td>
<td>±0.1%FS</td>
</tr>
<tr>
<td>Displacement sensor</td>
<td>±5mm</td>
<td>±0.05%FS</td>
</tr>
<tr>
<td>Temperature sensor</td>
<td>0-100°C</td>
<td>±0.5%FS</td>
</tr>
</tbody>
</table>

The hydraulic system is able to supply about 400L/min oil to the tested valve. The inlet pressure of the valve is adjusted by the electrical relief valve and the load pressure is adjusted by the throttle valve. The oil used in the test is HM46. The viscosity and density of the oil are calculated according to publication [13] based on the measured pressure and temperature. The experiment is conducted at the temperature around 50°C as the recommended temperature of the tested valve. The pressure differential and the flow rates are tested in P-B and A-T ports under the pressure differential of 5bar, 10bar, 15bar, 25bar and 35bar at the near end measurement points. The experiments for P-B port is conducted with the inlet pressure of 10MPa, and the T port is connected with the tank through the pipes when conducting the A-T tests.

RESULTS AND DISCUSSION

The flow rates under different control signal and different near end pressure differentials are shown in Figure 4 and Figure 5. Although the shape of the spool is symmetric, the flow rates and their trend under different control signal of P-B and A-T ports are quite different. The analysis on the effect of different pressure measurement positions and the inferential measurement method on the flow rates are proposed below. The minus sign only represents the polarity of the control signal.

The Effect of the Pressure Measurement Position

The pressure differentials at the near end and the far end of P-B and A-T ports are shown in Figure 6 and Figure 7. The near end pressure differential and the far end pressure differential both grow larger when the flow rates rise. And the pressure differential differences under different flow rates and control signals are demonstrated in Figure 8 and Figure 9. In P-B port, the flow rates become larger when the pressure differential rises. The pressure differential difference at the near end and the far end measurement points of P-B port is almost linear to the flow rates. However, in A-T port, the pressure differential difference is chaotic. The pressure differential difference first rises as that of P-B port when the spool displacement is small. But when the spool displacement is large, the pressure differential difference first rises and then drops at large flow rates. Also, it can be observed that the value of the pressure differential difference of A-T port is much smaller than that of the P-B port.
The pressure differential difference represents the flow states in the related ducts of the ports. The linearity of the pressure differential and the flow rates of P-B port represents that there is no mixture of jet and turbulence and the pressure loss is caused by the laminar or the turbulent flow in the ducts. The pressure loss will disturb the pressure differential measurement of the throttle orifice. The measurement points of 15mm at the pressure-tapping plate may be not close enough and the pressure sensors still have the potential to be installed closer to orifices of the spool valve.

FIGURE 6. Pressure differential of P-B port

FIGURE 7. Pressure differential of A-T port

FIGURE 8. Pressure differential difference of P-B port

FIGURE 9. Pressure differential difference of A-T port

However, the pressure differential difference of A-T port appears differently from that of the P-B port. And there are two factors that affect the pressure differential measurement. The first factor is the cavitation. The average velocity in the duct is quite high when the pressure differential is high at large spool displacement and can reach 27.7m/s maximally. Because the T port is connected directly through the pipes to the tank, the back pressure of the T port is low. Though the flow through the pipes builds up pressure, large flow through the orifice still causes cavitation because the built pressure cannot compensate the pressure loss caused by the flow speed. The pressure differential difference is partly compensated by the cavitation pressure.

The second factor is the jet flow through the bottom orifice. In P-B port, the flow is throttled mainly by the spool orifice in the valve. However, at the bottom of the T port, an orifice, which is shown in red circle in Figure 10, is designed. The orifice is measured to be with the diameter of 11.7mm and the thickness of 7.86mm. The comparison of the spool orifice area and the bottom orifice area is shown in Figure 11. The A-T port works as the spool orifice and the bottom orifice connected in series. When the control signal is larger than 6.5V, the bottom orifice replaces the spool orifice to work as the main throttling orifice. And the throttling effect of the bottom orifice becomes larger when the flow rates rise. The jet flow position of the bottom orifice changes with the change of the pressure differential and the spool displacement. So in the outlet ducts, the flow is mixed by jet and turbulent flow and the length of jet flow is uncertain. The pressure differential at the far end of A-T port consists of the jet effect and effect of the turbulent or the laminar flow in the ducts. The pressure at the near end point is higher than that at the vena contracta and the kinetic energy of the jet flow is still rising. Therefore the pressure sensors for T port are not appropriate to be installed inside of the valve for the effect of the designed orifice.
From the above analysis, the near end pressure differential measurement is better than the far end one for it excludes the duct effect though it is still affected by the inner structure of the valve. Another parameter to be looked into is the discharge coefficient to calculate the flow rate. Since the near end pressure measurement point is considered to be better, the discharge coefficient is compared with the near end pressure differential. The discharge coefficient is calculated and shown separately for P-B port and A-T port in Figure 12 and Figure 13. The discharge coefficient is affected by the pressure differential only at small spool displacement and small pressure differential. In most conditions of the experiments, the discharge coefficient is not sensitive to the pressure differential but is sensitive with the control signal and the spool displacement. The discharge coefficients are not the same in each port either. The discharge coefficient can be analyzed without considering the pressure differential and used in calculating the flow rate under most conditions.

Fitting and Calculation

From the above analysis, the discharge coefficient is not sensitive with the pressure differential. The discharge coefficient for P-B port and A-T port are shown separately in Figure 14 and Figure 15. The discharge coefficients at each control signal gather together and do not scatter for being at the different pressure differential. The trend of discharge coefficient in P-B port shows obvious difference at small control signal and large signal while that of A-T port remains similar. Therefore, the structure of the discharge orifices need to be analyzed because there is no obvious change of other external factors at different control signals. The schematic of the spool and the shape of the discharge orifice are shown in Figure 16. There are four grooves distributed evenly along the circumference. These four grooves form four separate discharge orifices at small spool displacement. At large spool displacement, the four discharge orifices connect with each other and form an annular discharge orifice. The discharge orifice area used in the calculation of the discharge coefficient changes exponentially with the change of the control signal and is shown in Figure 11. The control signal is chosen to be fitted with the discharge coefficient for being easy to use.
P-B Port Discharge Coefficient Analysis

When using the spool valve with pressure differential larger than 5 bar, the stream in the valve is erratic and the critical Reynolds number is low. Because the discharge coefficient is affected little by the pressure differential, the flow is considered to be highly turbulent that the discharge coefficient is affected much by the orifice shape. Though the annular orifice only appears when the absolute value of the signal is larger than 7.5V, the effect of leakage and small opening can be observed after -6V, which is therefore chosen to be the separate point. And the discharge coefficient is fitted in two separate regions to get accurate results.

The nonlinear fitting results are shown in Figure 17 and equation (3). In equation (3), \( C_d \) is the discharge coefficient and \( x \) is the control signal in voltage. The adjust R square defined in the file [14] to describe the fitting results are 0.99 and 0.93, which mean good fittings.
A-T Port Discharge Coefficient Analysis

Unlike in P-B port, the discharge coefficient of A-T port show similar relationship in small and large spool displacement. But to get more accurate fitting results, the relationship of the control signal and the discharge coefficient is still fitted separately.

\[ C_d = \begin{cases} 
1.72 \cdot e^{1.72x} + 0.36 & (x > -6) \\
1.6 \cdot e^{-0.037x^3 - 0.62x - 3.34} & (x \leq -6) 
\end{cases} \]  
(3)

The discharge coefficient and the nonlinear fitting result are shown in Figure 18 and equation (4). The adjust R square defined in the file [14] to describe the fitting results are 0.98 and 0.99, which mean good fittings. The fitted formula shows similar pattern with that of the P-B port. The series connected orifices make the discharge coefficient hardly affected by the pressure differential when it is large enough and make flow rate control much easier.

\[ C_d = \begin{cases} 
0.87 \cdot e^{0.27x} + 0.29 & (x > -6) \\
1.6 \cdot e^{0.89x} + 0.09 & (x \leq -6) 
\end{cases} \]  
(4)

It is found that when the control signal is given by -2V, the discharge coefficient of P-B port and A-T port are affected by the pressure differential when it is low. This phenomenon shows that the flow state is changed at low pressure differential and little spool displacement, when the Reynolds number and the pressure differential must be considered to calculate the discharge coefficient. However, this part of flow state is not discussed.

CONCLUSION

The flow rate of the spool valve, as well as its relationship between the pressure differential is studied through experiments. Pressure sensors are installed at the near end and the far end of the valve ports to get the pressure distribution near the valve. During the experiment, the following conclusions can be made:

1. The pressure measuring at the near end reduces the effect of the duct. And for P, B and A ports, the pressure sensors can be installed inside the valve while the pressure sensors need to be installed further than the near end points because of the effect of the valve structure for the T port.

2. The near end pressure differential greater than 5 bar at 50°C when the absolute value of the control signal is larger than 2V is studied in the experiments. The discharge coefficients are found to be affected little by the pressure differential in most conditions. So the flow in the valve is considered to be strongly turbulent and the discharge coefficients are affected mainly by the discharge orifice shape. Because of the shape difference of the P-B port and A-T port, the relationship of discharge coefficient with the spool displacement in these two ports is not the same though the spool shape is symmetric. The discharge coefficient curves with the control signal are fitted nonlinearly for each port and can be used to calculate the flow rate of the tested valve.
The relationship of the discharge coefficient with viscosity is not studied. Also, the flow rate under low pressure differential which may lead to laminar flow is also not studied. Further work will be focused on the above two questions to get a subtler analysis on the flow rate calculation.

ACKNOWLEDGMENTS

This work was supported by the National Natural Science Foundation of China [No. U1509204 and 51605425] and the National Basic Research Program of China (973 Program) [No. 2014CB046403].

REFERENCES

1 EATON. The brains to handle the brawn[Z]. Document No. V-VLPO-MR005-E3.
2 BOSCH RECROTH AG. 4/3-proportional directional valve direct operated, with pQ functionality[Z]. No. Re29050/03.13.
3 BOSCH RECROTH AG. Proportional directional valve, direct operated, with pQ functionality[Z]. No. Re29014/05.2015.
Oral Presentation | Oil hydraulics

[2C17-21] H11 (Hydraulic Valves 2)
Chair: Massimiliano Ruggeri (CNR-IMAMOTER), Kazuhiro Tanaka (Kyushu Institute of Technology)
Thu. Oct 26, 2017 5:00 PM - 6:20 PM  Room C (ACROS Fukuoka)

[2C17] SIMULATION OF THE PRESSURE CONTROL VALVE IMPROVING RESPONSIVENESS AND STABILITY BY VARIABLE RESTRICT ORIFICE
*Seiei Masuda1  (1. Control System Engineering Department, Aero-engine & Space operation IHI Corporation)
5:00 PM - 5:16 PM

[2C18] WORKING CHARACTERISTICS OF JET PIPE SERVO VALVE IN VIBRATION ENVIRONMENT
*yu wang1, yao bao yin1  (1. College of Mechanical Engineering, Tongji University)
5:16 PM - 5:32 PM

[2C19] CROSS-DOMAIN TOLERANCE DESIGN FOR DIRECTIONAL CONTROL VALVES
*Ralf TAUTENHAHN1, Jürgen WEBER1  (1. TU Dresden, Institute of Fluid Power)
5:32 PM - 5:48 PM

[2C20] THEORETICAL ANALYSIS ON SPOOL STUCK POSSIBILITIES OF ROTARY DIRECT DRIVE PRESSURE CONTROL SERVO VALVE
Yaobao YIN1, Feiyan XIA1, *Liang LU1,2, Jiayang YUAN1, Shengrong GUO3  (1. School of Mechanical Engineering, Tongji University, 2. State Key Laboratory of Fluid Power & Mechatronic Systems, 3. Aviation Key Laboratory of science and Technology on Aero Electromechanical System Integration)
5:48 PM - 6:04 PM

[2C21] VALIDATION OF AN ENHANCED MODEL OF STEADY-STATE FLOW FORCES FOR SPOOL VALVES
*Patrik Bordovsky1, Hubertus Murrenhoff1  (1. Institute for Fluid Power Drives and Controls (IFAS), RWTH Aachen University)
6:04 PM - 6:20 PM
Simulation of the pressure control valve improving responsiveness and stability by variable restrict orifice

Seiei, MASUDA *

* Control System Engineering Department, Research & Engineering Division, Aero-engine, Space & Defense Business Area, IHI Corporation, Japan (E-mail: seiei_masuda@ihi.co.jp)
229 Tonogaya, Mizuho-machi, Nishitama-gun Tokyo

Abstract. The differential pressure control structure of a jet engine fuel control system can be affected by variable operating condition. Depending on the operative conditions, the control system gain can change and as a result larger than normal oscillation and unstable movement can occur. In this paper, we analyzed the lag of a simplified differential pressure control system. It is empirically known that the differential pressure control system can be stabilized by reducing the damping orifice which increases the relief valve damping ratio. By Bode diagram analysis, it is found that by adjusting the damping ratio of the relief valve much larger than one, we can effectively stabilize the differential pressure control system. However, this can be compromised when there is a disturbance. To address this, we added a check valve as a variable orifice to maintain stability and to improve the responsiveness of the system.

Keywords: fuel metering system, pressure regulating relief valve, stability, responsiveness, Bode Diagram Analysis.

INTRODUCTION

In the fuel control mechanism of a jet engine, a relief valve constituted by a spool valve and a spring is used for differential pressure control of a metering valve. As for constant differential pressure, the flow rate passing through the orifice is proportional to the opening area. The differential pressure across the metering valve is kept constant by the relief valve, and the opening area of the metering valve is controlled to weigh the fuel to be supplied to the jet engine. Since the differential pressure control system is used in the entire engine operation region where the flight altitude and speed change greatly, the supply flow rate changes to 10 to 100% of the rated flow rate of the fuel pump and the control pressure changes to about 2 to 7 MPa. Because the gain also changes dramatically, unstable oscillating movement and fluctuation of the normal operation can happen. In the case of oscillating operation, it was empirically known that the differential pressure control system can be stabilized by reducing the damping orifice which increases the relief valve damping rate. However, the stabilization mechanism is not clear. Also by reducing the damping orifice, there is also the possibility of increasing the delay of the whole system and reducing the phase margin.

In this paper, we broke down the lag into two ways. The primary lag was caused from the fluid compressibility and the secondary lag by the spring mass system. We also analyzed the lag by the Bode diagram.

As a result, the differential pressure control system can maintain both stability and responsiveness without oscillation if the damping ratio of the relief valve is sufficiently larger than 1. As a result, keeping the damping ratio greater than 1 allows the phase margin of the crossover frequency in a wider operation range. This was found to be effective for stabilizing the system.

NOMENCLATURE

\[ A_o \] : area of the restrictor orifice
\[ A_{PR} \] : pressure-receiving area of the relief valve
\[ A_{VR} \] : opening area of the relief valve
\[ A_{VM} \] : opening area of the metering valve
\[ B \] : bulk modulus of the jet fuel
\[ c_r \] : coefficient of coulomb viscosity
\[ c_f \] : coefficient of flow rate
\[ C \] : coefficient of viscosity
\[ K_R \] : spring constant of the relief valve
\( M_R \) : mass of the relief valve \\
\( P_0 \) : pressure downstream the relief valve port \\
\( P_1 \) : pressure upstream the metering valve port \\
\( P_{10} \) : steady state value of \( P_1 \) at a stationary point \\
\( P_2 \) : pressure downstream the metering valve port \\
\( P_3 \) : back pressure of the metering unit \\
\( P_{RS} \) : reference servo pressure of a relief valve \\
\( \Delta P \) : differential pressure of the restrictor orifice \\
\( Q_P \) : Volumetric discharge flow of a fuel pump \\
\( Q_M \) : Metered-fuel-flow rate \\
\( Q_R \) : Fuel-flow rate crossing the relief valve \\
\( V \) : Volume of fuel upstream metering valve \\
\( s \) : Laplace operator \\
\( V_M \) : transient state value of \( V \) at a stationary point \\
\( \Delta X_R \) : transient state value of \( X_R \) at a stationary point \\
\( \Delta P_1 \) : transient value of \( P_1 \) \\
\( \Delta X_R \) : transient value of \( X_R \) \\
\( s \) : Laplace operator \\
\( V \) : Volume of fuel upstream metering valve \\
\( V_M \) : transient state value of \( V \) at a stationary point \\
\( \Delta X_R \) : transient state value of \( X_R \) at a stationary point \\
\( \Delta P_1 \) : transient value of \( P_1 \) \\
\( \Delta X_R \) : transient value of \( X_R \) \\

**FUNDAMENTAL EQUATION**

\[
\frac{V}{B} \frac{d[P_1]}{dt} = Q_P - Q_M - Q_R
\]

The fuel flow rate crossing the relief valve is expressed by the following equation.

\[
Q_R = c_d \cdot A_{FR} \cdot \sqrt{\frac{2 \cdot (P_1 - P_0)}{\rho}}
\]

Assuming \( P_0 \) is constant, equation (3) is acquired by linearization of equation (2).

---

**FIGURE 1.** Fuel System configuration and Relief valve configuration

FIGURE 1(a) shows the structure of a fuel control system. A relief valve is usually one of the important parts of fuel system that meters the demanded fuel flow rate. Dynamic behavior of a relief valve is mainly focused on in this paper. FIGURE 1(b) shows the structure of a relief valve. The fluid compressibility upstream of the metering valve is expressed by the following equation.

\[
\frac{V}{B} \frac{d[P_1]}{dt} = Q_P - Q_M - Q_R
\]
\[
\frac{\Delta Q_R}{Q_r} = \frac{\Delta A_{VR}}{A_{VR}} + \frac{1}{2} \frac{\Delta P_1}{P_1 - P_0}
\]  

(3)

Assuming AMV and P2 equal to constant, equation (4) is acquired as linearization of the fuel flow rate crossing the metering valve.

\[
\frac{\Delta Q_M}{Q_M} = \frac{1}{2} \frac{\Delta P_1}{P_1 - P_2}
\]  

(4)

With the metering valve front pressure being \( P_1 = P_{10} + \Delta P \), assuming \( Q_p \) is constant, the equation of motion of the variation is linearized near the stationary point.

\[
\frac{V}{B} \frac{d[\Delta P_1]}{dt} = -\frac{Q_p}{2} \frac{\Delta P_1}{(P_1 - P_0)} - \frac{Q_M}{2} \frac{\Delta P_1}{(P_1 - P_2)} - \frac{Q_R}{A_{VR}} \Delta A_{VR}
\]  

(5)

Laplace transform is carried out and it becomes the following expression.

\[
\Delta P_i = \frac{Q_p}{A_{VR}} \left( \frac{Q_p}{2 \cdot (P_1 - P_0)} + \frac{Q_M}{2 \cdot (P_1 - P_2)} \right) + \frac{V}{B} \cdot \Delta A_{VR}
\]  

(6)

The equation of motion of the relief valve is as follows.

\[
M_R \frac{d^2[X_R]}{dt^2} + c_r \cdot \frac{d[X_R]}{dt} + K_R \cdot X_R = A_{PR} \cdot (P_1 - P_{RS})
\]  

(7)

The relationship between the pressure drop due to the damping orifice and the relief valve speed is given by equations (8) and (9).

\[
A_{PR} \cdot \frac{d[X_R]}{dt} = c_d \cdot A_o \cdot \sqrt{\frac{2 \cdot (P_{RS} - P_2)}{\rho}}
\]  

(8)

\[
P_{RS} = \frac{\rho}{2 \cdot c_d \cdot A_o^2} \cdot A_{PR}^2 \left( \frac{d[X_R]}{dt} \right)^2 + P_2 = \frac{\rho}{c_d \cdot A_o^2} \cdot A_{PR}^2 \cdot \frac{d[X_R]}{dt} + P_{RS0} + P_2
\]  

(9)

Substituting equation (9) to equation (7), equation (10) and (11) are acquired. It is possible that the coefficient of viscosity is made be larger by reducing area of the restrictor orifice.

\[
C = c_r + \frac{\rho}{c_d \cdot A_o^2} \cdot A_{PR}^3 \cdot X_{R0}
\]  

(10)

\[
M_R \frac{d^2[X_R]}{dt^2} + C \cdot \frac{d[X_R]}{dt} + K_R \cdot X_R = A_{PR} \cdot (P_1 - P_2)
\]  

(11)

A damping ratio is shown in equation (12).

\[
\zeta = \frac{C}{2 \cdot \sqrt{\frac{K_R}{M_R}}}
\]  

(12)

An opening area of the relief valve is a function of the relief valve displacement.
\[ A_{PR} = f(X_R) \] \hspace{1cm} (13)

From the above, the transfer function of the differential pressure control system is the transfer function of the third-order lag system shown in FIGURE 2.

**FIGURE 2.** Block diagram of the linearized relief-valve transfer function

**ANALYSIS BY BODE DIAGRAM**

FIGURE 3(a) shows the Bode diagram of the open loop transfer function of the differential pressure control system with the fraction of critical damping of the relief valve varied. FIGURE 3(b) shows the Bode diagram of the closed loop transfer function. The phase margin increases as the crossover frequency decreases by increasing the damping ratio. Because the phase margin is large enough at the crossover frequency and the phase plot with a high damping ratio decreases gradually. FIGURE 3(a) indicates that the crossover frequency of the open loop gain plot whose damping ratio is 4 shall be almost 4000 rad/sec if the open loop gain changes 100 times which means that the gain plot raises 40 DB at all frequency. The phase margin of the open loop phase plot at 4000 rad/sec shall be almost 15 degrees and the closed loop system shall be stable. On the other hand, the phase plot with a low damping ratio, which is 0.3, decreases very sharp and the system shall be unstable and the phase margin shall be zero at almost 3000 rad/sec if the gain changes almost 10 times.

**FIGURE 3.** Bode diagram of the differential pressure control valve.
STABILIZATION OF A PRESSURE CONTROL SYSTEM

In the differential pressure control system consisting of the relief valve and the fluid compressibility, the stabilization can be achieved by adjusting the damping ratio to be sufficiently larger than 1. By reducing the restrict orifice area, it is possible to adjust the crossover frequency below enough the natural frequency of the relief valve while open loop transfer function may be varied largely. Analysis of linearized model indicates that the phase margin can be increased and the system stabilized. The above-described damping adjustment can acquire a large phase margin in a wider frequency range around the crossover frequency even at the expense of system responsiveness. Therefore, this is considered to be effective for preventing reduction of the phase margin due to gain varying and stabilization in the whole operation region of the system.

FIGURE 4 shows the AMESim model of the differential pressure control valve, and TABLE 1 shows the simulation conditions. The results of step response are shown in FIGURE 5. It can be seen that the differential pressure control valve can be stabilized by reducing the damping orifice diameter. FIGURE 5(b) shows that the fluctuation of differential pressure occurs if the diameter of a restrict orifice is larger than 0.6mm. Comparison of FIGURE 5(b) and FIGURE 5(a) indicates that changes of simulation condition affect the stability of differential pressure control valve. The stability margin of differential pressure control valve with a damping orifice diameter 0.6 mm is not high enough under step response of metering valve displacement condition. To ensure the stability of the valve all operating condition, it is desirable to choose the orifice whose diameter is smaller than 0.6mm.

FIGURE 4. Amesim simulation model of a differential pressure control valve

<table>
<thead>
<tr>
<th>Simulation condition</th>
<th>Setting value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rated discharge flow rate of a fuel gear pump</td>
<td>150 (Litter/min)</td>
</tr>
<tr>
<td>Diameter of the relief valve</td>
<td>15 (mm)</td>
</tr>
<tr>
<td>Volume of upstream of a relief valve spring room</td>
<td>0.001 (Litter)</td>
</tr>
<tr>
<td>Volume of a relief valve spring room</td>
<td>2 (Litter)</td>
</tr>
<tr>
<td>Rectangle port width</td>
<td>20 (mm)</td>
</tr>
<tr>
<td>Mass of a relief valve</td>
<td>0.04 (kg)</td>
</tr>
<tr>
<td>Diameter of restrict orifices</td>
<td>0.15, 0.3, 0.6, 1.5 (mm)</td>
</tr>
<tr>
<td>Spring constant of the relief valve</td>
<td>70 (N/cm)</td>
</tr>
<tr>
<td>Spring preload</td>
<td>60.92 (N)</td>
</tr>
<tr>
<td>Back pressure of the metering valve</td>
<td>6.895 (MPa)</td>
</tr>
<tr>
<td>Step input of the metering valve differential pressure</td>
<td>0.3447 (MPa)</td>
</tr>
<tr>
<td>Step input of the metering valve differential pressure</td>
<td>0.8274 (MPa)</td>
</tr>
<tr>
<td>Step input of the metering fuel flow rate (area of the</td>
<td>0.3447 (MPa)</td>
</tr>
<tr>
<td>metering valve)</td>
<td></td>
</tr>
<tr>
<td>metering fuel flow rate</td>
<td>15 (Litter/min)</td>
</tr>
<tr>
<td>metering fuel flow rate</td>
<td>135 (Litter/min)</td>
</tr>
</tbody>
</table>
When a disturbance is too large, such as when the back pressure of the relief valve fluctuates, or when a disturbance displaces, the relief valve responsiveness may be reduced. In these ways, the speed of the relief valve hits a plateau and saturates. Long settling time is required. Table 2 shows the large step response simulation conditions. The results of large step response are shown in Figure 6. In the event a disturbance causes a large oscillation to the metering valve, the differential pressure control valve will indicate a large step response, a ramp like response, which has a longer settling time. On the other hand, the smaller step response is similar to the response of the second order lag system.

Table 2 Conditions of large step response simulations

<table>
<thead>
<tr>
<th>Simulation condition</th>
<th>Setting value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter of restrict orifices</td>
<td>0.15, 0.3, 0.6, 1.5 (mm)</td>
</tr>
<tr>
<td>Back pressure of the metering valve</td>
<td>6.895 (MPa)</td>
</tr>
<tr>
<td>Step input of the metering valve differential pressure</td>
<td>0.4, 1.4 (MPa)</td>
</tr>
<tr>
<td>the metering valve differential pressure (area of the metering valve step response)</td>
<td>0.4 (MPa)</td>
</tr>
<tr>
<td>Step input of the metering fuel flow rate (area of the metering valve)</td>
<td>15, 135 (Litter/min)</td>
</tr>
<tr>
<td>metering fuel flow rate (Spring preload step response)</td>
<td>45 (Litter/min)</td>
</tr>
</tbody>
</table>

(a) Step response of demanded differential pressure (b) Step response of metering valve displacement

Figure 5. Step responses of the differential pressure control valve.

LARGE STEP RESPONSE OF A PRESSURE CONTROL SYSTEM

(a) Step response of demanded differential pressure (b) Step response of metering valve displacement

Figure 6. Large step responses of the differential pressure control valve.
IMPROVEMENT OF RELIEF VALVE SPEED SATURATION

In addition to controlling steady state stability by increasing the damping ratio of the secondary delay system of the relief valve, we applied a check valve as a variable orifice to the damping orifice. By applying this (i.e. variable orifice), we attempted to avoid speed saturation and stabilize the system as a whole. Responsiveness was verified by AMESim simulation for the differential pressure control valve using the variable orifice, and the results are reported.

The ramp-like response occurs when the damping force is too large and the valve response speed is high. Since the damping force is inversely proportional to square of the orifice area, we tried to simulate the speed saturation by improving the orifice opening area when the speed is large and by reducing the orifice opening area when the speed is small. A differential pressure control valve model incorporating a variable damping mechanism combining an orifice and a check valve is shown in FIGURE 7. TABLE 3 shows the simulation conditions. FIGURE 8(a) and FIGURE 8(b) show the simulation results when the disturbance (similar to Figure 6(b)) was applied to the differential pressure control valve incorporating the variable damping mechanism. By incorporating the variable damping mechanism, responsiveness was improved, the settling time was shortened by 0.14 second in FIGURE 8(a) by 0.08 second in FIGURE 8(b) and the fluctuation range of the metering valve differential pressure deviation was also reduced from 0.32 MPa to 0.08 MPa in FIGURE 8(b).

FIGURE 8(c) shows the comparison result of the relationship between the fuel flow rate passing through the damping orifice and the differential pressure. The differential pressure proportional to the damping force generated by the orifice is equalized by the variable damping mechanism. The check valves soon increases the flow rate across the damping orifice and check valves if the pressure become greater than cracking pressure of check valves. After check valve begin to open the flow rate increase linearly according to the check valve flow rate pressure gradient.

FIGURE 7. Amesim simulation model applying large step response improvement

<table>
<thead>
<tr>
<th>Simulation condition</th>
<th>Setting value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter of restrict orifices</td>
<td>0.3, 0.6 (mm)</td>
</tr>
<tr>
<td>Back pressure of the metering valve</td>
<td>6.895 (MPa)</td>
</tr>
<tr>
<td>Step input of the metering valve</td>
<td>0.3447 (MPa)</td>
</tr>
<tr>
<td>differential pressure (spring preload)</td>
<td>0.8274 (MPa)</td>
</tr>
<tr>
<td>Step input of the metering fuel</td>
<td>15 (Litter/min)</td>
</tr>
<tr>
<td>flow rate (area of the metering valve)</td>
<td>135 (Litter/min)</td>
</tr>
</tbody>
</table>
4. CONCLUSION AND FUTURE TASKS

Simulations showed that it is possible to improve responsiveness by applying a variable damping mechanism to the differential pressure control valve. Using the damping orifice and check valves, both responsiveness and stability can realize as following.

1. Adjust the damping ratio of the relief valve as large as 1 by varying a diameter of the damping orifice.
2. Verifying the stability of the relief valve. Reducing the diameter of orifice if the system is unstable.
3. Improving the responsiveness using the check valve whose opening area is as 3 times as the orifice.
4. Increase the cracking pressure if the relief valve is unstable.

The simulations results shows that the damping ratio is almost 3, the check valve opening area is as 3 times as the orifice and the cracking pressure is almost tenth of the demand differential pressure typically.

The future task is to verify this by using test data.

REFERENCES

1. Masuda Seiei., A relief valve to be a third-order lag system and stabilization adjustment method of fluid compressibility, FY2012 Fluid Power System Conference lecture meeting proceedings
2. Masuda Seiei., Improvement of a large step response of a differential pressure control system using variable area restrict orifices, FY2015 Fluid Power System Conference lecture meeting proceedings
WORKING CHARACTERISTICS OF JET PIPE SERVO VALVE IN VIBRATION ENVIRONMENT

Yaobao YIN and Yu WANG

College of Mechanical Engineering, Tongji University, Shanghai 200092, China,
(E-mail: 1310323@tongji.edu.cn)

Abstract. In order to focus on the performance of jet pipe servo valve in periodic vibration and design the jet pipe servo valve in extreme environment, mathematical model of working characteristics of jet pipe servo valve in periodic vibration environment is established. Influence of amplitude and frequency of the periodic vibration and the structure parameters on performance of jet pipe servo valve in periodic in vibration environment was studied. The system’s phase plane and amplitude-frequency characteristic curve are drawn. The results indicate that within the frequency range of [15, 200], resonance peak appears in about 200Hz. When the stiffness of feedback rod is 200 N/m, there is an evident resonance peak.

Keywords: Jet pipe servo valve; Periodic vibration; Structure parameters; Resonance peak; Feedback rod

INTRODUCTION

Hydraulic jet pipe servo valve is the crucial component in electro-hydraulic servo control system with advantages of high precision, fast response, reliable operation, light weight, high power density and easy installation, and the particular advantage of antipollution. Originating from the 1940s, there are several challenging issues associated with servo valve, such as basic operating characteristics, mechanism of vibration and noise, prediction of performance degradation and lifetime and improvement of extreme environment resistant ability.

These years, scholars in the entire world pay many efforts on the basic characteristics of jet pipe servo valve[1-4]. The influence of diameter of jet pipe and pressure of pre-stage on the stability of pneumatic jet pipe servo valve was researched in [5]. Pre-stage simplified model and parameter matching of jet pipe servo valve and analyzed mapping relationship of the structure parameters and the steady and dynamic characteristics were studied in [6]. Electro-mechanical-fluid interaction mathematical model was established and steady and dynamic characteristics of jet pipe servo valve were analyzed in [7]. Test was put to get the pressure characteristics of pneumatic jet pipe servo valve in [8]. CFD method was used to analyzed the interior flow field of jet pipe servo valve and jet pre-stage recovery pressure and flow characteristics of different structural parameters was discussed in [9]. New materials were used in jet pipe servo valve and steady and dynamic characteristic was studied in [10].

Vibration is one of the extreme environments which the jet pipe servo valve may be used in. But the characteristics of jet pipe servo valve in vibration environment are seldom researched. Period vibration is the simplest vibration mode. It is usually often used in the study and theoretical research and experimental research. In this paper, the resonance characteristics and the stability of jet pipe servo valve in period vibration are studied.

WORKING PRINCIPLE AND STRUCTURE DYNAMICS

Working principle of jet pipe servo valve
Structure of jet pipe servo valve is shown in Fig.1. The jet pipe servo valve consists of the permanent magnet torque motor, jet pipe pre-amplifier and slide valve. Oil is introduced by jet pipe to the amplifier and jet pipe swings with the armature. Oil jets from the jet pipe and the receiver receives the oil. Thus pressure acts on both sides of the slide valve. When the jet pipe deflects, the two holes in the receiver receive different volume of oil and therefor there is pressure difference at the two sides of the slide valve. Slide valve moves until feedback torque is equal to the pressure difference. The jet pipe servo valve output flow.
FIGURE 1. Structure of Jet Pipe Servo Valve

Structure dynamics
On the basis of the following assumptions, movement model of jet pipe servo valve is built by lumped parameter method.

(1) Slide valve moves at the same plane as the armature and the two dimensional movement in the plane is only taken into consideration.
(2) The mass of feedback rod is not considered and feedback rod is equivalent to a spring.

After the input current, the torque provided by torque motor is

\[ T_d = k_m i + k_m \theta \]  \hspace{1cm} (1)

The movement equation of armature is

\[ T_d = J \ddot{\theta} + B_a \dot{\theta} + k_a \theta + T_r \] \hspace{1cm} (2)

Substituting (1) into (2) yields

\[ J \ddot{\theta} - k_m \theta + k_a \theta + B_a \dot{\theta} + T_r = k_m i \] \hspace{1cm} (3)

\[ T_r \] is shown in Eq. 4

\[ T_r = (r + b) F_k \]

\[ F_k = k_f [x + (r + b) \theta] \] \hspace{1cm} (4)

The movement equation of slide valve is

\[ m \ddot{x} + B_x \dot{x} + f_f x + f_k = p_L A_x \] \hspace{1cm} (5)

But there is no accurate mathematical model describe the pressure difference \( p_L \), the \( p_L \)-current curve is tested as shown in Fig. 2.

As seen in Fig. 2, the pressure difference has a linear relationship with current.

\[ p_L = k_p \Delta I \] \hspace{1cm} (6)
According to the working principle of jet pipe servo valve, the pressure difference is actually the function of $\theta$. Combine Eq.3, Eq.5 and Eq.6. The experimental curve is obtained under steady state conditions, so differentials of $\theta$ and $v$ versus time is 0. Thus

$$p_L = k_{pi} \frac{(k_a - k_m)(B_f + k_f) + B_f (r + b)^2 k_f}{k_f (B_f + k_f) - (r + b)k_f k_{pi}} \theta$$  \hspace{1cm} (7)$$

Convenient for expressing, the pressure-angle coefficient is defined

$$k_{p\theta} = k_{pi} \frac{(k_a - k_m)(B_f + k_f) + B_f (r + b)^2 k_f}{k_f (B_f + k_f) - (r + b)k_f k_{pi}}$$ \hspace{1cm} (8)$$

**MATHEMATICAL MODEL OF JET PIPE SERVO VALVE IN PERIOD VIBRATION ENVIRONMENT**

Physical model of jet pipe servo valve in period vibration environment is shown in Fig.1. Jet pipe servo valve consists of two parts, armature and slide valve. Force provided by torque motor acts on the armature and armature deflects an angle $\theta$. Electromagnetic spring, damping of torque motor, spring tube and the feedback rod provide counter-force on armature. When armature deflects, there will be pressure difference on the two sides of the slide valve and the feedback rod and damping of slide valve offer counter-force.

Lagrange equation is used to analyze the composition motion.
Select coordinates relative to the body, $\theta$ and $x$. The kinetic energy, potential energy and dissipation function of the system can be obtained as follows.

**Kinetic energy**

When jet pipe servo valve is working in the period vibration environment, armature does rotation motion around the center and the center does period vibration.

According to the theoretical mechanics, the actual motion of armature is circular motion and the distance between the actual center and rotation center is $r$.

So the rotational inertia of the actual motion is

$$J_o = J_a - \frac{1}{2} m_r e^2$$

$$J_c = J_o + \frac{1}{2} m_l (e - \frac{\dot{y}}{\theta})^2$$

The kinetic energy of armature is

$$T_a = \frac{1}{2} J_o \dot{\theta}^2$$

(11)

When jet pipe servo valve is working in the period vibration environment, slide valve does period vibration. The kinetic energy of slide valve is

$$T_s = \frac{1}{2} m_s (\dot{x} + \dot{y})^2$$

(12)

So the kinetic energy of the system is

$$T = T_a + T_s$$

(13)

**Potential energy**

Potential energy is because of the elastic deformation of spring tube and feedback rod. So the potential energy is

$$U = \frac{1}{2} (k_a - k_m) \dot{\theta}^2 + \frac{1}{2} k_f [x + (r + b) \theta]^2$$

(14)

**Dissipation function**

Potential energy is because of the elastic damping of armature and slide valve and related with the relative speed of armature and slide valve to the valve housing. So the dissipation energy is

$$D = \frac{1}{2} (B_o \omega^2 + B_v \dot{v}_s^2)$$

(15)
Substituting equation (10)-(15) into equation (9), the motion equation of jet pipe servo valve in period vibration is

$$J \ddot{\theta} + B_a \dot{\theta} + k_i \theta + k_j (r + b) \ddot{\theta} + k_j (r + b) x = \frac{1}{2} m_r \ddot{\varphi} + k_j$$

$$m_2 \dddot{x} + B_r \dot{\varphi} + k_j (r + b) \dot{\varphi} + (k_j + B_j) x = k_p \dot{\varphi}_A - m_2 \ddot{\varphi}$$

(16)

**RESULT AND DISCUSSION**

The mathematical model can be used to predict the performance of jet pipe servo valve in period vibration environment and match the optimum parts.

Fig.5 shows the leakage of slide valve with vibrations of different frequencies and amplitude. There is an obvious resonance peak in the curve and the resonance frequency is about 200Hz. The Fig4 show the leakage of slide valve in period vibration. The value is shown as the percentage of leakage of slide valve to the flow rage. According to the National Aerospace Standard when the flow rate of jet pipe servo valve is less than 7L/min, the internal leakage should be less 4% of the flow rate plus the leakage of pre-stage. So the value should be less 4%.

The safe region is the red region and the frequency and amplitude of vibration should be limited.

![FIGURE 5. Leakage of Slide Valve with Vibrations of Different Frequencies and Amplitude](image)

**Effect of stiffness of feedback rod**

Fig.6 shows the amplitude of slide valve with different stiffness of feedback rod in vibration of different frequencies. It shows that the amplitude of slide valve will appear an resonance peak with the variation of stiffness of feedback rod.

![FIGURE 6. Amplitude of Slide Valve with Different Stiffness of Feedback Rod in Vibration of Different Frequencies](image)

To examine the relationship of the amplitude of slide valve and the stiffness of feedback rod, the curve is shown as Fig 6. It shows that when the stiffness of feedback rod is 200 N/m, the resonance peak is the biggest. So when the jet pipe servo is used in the vibration environment, the stiffness should not be near 200 N/m.
CONCLUSION

In this work, the working characteristics of jet pipe servo valve in period vibration have been investigated. A mathematical model is developed to predict performance and examine the parts. Resonance peak appears in about 200Hz, so the vibration of working environment should avoid 200Hz. When the stiffness of feedback rod is 200 N/m, there is an evident resonance peak.

This paper contributes to established a more accurate mathematical to provide basis for the design, manufacture and control of jet pipe servo valve.

NOMENCLATURE

- $T_d$: torque motor output torque
- $k_t$: electromagnetic torque coefficient
- $i$: input current
- $k_m$: magnetic torque spring stiffness
- $\theta$: paddle angle
- $J_a$: rotational inertia of the armature component
- $B_a$: damping coefficient of armature component
- $k_a$: spring tube stiffness
- $T_r$: drag load of armature movement
- $r$: distance of spring tube center of rotation to the center line of the nozzle
- $b$: the distance from the centerline of the nozzle to the centerline of the valve core
- $k_f$: feedback pole stiffness
- $x_v$: spool displacement
- $m$: mass of the slide valve
- $B_s$: the damping coefficient of the slide valve
- $f_f$: the stiffness of fluid dynamic
- $f_k$: friction force of slide valve
- $p_{L}$: the pressure difference of two sides of the slide valve
- $A_s$: the effective area of the slide valve
- $k_{pi}$: the pressure difference-current coefficient
- $J_o$: rotational inertia of the armature component to gravity center
- $m_i$: mass of armature
- $y$: equation of the vibration
- $Y$: amplitude of the vibration
- $\omega$: frequency of the vibration
- $t$: time
- $e$: the distance between gravity and the rotation center
- $J_c$: rotational inertia of the armature component to the actual center
- $T_a$: kinetic energy of armature
- $T_s$: kinetic energy of slide valve
- $T$: kinetic energy of the system
- $U$: potential energy
Dissipation energy

ACKNOWLEDGMENTS

This work was supported by the National Natural Science Foundation of China (51475332) and Aeronautical Science Foundation of China (20128038003).

REFERENCES

CROSS-DOMAIN TOLERANCE DESIGN FOR DIRECTIONAL CONTROL VALVES

Ralf TAUTENHAHN\(^1\) and Prof. Dr.-Ing. Jürgen WEBER\(^1\)

\(^1\)Chair of fluid-mechatronic systems, Technische Universität Dresden, Dresden, Germany
E-mail: ralf.tautenhahn@tu-dresden.de, fluidtronik@mailbox.tu-dresden.de

Abstract. The task of tolerance analysis usually addresses the question of the mechanical mountability of an assembly. We extend this viewpoint when talking about directional control valves in a cross-domain tolerance analysis; an analysis whose task is to determine the possible variation in the key product characteristics such as response dynamics, or flow gain, induced by a specific tolerance concept. On the other hand, tolerance synthesis aims at the determination of an optimal tolerance concept resulting in the compliance of the demanded tolerances for key product characteristics. Both issues require a way to identify the noise factors to be tolerated, a mathematical representation of the tolerances and a method to propagate their impact on the key product characteristics.

Keywords: manufacturing tolerances, analysis, synthesis, multiphysics, uncertainty

INTRODUCTION

In general, the key product characteristics of a directional control valve such as response dynamics or flow gain vary from piece to piece, e.g. due to: manufacturing variations, variations of material properties, and environmental influences. The conscious exploit of the individual component’s design limits and the use of special material properties generally require tighter tolerances in order to prevent a failure of the required functionality as a result of these variations. Nevertheless, these tolerances cannot be chosen arbitrarily tight in cost-effective production.

The aim of our research activities was therefore to identify suitable methods and tools for cross-domain tolerance analysis and for the systematic and knowledge-based definition of the tolerances allowed during valve design. The chosen approach was exemplified by both pneumatic switching valves and hydraulic proportional valves. To avoid confusion, this paper is restricted to an 5/2 indirect solenoid actuated spool valve for pneumatic applications.

HANDLING OF NOISE FACTORS

Tolerances need to be established for all noise factors that occur during production, e.g. variations of geometric features or material properties, taking into consideration noise factors arising from the environment, e.g. temperature (compare figure 1). Based on a structural analysis possible noise factors for typical directional control valves can be identified and logged according to different requirements areas along the phases of the product life cycle. Since the mechanical components are function carriers these noise factors can be taken into account during the evaluation of the system’s transfer function via the physical technical operating principle of the elementary functions.

The considered key product characteristics for this investigation are shown on the right side of figure 1. In particular

FIGURE 1. System context for tolerancing illustrated by a parameter diagram following Taguchi’s quality engineering method
these are the switching times $t_{A2}$ of the armature and $t_{S2}$ of the spool as well as the force reserve $F_{M,\text{endstop}}$ at the endstop of the armature and the current reserve $\Delta I_{M,\text{release}}$ when the armature releases from the valve seat.

Unfortunately, the available information about the noise factors is almost always imperfect. This imperfection constitutes of imprecision and/or uncertainty, e.g. different expert opinions about the possible value range of a parameter or an experimentally determined probability distribution function [1]. Especially in early design phases the available information is limited. Therefore, an appropriate mathematical formulation has to be chosen carefully. While imprecision can be modelled by crisp or fuzzy sets, uncertainty is typically expressed by measures like probability, possibility or plausibility. Probability approaches are most often used in literature as they allow wider tolerance ranges for the individual features to meet the acceptable tolerance ranges of the key product characteristics. This is due to the fact that they do not overestimate the coincidence of extreme values. On the other hand, the knowledge of the actual distribution functions is often not guaranteed; something that calls the trustworthiness of the results in question. The more detailed the knowledge about the noise factors gets during product development cycle, the more reasonable a probability treatment is.

The methods for propagating imperfect information through system models depend on the mathematical representation of these imperfections. There are intrusive and non-intrusive methods. To be able to use simulation tools which have already proven their suitability for the analysis of fluid power components, only non-intrusive methods were further considered. Probability methods in particular are the focus of a lot of research activities, e.g. [2] differentiates between simulation based methods, local expansion based methods, most probable points based methods, functional expansion based methods, and numerical integration based methods.

### COMPUTATIONAL MODELS

Directional control valves show strong interactions between the electrical, magnetic, mechanical and fluidic domains as well as non-linear transfer functions. A closed analytical equation for the calculation of the key product characteristics is not available and simulation models are needed to calculate the system’s response.

The known propagation methods for imperfect information usually require numerous model evaluations. Therefore, the computational costs are critical and the simulation with distributed parameters, e.g. FEM for magnetic fields, CFD for fluid flows, is not or only to a limited extent permissible. On the other hand, simulations with concentrated parameters allow for the consideration of cross-domain interactions and provide a sufficiently fast model computation, e.g. it is possible to model an electromagnetic actuator as a reluctance network. This allows for the calculation of the armature movement in a fraction of the time that an FEM calculation would need. If only static characteristics are required (e.g. for the analysis of a proportional valve’s flow gain) the phenomenological representation of these three dimensional and non-linear relationships by means of surrogate models has proven to be advantageous. Such a decrease in model complexity is often accompanied by an increase of the model uncertainty and requires a careful model validation. One must ensure to cover all relevant noise factors and their impact on the key product characteristics despite the simplification.

<table>
<thead>
<tr>
<th>sub-function</th>
<th>geometry based implementation</th>
<th>phenomenological implementation</th>
</tr>
</thead>
<tbody>
<tr>
<td>pilot stage: convert $E_{\text{el.}} \rightarrow E_{\text{mech.}}$</td>
<td>iron core reluctances</td>
<td>number of windings</td>
</tr>
<tr>
<td>pilot stage: adjust shift element</td>
<td>stroke range &amp; coil (temperature-dependent)</td>
<td>magnetic permeability</td>
</tr>
<tr>
<td>pilot stage: change resistance</td>
<td>spring pre-stress &amp; mass of armature</td>
<td>spring constant</td>
</tr>
<tr>
<td>main stage: convert $E_{\text{fluid.}} \rightarrow E_{\text{mech.}}$ and main stage: adjust shift element</td>
<td>stroke range &amp; spring pre-stress</td>
<td>friction force (pressure- and velocity-dependent)</td>
</tr>
<tr>
<td></td>
<td>piston force &amp; mass of spool</td>
<td>spring constant &amp; damping</td>
</tr>
</tbody>
</table>

To investigate the tolerance concepts all identified noise factors need to be related to the key product characteristics within the simulation model. Unfortunately, some of the underlying physics cannot be formulated with respect to
geometric tolerances or other noise factors. A good example is the resulting friction force at a piston sealing. It is a complex time/geometry/material/load - depending effect which is currently neither completely understood nor are there model approaches available which can be reduced to ordinary differential equations for implementation in concentrated parameter simulations [3]. In such cases it is again possible to describe these physics phenomenologically with different kinds of surrogate models that take into account noise factors as multipliers estimated from limiting samples. These surrogate models can be empirically motivated algebraic expressions or families of characteristics gained from measurements. When there is no data available, these multipliers can be used to develop limits that must be ensured during product development.

Table 1 summarizes the chosen implementation of the identified noise factors for the analyzed pneumatic directional control valve in a simulation model with concentrated parameters. As shown on the left side in figure 2 all relevant domains are considered within the same simulation model. Furthermore, the simulation results from the whole system’s model and the corresponding measurements are in alignment as depicted on the right side of figure 2.

**FIGURE 2.** General structure of system model with different sub functions and validation of simulation results against measurement

In some cases (e.g. the calculation of variance based sensitivity indices or the iterative evaluation of dispersion measures within the numerical optimization process for tolerance synthesis) the computational cost for simulations with concentrated parameters are still too high. A common approach is to use surrogate models for estimating the whole system’s behaviour with less computational effort. According to [4] artificial neural networks are especially suitable for systems with much more than ten noise factors. It is important to check the quality of the surrogate models with respect to the resulting error and against overfitting. The latter is done by using an additional test set; one that is not used for the generation of the surrogate model. When there is only a small error for these additional samples, the model is said to have a good generalization: something that is essential for utilization in tolerance analysis.

**TOLERANCE ANALYSIS**

The task of tolerance analysis usually addresses the question of the mechanical mountability of an assembly. We extend this viewpoint when talking about directional control valves in a cross-domain tolerance analysis; an analysis whose task is to determine the possible variation in the key product characteristics such as response dynamics, or flow gain, induced by a specific tolerance concept. Different methods for the propagation of imperfect information through a simulation model are described in literature. Monte-Carlo-Simulation is one example which is straightforward and easy to implement. In this case (quasi-)random samples are generated within the tolerance range for each noise factor and the corresponding key product characteristics are determined by iterative model evaluations, as illustrated in figure 3. Latin-Hypercube-Sampling with iteratively reduced correlation has proven an efficient sampling method capable of accounting for different distribution functions [5]. The following discussion explains how descriptive statistics can be used for the characterisation of the resulting variations and the determination of failure rates. For demonstration purposes, an initial tolerance concept was chosen based on the tolerance class ‘fine’ according to the DIN ISO 2768-1:1991 norm for all features.
FIGURE 3. Procedure of Monte-Carlo-Simulation: iterative model evaluation for randomly sampled design points

Histograms like shown in figure 4 can be used for a first visualisation of the calculated deviations. More detailed statistical parameters are depicted in the box plots aligned above these histograms. This type of diagram contains the following information:

- The red line shows the median of the data set.
- The limits of the box mark the first and the third quartile of the data set.
- The whiskers are used to determine outliers which themselves are labelled by plus-signs.

FIGURE 4. Results from a Monte-Carlo-Simulation for a directional control valve: histograms and box plots of the key product characteristics and empirical cumulative distribution function plot for the determination of failure rates

The switching time $t_{A2}$ of the armature shows a small variation of about 3.3 % around the mean value. Also, the electrical current reserve $\Delta I_{M,\text{release}}$ when the armature releases from the valve seat varies only within 3.8 %. Even when the reserve of the magnetic force at the armature end stop $F_{M,\text{endstop}}$ possesses an estimated standard variation of 12.2 % with respect to the mean value it will not become zero and therefore proper functioning of the pilot stage is ensured. On the contrary the time for the spool travel $t_{S2}$ is divided into two groups. For the first group of samples the spool reaches the end stop shortly after the pilot stage pressurizes the piston. For the second group of samples the spool will not arrive at the end stop until the end of simulation time and is, therefore, faulty. As shown on the right side of figure 4 an empirical cumulative distribution function plot can be used to estimate the corresponding failure rates.

It was possible to identify the failure cause as the interaction between the spring force, the pressurisation of the piston, and the friction force as a function of the pilot pressure and the spool velocity. An increasing spring pre-stress requires a higher pilot pressure on the piston to move the spool. Due to the higher pressure, the piston sealing lays tighter to the housing and, therefore, the friction force increases. This slows down the spool movement and finally the spool stops at an inclined position. In figure 5 the resulting threshold level between operating (labelled ‘1’) and faulty (labelled ‘0’) samples is clearly visible.

Furthermore, the calculated samples can be utilised to perform a sensitivity analysis. Without prior knowledge of the underlying transfer function characteristics, it is nearly impossible to choose an adequate sensitivity measure from the plurality of possible ones (e.g. screening or variance decomposition, a detailed review is given in [6]). As this holds for the analysed directional control valve different sensitivity measures where calculated and compared. The results are shown in figure 6.

In the direct comparison of figure 6, each of the different sensitivity measures identify the same noise factors as significant. One should keep in mind that Sobol’s main effects sum up to one, while every correlation coefficient and every regression coefficient can take values in the interval $-1$ to $1$. Since only variance based sensitivity measures quantify the ratio of the noise factor variation to the variation of the key product characteristics, they seem to filter the data better than correlation or regression measures which measure linearity/monotonicity and are
FIGURE 5. Analysis of Failure Mode: threshold level between operating (labelled ‘1’) and faulty (labelled ‘0’) samples and comparison of switching process for one operating and one faulty sample

FIGURE 6. Comparison of different sensitivity measures calculated for a directional control valve ($S_T$ and $S_W$: main and total sensitivity indices from Sobol’s method; SRC: standardized regression coefficient; SRRC: standardized rank regression coefficient; $r$: Pearson’s correlation coefficient; $\rho$: Spearman’s rank correlation)

proportionality factors, respectively. This becomes evident by the sensitivities of $t_{S2}$ where Sobol’s indices only identify two significant noise factors whereas the other measures identify up to six significant noise factors.

There is nearly no difference between the correlation coefficients and the regression coefficients. That indicates not only monotonicity, but also a linear relationship within the range of tolerances. Furthermore, there seems to be no strong interactions between noise factors due to the fact that Sobol’s main and total effects show nearly no differences. The only exception is the switching time $t_{S2}$ whose cause of failure was already discussed as the interaction between spring stiffness and friction force.

Variance based sensitivity measures require no restrictions about the linearity of the system’s transfer function, but their calculation is far more computationally expansive. This is especially relevant when a large amount of noise factors are taken into account for the analysis. In this case surrogate models were used to gain convergence within a bearable computational timeframe.

Concerning the valve’s key product characteristics strong dependencies between the switching time $t_{A2}$ of the pilot stage and the noise factors related to the supply voltage, the parasitic air gaps, and the stiffness of the armature return spring are visible. Most influence to the switching time of the main stage $t_{S2}$ comes from the noise factors for the friction force and the stiffness of the spool return spring. The variations of parasitic air gaps, the supply voltage and the magnetic permeability of the yoke’s material dominate the variations of the force reserves for release and hold of the armature.

TOLERANCE SYNTHESIS

Tolerance synthesis, on the other hand, aims at the determination of an optimal tolerance concept resulting in the compliance of the demanded tolerances for the key product characteristics. It is possible to determine tolerances for the individual part and material properties based on the calculated sensitivity measures. But in doing so, there is no
guaranty of finding the most cost-efficient tolerance concept. Considering tolerance synthesis as an optimization problem allows for the utilization of a broad variety of established numerical optimization tools which results in a substitution of heuristics by algorithms, as explained in [7].

\[ T_B \]

**T**

B

Pareto

front

Min

F

= 

\[ f_1 \]

\[ \vdots \]

\[ f_n \]

constraints:

objective:
sampling

T

u

T

B
tolerance analysis
metamodel
comparative cost calculation numerical optimization
TB

vQ,C
nominal
value

**FIGURE 7.** Chosen procedure for tolerance synthesis: numerical optimization of tolerance concept \( T_B \) by iterative calculation of dispersion measures and comparative costs

Figure 7 depicts the chosen approach for the tolerance synthesis of directional control valves. For realizing this concept, one needs to pay attention to the following three aspects:

**Tolerance-Cost-Function** In order to determine the total costs associated with a tolerance concept, the specific manufacturing costs of each feature within the given individual tolerance range need to be known. Tolerance-cost functions typically increase strongly the smaller the tolerance range is. Many authors use reciprocal potential functions like the one shown in the middle of figure 7 as a mathematical description for that relationship. The main difficulty lies in determining suitable regression parameters \( a, b \) and \( k \) for the specific manufacturing processes for the considered feature. There are almost no publicly available data sets as the data is company specific and usually underlying an obligation of confidentiality. An exception is a data set for metal working found in [8]. It is based on a study of the U.S. Army from the 1940s which seems quite outdated. But by being formulated as relative costs, this data set allows for the testing of the chosen approach for tolerance analysis. In addition, the tolerance-cost-function for standard parts can be reconstructed from available grades and the corresponding purchasing price.

**Adapted tolerance analysis** During the optimization process the resulting deviations of the key product characteristics associated to each specific tolerance concept need to be determined. Almost all available methods for propagating imperfect information about the noise factors through system models share the necessity for running a huge number of model evaluations. Surrogate model can reduce the calculation time strongly but compared to their use in sensitivity analysis an extended range of validity is required in the case of the algorithm is trying to expand the tolerances beyond their initial range. By doing so usually a significant reduction of the goodness of fit can be observed. Two possible countermeasures have proven themselves in practice: a) the restriction to the operative valve samples during surrogate model generation by utilising an additional pattern recognition neural network and b) a sensitivity based selection of the noise factors whose range of validity is extended for surrogate model generation.

**Numerical optimization** Furthermore an suitable optimization method needs to be selected with regard to the number of noise factors and the characteristics of the objective function. The restriction expressing the compliance to the tolerable deviations of the key product characteristics can be transformed into an additional objective function. This could be the minimization of a weighted sum of a measure of dispersion for each key product characteristic, e.g. the quartile dispersion coefficients which are robust against outliers. This is particularly advantageous because one solution can then be chosen according to the desired cost or precision demands out of a set of so called pareto optimal solutions afterwards. Within this special solution set an objective function cannot be increased without decreasing another objective function. Unfortunately, there is no gradient information available for this multicriteria objective function which results in the necessity to calculate them numerically. This is prohibitive, especially in the case of many noise factors. To avoid this, a stochastic optimization algorithms called ‘Non-dominated Sorting Genetic Algorithm’ (NSGA-II) was chosen for the cross-domain tolerance synthesis.
of directional control valves which is also able to identify not only local minima but also global ones.

Three different resulting tolerance concepts are compared to the initial design in figure 8. They are highlighted in figure 7 with the same colours. The pink dot marks the initial tolerance concept. Compared to the concept marked in red, only a small reduction of costs can be achieved at the expense of the resulting deviations for the key product characteristics. On the other hand, the concept marked in blue reduces the deviations by half, but for double the price. A compromise solution is marked by the green colour. Compared to the initial design, costs, as well as the resulting deviations, can be significantly reduced. In this case it is possible to extend the allowed tolerances for most of the system’s parameters. Only a couple of parameters with especially high sensitivity measures are tightened to ensure the precision of the valve characteristics. The tolerance restriction is strongly coupled to the tolerance-cost-function which is extremely important for the results.

SUMMARY AND OUTLOOK

This paper demonstrates a method for computational tolerance analysis and tolerance synthesis for a directional control valve. Questions like identifying noise factors to be toleranced, a mathematical representation of these tolerances, and a way to propagate their impact on the key product characteristics through simulation models, were all addressed. A comparison of different approaches to describe the noise factors, as well as further application to other valve types (or even to complex fluid power systems) could be the focus of future research.

ACKNOWLEDGMENTS

The presented research activities are part of the project ‘Toleranzanalyse für den Ventilentwurf’ (Ref. No. 702850). The authors would like to thank the Fluid Power Research Fund of the VDMA for the funding and support.

REFERENCES


THEORETICAL ANALYSIS ON SPOOL STUCK POSSIBILITIES OF ROTARY DIRECT DRIVE PRESSURE CONTROL SERVO VALVE

Yaobao YIN *, Feiyan XIA *, Liang LU *,** , Jiayang YUAN *, Shengrong GUO ***

* School of Mechanical Engineering, Tongji University, Shanghai 200092, China
(E-mail: luliang829@tongji.edu.cn)
** State Key Laboratory of Fluid Power & Mechatronic Systems, Hangzhou 310027, China
*** Aviation Key Laboratory of Science and Technology on Aero Electromechanical System Integration, Nanjing 210061, China

Abstract. In order to figure out the problem that, a rotary direct drive electro-hydraulic pressure control servo valve output pressure curve appears flat and overshoot under the high oil supply pressure, a mathematical model is established by means of SIMULINK to analyze the force of each component and the movement of the spool. The theoretical analysis shows that too small eccentric distance from drive ball center to spool axis will cause the spool to be permanent stuck. However, the real cause of the phenomenon that the output pressure appears multiple times of flat and overshoot as the experimental curve shows may be because of the size and direction of the radial unbalance force constantly changes during the movement of the spool. And this unbalance force results in the spool attach to and divorce of the wall of the valve sleeve discontinuously.

Keywords: rotary direct drive, eccentricity parameter, spool stuck, mathematical model, force and movement

INTRODUCTION

Servo valves appeared in the 1950s. MOOG invented the first two-stage electro-hydraulic servo valve, used the nozzle baffle structure which has high power density as a front hydraulic amplifier to drive spool move [1-2]. The hydraulic amplifier in the two-stage electrohydraulic servo valve converts the weak current signal into a pressure signal and the spool is used to output flow or pressure. However, the two-stage valve has the problem of leakage and its anti-pollution ability is poor. With the motor drive capacity to enhance, direct drive servo valve has been developed, mostly are straight drive type. Initially, the direct drive valves are only used when the output flow is small. Such as D633-type linear direct-drive valve developed by MOOG Company, the dynamic and static response characteristics of the valve are pretty well, but because of large size of linear torque motor, the valve is not suitable for use in limited space. In the late 1980s, a number of patents on rotary direct drive servo valves began to emerge mainly focused on the mechanical and electrical conversion device innovation [3-7]. These innovative valves need to be verified whether have practical application value by further experiments and theories. In this paper, the experimental and theoretical research on the rotary direct drive electro-hydraulic pressure control servo valve [8-9] driven by the ball connected to a motor is carried out. Experimental tests found that under high oil supply pressure, the output pressure appears multiple flat and overshoot. This paper considers several possible theoretical assumptions, establishes mathematical model combined with the dynamic movement of the spool, gives the physical explanation and mathematical description of this practical problem.

STRUCTURE AND PRINCIPLE

The rotary direct drive electro-hydraulic pressure control servo valve (RDDPV) constitute by motor, drive interface, spool, valve sleeve, valve body and pressure closed-loop feedback control circuit. It sets motor axis and spool axis vertically to save space. The basic structure is shown in FIGURE 1. The rotational motion of the motor is converted into a linear reciprocating motion of the spool by means of a conversion element. And the sliding of spool control the valve ports opening to achieve the output pressure. The main components of the valve are the drive interface and spool structure.

The drive interface consists of a ball and a cylindrical hole that both have same radius. The ball is welded to the output shaft of the motor and its center away from the motor axis at the distance of \( e_0 \). The cylindrical hole and the spool are made as a whole. At the time of installation, set the center of the ball at \( h_0 \) above the spool axis. Such as FIGURE 2.

When command current input, the ball at the end of the motor shaft driven by the motor and do circular motion. The driving force of the ball on the cylindrical hole can be decomposed into X and Z directions. The X-acting...
force drives the spool to slide along the axis, as to control valve ports open and close, and output the corresponding pressure. The force generated by the Z-acting drives the spool to rotate about its axis in case the freedom of spool movement limited.

FIGURE 1. The basic structure of the valve

FIGURE 2. The drive interface of RDDPV

FIGURE 3. Spool and valve sleeve structure of RDDPV

The valve is used to output pressure proportional to input command current. The spool uses two shoulders three-way structure, as shown in FIGURE 3. Spool sliding form a joint throttling effect between the oil supply port and the return port. The output port is placed between the oil supply port and return port, and the output pressure adjustment from the spool sliding displacement.

The valve uses the control method of closed-loop output pressure feedback. The feedback signal is the error of the theoretical command value corresponding to the actual output pressure. The signal will be proportioned and integrated (PI), and then output the appropriate control signal to drive the motor. The control method is conventional and easy to build analog circuits.
PROBLEM DESCRIPTION

For the valve, the output pressure characteristics were tested at different oil supply pressures. FIGURE 4 shows the test results at oil supply pressures of 12MPa and 21MPa. The input command is a 0 to 3.2mA to 0 triangular current signals. A solid piezo resistive pressure sensor with the test range of 0 to 25 MPa is used to measure the output pressure and feedback to control the motor drive voltage. The test results show that the output pressure characteristics of the valve are good at low oil supply pressure. It indicates that the valve has practical value.

![FIGURE 4. The experimental output pressure curve](image)

The results show that when the oil supply pressure is high, the output pressure curve will appear flat and overshoot. The experimental curve has the following characteristics:
- there are multiple flat for output pressure
- there will be pressure overshoot after the pressure flat
- at the beginning of the spool closure, pressure fluctuation is small

Theoretically, the output pressure increases with the valve port opening, it can be deduced that the stage of the output pressure flat corresponds to the spool displacement of 0, that is, the spool is stuck during the sliding process.

ANALYSIS ON STUCK POSSIBILITIES

Reasons for the spool stuck may be summarized as two categories. One is improper choose of structure parameters. If the distance from driving ball center to the spool axis is too small, the motor rotation will be limited and resulting in spool stuck. The other is mutative resistance. The size and direction of the spool sliding resistance changing will cause spool speed instability. The resistances are mainly due to flow force and the potential contact friction between the spool and sleeve wall during the spool sliding process. So, firstly analyze the influence on the output pressure by flow force, and then consider the contact friction between spool and sleeve wall due to floating and inclining of the spool under the action of eccentric driving force.

**Improper structure parameter**

On analysis of the spool, the relationship between the output pressure and the spool sliding displacement can be expressed by the following equation

\[
C_d \pi D x_1 \sqrt{2 \frac{p_e - p_L}{\rho}} - C_d \pi D (U - x_0) \sqrt{2 \frac{p_L}{\rho}} = \frac{V}{E} \frac{dp_L}{dt} \tag{1}
\]

In general, to the steady process, the effects of oil compressibility will not be taken into account, which is

\[
\frac{V}{E} \frac{dp_L}{dt} = 0 \tag{2}
\]

The formula (1) can be written as
Where $p_s$ is the oil supply pressure and $p_L$ is the output pressure, $V$ is the volume of the load capacity, $E$ is the oil volume elastic modulus, $\rho$ is the oil density, $U$ is the pre-opening of the valve port, $D$ is the spool diameter, $C_d$ is the flow coefficient and $x_v$ is the spool displacement.

On kinematic analysis of the drive interface, there are

$$x_v = e_0 \sin \alpha$$

(4)

$$\beta_v = \arcsin \frac{e_0(1 - \cos \alpha)}{h_0 + y_v + \gamma_v l}$$

(5)

The drive ball is connected by a shaft has certain diameter to the motor end, when the shaft collides with the inner wall of the cylindrical hole, the spool’s rotation will be restricted to a limited position. When installed, if the distance $h_0$ that drive ball center away from the spool axis is too small, the cylindrical hole will not rotate freely and the motor corner will be limited and the spool will be stuck.

As the spool limited in the valve sleeve, then

$$y_v + \gamma_v l \leq \delta$$

(6)

In conjunction with the formula (3), the initial distance $h_0$ must be met the requirement

$$h_0 \geq \frac{e_0 \left(1 - \cos \left[\arcsin \frac{\sqrt{p_s(p_s - p_L)} - p_L}{e_0(p_s - 2p_L)U}\right]\right)}{\sin \beta_0} - \delta$$

(7)

Where $l$ is the spool structure parameter, represents the distance from the spool centroid to the inner wall of the cylindrical hole, $\delta$ is the initial gap between the spool shoulder and the valve sleeve, $\alpha$ is the motor rotation, $y_v$ is the floating displacement of the spool with the center of mass, $\gamma_v$ is the tilting angle of the spool around the center of mass, $\beta_v$ is the rotation angle of the spool around the axis. $\beta_0$ is the limited angle of spool rotation.

**FIGURE 5.** Spool stuck under an unreasonable $h_0$

As shown in **FIGURE 5**, a spool displacement curve is given that $h_0$ does not meet the formula (7). The input signal is 0~3.2mA electric current. When the motor is limited, the drive current continues to increase, but the spool stuck, and it can’t re-slide.
Mutative resistance

The resistances of the spool during sliding are mainly flow force and possible friction caused by contact of spool and sleeve wall. Firstly, assume that the spool is not contact with the sleeve wall during sliding process and analyze the influence of the output pressure by the changing flow force. If the model cannot reproduce the characteristics in the tested output pressure curve, consider to add the radial force caused by the tilt spool in the valve sleeve into the mathematical model. Analyze the movement state of the spool under the radial force. And if the spool contacts the valve sleeve, the frictional resistance is added to the model.

Flow force

When the spool is not in contact with the wall during the sliding process, the main resistance is the flow force. Flow force contains steady and transient flow force. Transient flow force value is small and can be ignored. Steady flow force can be expressed by the following formula:

\[
F_s = 2C_vC_dW \cos \theta x_v (p_s - p_L) - 2C_vC_dW \cos \theta (U - x_v) p_L
\]

Where \( \theta \) is the jet angle, \( C_v \) is the velocity coefficient, \( W \) is the area gradient, \( W = \pi D \).

In the working range, set oil supply pressure in the model to 21MPa, the maximum output pressure is 8 MPa. And the input command is 0 to 3.2mA to 0 triangular current signals. Choose left as the positive direction for spool displacement and force. It can be seen from FIGURE 6 that at the period of valve port opening, the spool move to left and the flow force towards right act as resistance. And at the period of valve port closing, the spool move to right and the flow force still towards right but it act as a part of driving force. The driving force curve shows that when the spool is commutated, as the flow force changes from the resistance into driving force at a sudden time, results in driving force shock. However, from the spool displacement curve can be seen, the changing of flow power is not enough to cause the spool stuck, the output pressure characteristics are good, as shown in FIGURE 6.
Contact friction

Restricted by the drive principle, the driving ball center must deviates from the spool axis. The overturning moment resulting from the driving force off the axis causes the spool to tilt an angle, and it will produce an unbalance force $F_c$ in the radial direction of the spool. The unbalance force not only changes the tilting angle but also drives the spool float or sink at the radial direction of the spool. Integrated spool tilting and floating, when stuck, the spool may have two states in the valve sleeve. As Figure 7 (a) shows is the spool fully tilt state and the Figure 7 (b) shows is the spool attach to the wall of the sleeve.

\[
F_c = \frac{2\pi \delta LR \Delta p (e_1 - e_2)}{(e_1 + e_2 - 2 \gamma_v)^2} \left\{ 1 - \frac{1}{\sqrt{1 - \frac{(e_1 + e_2 - 2 \gamma_v)^2}{2\delta}}} \right\} \quad (9)
\]

For the oil supply side of the valve shoulder

\[
\begin{align*}
  e_1 &= (a - L) \gamma_v \\
  e_2 &= a \gamma_v
\end{align*} \quad (10)
\]

For the oil return side of the valve shoulder, because $\Delta p = 0$, then $F_c = 0$.

Where $R$ is the shoulder radius, $\Delta p$ is the differential pressure across the shoulder. $e_1$ and $e_2$ represent the entrance and the outlet gap of the spool shoulder caused by the tilting, $L$ is the shoulder width, $a$ is the distance from the spool left end to the center of mass.

If during the spool sliding process, meet the formula (11)

\[
W_{\rho \max}a \leq F_c h_0 \quad (11)
\]

That is the overturning moment caused by driving force is greater than the recovery moment of the maximum radial unbalance force, the spool will finally be fully tilted in the valve sleeve as the Figure 7 (a) shows and under this circumstance the spool will subjected to a sudden increase resistance and slow down or even stop. The driving force continues to increasing. The overturning moment of the driving force is always greater than the recovery torque generated by the maximum radial unbalance force. The spool can’t release from the fully tilted state and can’t re-slide. Therefore, during the design process, on the basis of satisfying formula 7, $h_0$ cannot be too large. Excessive $h_0$ will increase the tendency of the spool to fully tilt. Proper structural parameters $h_0$ must be choose to avoid the situation, so that

\[
h_0 < \frac{W_{\rho \max}a}{T e \cos \alpha} \quad (12)
\]

Where, $T$ is the maximum torque that the motor can provide. Although the actual output pressure curve shows that the spool will be stuck but it can re-slide. So it can determine the spool in the valve sleeve is not completely tilted stuck.

If during the spool sliding process, meet the formula (13)
\[ W_{p_{\text{max}}} \alpha \geq F_{sv}h_0 \tag{13} \]

That is the recovery moment of the maximum radial unbalance force is greater than the overturning moment caused by the driving force. As the spool gravity is small and can be ignored, in the radial, the unbalance force is the main force. Because of the radial unbalance force, the spool will float and eventually be possible to attach to the wall of the valve sleeve, as FIGURE 7 (b) shows. If the sleeve wall is rough or even has burr, when the spool is fully affixed to the wall of the valve sleeve, it will be subject to sudden friction effect. The spool will slow down and even stuck.

And under the state of stuck, because of the closed loop feedback of output pressure, driving force will be continue to increase and once the spool has a small inclination under the overturning moment, the radial force reappears and the spool will be disengaged from the wall of the sleeve. Once the driving force increases to larger than resistance when the spool stuck, the spool will have a certain speed and inclination. The spool will releases from the state of stuck with the help of radial unbalance force by the time slides. Once the spool not stuck, it accelerates. This process produces a large pressure overshoot.

When \( h_0 \) meets the requirement of formula (7) and (12), consider the radial unbalanced force of formula (9), simulation results show that the state of spool movements switches between ‘attach to wall and friction’ and ‘divorce from wall and slides’.

In FIGURE 8, \( H_1, H_2, H_3 \) and \( H_4 \) are the distance between the four vertexes and the valve sleeve. Through the output of these four parameters, the movement of spool in the radial direction can be seen in the valve. The gap between the spool shoulders and the valve sleeve in the model is 3 μm. The horizontal lines segment in the figure indicate that the spool attaches to the wall and the rest indicate that the spool tilts and floats in the valve sleeve. FIGURE 8 shows that the movement of the spool in the valve sleeve is discontinuously attaching to wall and shaking up and down.

FIGURE 8. Spool movement state

FIGURE 9. The output pressure curve in the model
FIGURE 9 shows the theoretical output pressure curve consistent with the experimental conditions. Although it is smoother than the experimental curve, it tends to reflect the problem anyway. It can be seen that pressure fluctuation in the valve opening stage is greater than the closing stage. And there will be pressure overshoot after tending to flat. The pressure flat and the overshoot of the theoretical curve are smaller may be due to oil damping and frictional resistance in the actual process are larger than the theoretical setting value.

CONCLUSIONS

1) Experimental tests have been down on a ball-type rotary direct drive electro-hydraulic pressure control servo valve under different oil pressure, and output pressure curves of the valve is obtained. The results show that the output pressure appears multiple flat and overshoot under high oil pressure. According to the relationship between the output pressure and the displacement of the spool, it is inferred that the spool stuck many times during the sliding process.

2) The possible causes of spool stuck are improper parameter selection at the drive interface and sudden change of the spool sliding resistances. Among them, if the distance from driving ball centre to spool axis is too small, the spool will permanently stuck. And although the size and nature of the flow force changes, but because the value is small, it is not the main reason leading to spool stuck. As the driving ball centre deviates from the spool axis, the spool is tilted by the overturning moment during the sliding process. The tilting spool will float in the valve sleeve under the effect of radial unbalance force. Through the analysis, it can be seen that when the spool completely tilted, it cannot release from the stuck state, and the spool will be permanently stuck. However, the discontinuously ‘attach to sleeve wall and friction’ model can well reflect the characteristics of the problem curve in the experiment process. Despite the magnitude of the model calculation results cannot be consistent with the experimental output curve, but the trend is basically the same. Theoretical output pressure curve’s flat and overshoot smaller than that in the experimental curve may be due to the actual oil damping and frictional resistance are greater than the setting value in the model.

ACKNOWLEDGMENTS

This work is supported by the National Natural Science Foundation of China (No. 51605333 and No. 51475332), the Open Foundation of the State Key Laboratory of Fluid Power and Mechatronic Systems (No. GZKF-201518), the Program for Young Excellent Talents in Tongji University (No. 2014KJ033). Besides, particular thanks to Aviation Key Laboratory of science and Technology on Aero Electromechanical System Integration for their technical and financial support.

REFERENCES

VALIDATION OF AN ENHANCED MODEL OF STEADY-STATE FLOW FORCES FOR SPOOL VALVES

Patrik BORDOVSKY*, Hubertus MURRENHOFF*

*Institute for Fluid Power Drives and Controls (IFAS)
RWTH Aachen University
Steinbachstraße 53, 52070 Aachen, Germany
(E-mail: Patrik.Bordovsky@ifas.rwth-aachen.de)

Abstract. One of the valve types often used in hydraulics is the spool-type valve. A typical example of a spool valve is a directional valve, which controls the flow and its direction. In order to move the spool, an actuator is needed to overcome forces acting on the spool. Beside spring and friction forces, flow forces are the most relevant. They influence the static and dynamic performance of valves. There are steady-state and transient flow forces. In this paper, the validation of an enhanced model for the calculation of steady-state flow forces in spool valves for a defined flow is described. Both the momentum influx and the efflux are considered in the model. In addition, it was found that the accuracy of the calculation of steady-state flow forces can be improved by using variable quantities, such as the inlet and outlet flow angles and the discharge coefficient, instead of constants.

Keywords: Spool Valve, Steady-State Flow Force, Momentum Theory, Analytical Calculation

INTRODUCTION

A research work was conducted to develop a mathematical model for the estimation of steady-state flow forces (hereinafter referred to as flow forces). The origin of the flow forces can be explained by the theory of momentum conservation. This approach was used by several researchers to derive a suitable model for the calculation of flow forces acting on square-edged spools. Lee [1] introduced the first model in 1952. He considered the outflow from a 2/2-way square-land spool (shown in fig. 1) and assumed a nonviscous and incompressible fluid and a two-dimensional, quasi-irrotational flow.

\[ F_{Fl, \text{I}} = 2 \alpha_D \cos \varepsilon \Delta p \cos \varepsilon \]

where \( \alpha_D \) is the discharge coefficient, \( w \) the area gradient, \( x \) the spool stroke, \( \Delta p \) the pressure difference across the orifice and \( \varepsilon \) the jet angle. Lee assumed an inlet jet angle of 90°, an outlet jet angle of 69°, and a jet velocity between the points \( a \) and \( b \), i.e., at the narrowest area called “vena contracta”. Lee pointed out that the assumption of quasi-irrotational flow is generally not fulfilled. Based on Lee’s assumptions and jet-angle values, Backé [2] presented an analytical model expressed by eq. (2) in 1962

\[ F_{Fl, \text{II}} = \rho Qv \cos \varepsilon \]

FIGURE 1. Valve geometry considered by Lee

For the control volume (CV), respectively the control surface (CS) depicted in fig. 1, Lee derived an analytical model given by eq. (1)
where \( \rho \) is the fluid density and \( Q \) the flow rate.

However, Backé considered the jet velocity on the metering-out area \( A = \pi Dx \) and a valve body with a circumferential groove on the outlet as illustrated in fig. 2.

\[ F_{III} = 2 \alpha_D C_w \Delta p \cos \epsilon \cos \theta \]

Merritt [3] presented an analytical model given in eq. (3), which is similar to that from Lee in 1967.

\[ F_{IV} = \rho Q \cos \epsilon \sin \epsilon \]

Merritt also considered the control volume enclosed by the vena contracta. Compared to Lee, he expressed the discharge coefficient as a product of a dimensionless contraction coefficient \( C_c \) and a dimensionless velocity coefficient \( C_v \approx 0.98 \). Further researchers, e.g. Tatar [4], Feigel [5], Kipping [6], formulated slightly different analytical models compared to the above-mentioned ones. In 2005, Schuster [7] considered an average axial jet velocity \( v_{j,x} = v_1 \cos \epsilon \sin \epsilon \) and derived an analytical model given in eq. (4).

\[ F_\Delta = \pi \rho \Delta \epsilon \cos \cos \Delta \]

Murrenhoff [8] assumes the metering area \( A = \pi Dx \) and approximated the jet velocity according to eq. (5).

\[ v_1 = \frac{Q}{A \sin \epsilon} \]

Consequently, Murrenhoff described the flow forces by eq. (6)

\[ F_{IV} = 2 \alpha_D \frac{\cos \epsilon}{\sin \epsilon} \pi Dx \Delta p \]

As it can be seen, the flow force was approximated by different analytical models. All of them have in common that they considered only the greater momentum flux, i.e., the influx for the inlet-throttling, respectively the efflux for the outlet-throttling.

Within this study, both the momentum influx and the efflux are considered for the analytical calculation of flow forces in spool valves for the inflow. Moreover, the inlet and the outlet flow angles, presented in [9], are used.

The enhanced model is validated for different spool geometries based on measurements and simulations.

**ANALYTICAL MODEL OF FLOW FORCES**

In mechanics, Newton’s second law can be used for the analysis of forces acting on a system. As a “system”, an arbitrary mass quantity of fixed identity is defined. The term “surroundings” describes everything outside of the system. The interface between the system and the surroundings is named as “boundary”, [10]

In order to estimate the system net force \( \vec{F}_{sys} \) eq. (7) is set up as the time derivative of momentum \( \vec{I} \) for fluid elements with infinitely small mass \( dm \) and with the velocity \( \vec{v} \).
In fluid mechanics, a definite volume, called a control volume, is usually considered instead of the system to investigate the forces between the system and the surroundings. The boundary of the control volume is referred to as control surface. Figure 3 shows the control volume considered in this paper.

In order to apply the system laws for the control volume, a conversion has to be done according to the Reynolds transport theorem as expressed by eq. (8)

\[
\dot{F}_{\text{sys}} = \frac{d \dot{I}}{dt} = \frac{d I}{dt} - \int_{syst} v \, dm = \frac{d}{dt} \int_{syst} v \rho \, dV
\]

(7)

where \( \hat{n} \) is the normal unit vector. The quantity \( \dot{F}_{\text{sys}} \) represents the sum of the surface and the body forces. The first term on the right-hand side of eq. (8) describes the transient forces, while the second term expresses the steady-state forces. When neglecting the transient forces and assuming the one-dimensional inlet and the outlet, eq. (8) can be simplified to eq. (9)

\[
\dot{F}_{\text{sys}} = \dot{m}_i \dot{v}_i - \dot{m}_o \dot{v}_o
\]

(9)

where \( \dot{m}_i \) is the mass flow rate on the inlet \((i = 1)\), respectively on the outlet \((i = 2)\). Eq. (9) can be applied for the calculation of flow forces in spool valves. For this purpose, only the axial forces are considered and the following assumptions are made. First, the fluid is incompressible, second the spool is ideally square-edged, third the control volume is an ideal rotationally symmetric hollow cylinder, and fourth the radial clearance between the spool and the sleeve is neglected. This results in eq. (10)

\[
F_{\text{sys},x} = \dot{m}_i (v_{2,x} - v_{1,x})
\]

(10)

where \( v_{i,x} \) is the axial velocity. From the practical point of view, the net axial force \( F_{\text{sys},x} \) acting on the spool has to be estimated to design the valve actuator. Beside other forces, the flow forces \( F_{\delta} \) are usually the most dominant disturbance forces. As it is apparent from eq. (10), the average axial velocities have to be approximated to estimate the flow forces. The derivation of the approximation is introduced in [9]. Within this paper, the following formulas are considered for the inlet axial velocity (eq. (11)), respectively for the outlet axial velocity (eq. (12)).

\[
v_{1,x} = -\alpha_p \sqrt{\frac{2 \Delta \rho}{\rho} \cot \varepsilon_i}
\]

(11)
The mass flow rate can be calculated according to eq. (13).

$$\dot{m} = \rho \alpha_p A_1 \sqrt{\frac{2 \Delta p}{\rho}}$$  \hspace{1cm} (13)

By inserting eq. (11 - 13) into eq. (10), an enhanced analytical model of the flow forces is derived as given by eq. (14).

$$F_{pl} = -2 \alpha_p^2 A_1 \Delta p \left( \cot \varepsilon_1 + \frac{A_1}{A_2} \cot \varepsilon_2 \right)$$  \hspace{1cm} (14)

This formula can generally be applied for arbitrary spool geometries when considering the above-mentioned assumptions. This model is validated within this paper with the help of measurement and CFD simulation results.

**METHODS**

The enhanced analytical model is validated with the measurement and CFD simulation results. Therefore, both methods are explained in the following subchapters.

**Measurement**

The flow forces were measured on a spool-type test valve, which is a 2/2-way proportional directional valve. A detailed description of the test valve, respectively of the measurement method, can be found in [9], and [11]. Hence, only a short summary is provided in this paper.

The hydraulic circuit of the experimental set-up is shown in fig. 4. The flow forces of the test valve (TV) are measured. The test valve is actuated by a displacement unit (DU) with the velocity of approximately 0.01 mm/s. The pressure on the outlet of the test valve is kept at 100 bar by a pressure relief valve (PRV). The pressure difference across the test valve is controlled by the servovalve (SV) with a PI controller. The measurements were carried out with the oil HLP46 at an oil temperature of 60°C.

**FIGURE 4.** Simplified hydraulic circuit of the test rig

For the flow-force determination from the measurements, it is assumed that the friction force has the same magnitude independent of the spool motion’s direction. Consequently, the flow force can be determined as the average axial force from the net force measured during the opening of the test valve and from the net force measured during the opposite motion. Figure 5 illustrates this principle, which utilises the fact that the direction...
of friction force depends on the direction of the spool motion while the direction of the flow force is independent of the direction of the spool motion.

\[ F = F_{Fl} \]

**FIGURE 5.** Principle of the flow-force determination from measurements

The flow forces were measured on the spool geometries shown in fig. 6. The bevel angle of the spool BE equals 45°. The spool CG has four cylindrical grooves which are drilled perpendicularly to the spool axis. The radius of the cylindrical grooves is 2.5 mm.

**FIGURE 6.** Spool geometries used for measurements

**CFD Simulation**

The CFD simulations used for the validation of the enhanced analytical model were described in detail in [11]. Hence, a necessary summary is presented in this paper. The steady, three-dimensional, isothermal, incompressible CFD simulations were solved in ANSYS CFX 17.0. The radial clearance and hence the friction between the sleeve and the spool were neglected in this study. All spool geometries (see fig. 6) were simulated at discrete openings and at discrete pressure differentials. A full fluid domain with two inlets and two outlets was considered, although the valve geometry has two symmetry planes. At the inlets, a mass flow rate was defined. The static pressure of 100 bar was set at the outlets to avoid cavitation. These boundary conditions were adapted from measurements.

The inflow and the outflow angles were evaluated from the CFD simulations according to eq. (15)

\[ \varepsilon_{i,v} = \arccot \frac{v_{i,v}}{v_{i,r}} \]  

where \( v_{i,v} \) is the axial velocity and \( v_{i,r} \) the radial velocity according to fig. 3.

**RESULTS AND DISCUSSION**

From eq. 14 it is apparent that there are three unknowns, i.e., the discharge coefficient, as well as the inflow and the outflow angles, which are difficult to estimate. Usually, the following values are considered for the calculation of the flow forces: \( \alpha_{D,ref} = 0.7 \), \( \varepsilon_{1,ref} = 69^\circ \) and \( \varepsilon_{2,ref} = 90^\circ \). These values are considered as reference values for the following exemplary calculation of the flow force. The objective of this example is to analyse the impact of these three variables on the calculated magnitude of the flow force. For this purpose, the square-edged spool with a stroke \( x = 0.5 \) mm, a spool diameter \( D = 10 \) mm and an outlet-metering area \( A_2 = 219.91 \) mm\(^2\) is considered. The discharge coefficient is varied from 0.5 to 0.9, the inflow angle from 45 to 75°, and the outflow angle from 80 to 150°.

For the sensitivity analysis, eq. 14 is rearranged to eq. 16 by the introduction of the coefficient \( C_1 = 2A_2\Delta p \) and the coefficient \( C_2 \), which is the term in the brackets from eq. 14.
Table 1 shows the values of the coefficient $C_2$ for different inflow and the outflow angles. It is apparent that the inflow angle has the predominant influence on the coefficient $C_2$. This is plausible since the outflow velocity is much smaller than the inflow velocity for the studied geometry.

**TABLE 1. Values of coefficient $C_2$ for different inflow and outflow angles**

<table>
<thead>
<tr>
<th>$C_2$</th>
<th>$\varepsilon_2$ [°]</th>
<th>80</th>
<th>90</th>
<th>100</th>
<th>110</th>
<th>120</th>
<th>130</th>
<th>140</th>
<th>150</th>
</tr>
</thead>
<tbody>
<tr>
<td>45</td>
<td>1.01</td>
<td>1.00</td>
<td>0.99</td>
<td>0.97</td>
<td>0.96</td>
<td>0.94</td>
<td>0.91</td>
<td>0.88</td>
<td></td>
</tr>
<tr>
<td>50</td>
<td>0.85</td>
<td>0.84</td>
<td>0.83</td>
<td>0.81</td>
<td>0.80</td>
<td>0.78</td>
<td>0.75</td>
<td>0.72</td>
<td></td>
</tr>
<tr>
<td>55</td>
<td>0.71</td>
<td>0.70</td>
<td>0.69</td>
<td>0.67</td>
<td>0.66</td>
<td>0.64</td>
<td>0.62</td>
<td>0.58</td>
<td></td>
</tr>
<tr>
<td>60</td>
<td>0.59</td>
<td>0.58</td>
<td>0.56</td>
<td>0.55</td>
<td>0.54</td>
<td>0.52</td>
<td>0.49</td>
<td>0.45</td>
<td></td>
</tr>
<tr>
<td>65</td>
<td>0.48</td>
<td>0.47</td>
<td>0.45</td>
<td>0.44</td>
<td>0.43</td>
<td>0.41</td>
<td>0.38</td>
<td>0.34</td>
<td></td>
</tr>
<tr>
<td>70</td>
<td>0.38</td>
<td>0.36</td>
<td>0.35</td>
<td>0.34</td>
<td>0.32</td>
<td>0.30</td>
<td>0.28</td>
<td>0.24</td>
<td></td>
</tr>
<tr>
<td>75</td>
<td>0.28</td>
<td>0.27</td>
<td>0.26</td>
<td>0.24</td>
<td>0.23</td>
<td>0.21</td>
<td>0.18</td>
<td>0.14</td>
<td></td>
</tr>
</tbody>
</table>

In order to indicate the impacts of the three unknowns on the magnitude of flow force, the minimum and the maximum relative errors $E_R$ [%] are calculated according to eq. 17. The errors are related to the reference values of $\alpha_D, \varepsilon_1, \text{and} \varepsilon_2$.

$$E_R = \left(\frac{-\alpha_D^2 C_2 \alpha_D^2 C_2 \text{ref}}{-\alpha_D^2 C_2 \text{ref}}\right) \cdot 100$$

For the minimum values $\alpha_D = 0.5$ [-] and $C_2 = 0.14$ [-], the minimum relative error $E_{R,\text{min}}$ is $-79.8$ %. The maximum relative error $E_{R,\text{max}}$ of 359.9 % was calculated for the maximum values $\alpha_D = 0.9$ [-] and $C_2 = 1.01$ [-]. The limit values of the relative errors emphasise how the assumed values of $\alpha_D$, $\varepsilon_1$, and $\varepsilon_2$ impact the magnitude of the calculated flow force. The calculation was done only for the square-edged spool. However, similar results can be expected for the other spool geometries. It is apparent from tab. 1 that the inflow angles significantly influence the magnitude of the calculated flow force. Since the second power of the discharge coefficient is calculated, this coefficient should also be estimated as accurate as possible.

Considering the spool SE20 and the pressure differential of 30 bar, Fig. 7 shows the comparison of the flow force calculated with $\alpha_D$, $\varepsilon_1$, and $\varepsilon_2 = \text{const}$ (“Calculation A”) and the flow force calculated with $\alpha_D$, $\varepsilon_1$, and $\varepsilon_2 \neq \text{const}$ (“Calculation B”) with the measured one. The constants $\alpha_D$, $\varepsilon_1$, and $\varepsilon_2$ were calculated as the arithmetic averages over the relevant areas of the control surface of all determined values. The flow angles $\varepsilon_1$ and $\varepsilon_2$ were adapted from the CFD simulations and the discharge coefficient $\alpha_D$ from the measurements.

**FIGURE 7.** Spool SE20: Comparison of the flow force calculated with $\alpha_D, \varepsilon_1$ and $\varepsilon_2 = \text{const}$ (“Calculation A”) and the flow force calculated with $\alpha_D, \varepsilon_1$ and $\varepsilon_2 \neq \text{const}$ (“Calculation B”) with the measured one.
Figure 7 confirms that the calculation with variables of $\alpha_D$, $\varepsilon_1$, and $\varepsilon_2$ is more accurate compared to the calculation with constants. Especially beyond a spool stroke of $x_1 = 0.6$ mm, the flow force calculated with variables qualitatively follows the measured flow force. Therefore, the variables of $\alpha_D$, $\varepsilon_1$, and $\varepsilon_2$ were used for the validation of the analytical model.

Using the derived analytical model (eq. 14), the flow forces were calculated for all investigated spool geometries. The calculated flow forces are compared with the measured flow forces for various pressure differentials and are shown in fig. 8 and 9.

![Figure 8](image1.png)  
**FIGURE 8.** Spool SE20 (left); Spool SE13 (right): Comparison of the calculated (C) and measured (M) flow force.

![Figure 9](image2.png)  
**FIGURE 9.** Spool CG (left); Spool BE (right): Comparison of the calculated (C) and measured (M) flow force.

It can be seen in fig. 8 that the analytical model prediction of the flow force is well suited for the square-edged spools well. Furthermore, it is apparent from fig. 8 that the curves of the calculated and the measured flow forces cross each other at a specific spool stroke (between 0.5 and 0.8 mm). This can result from the errors of the CFD simulation or the measurement. Particularly, the flow angles determined from the CFD simulation can cause deviations as shown in tab. 1. Related to the measurements, the calculation of the flow forces was achieved within the relative errors of $-10$ to $+9$ % for the spool SE20, and of $-13$ to $+11$ % for the spool SE13. In analogy to fig. 8, the calculated flow forces are compared with the measured flow forces for the spool BE and CG in fig. 9. It can be seen that the prediction of the flow force is limited for the spool BE. The minimum relative error is $-26$ %, and the maximum relative error is $32$ %. For design purposes, the actuator force is usually multiplied by a factor greater than one. Thus, the deviations of the calculated flow force are still acceptable.

The calculated flow force shows large deviations for the increasing stroke of the spool CG. At smaller spool strokes, the relative error is approximately 18 %. However, the maximum relative error is 170 % at the maximum spool stroke and a pressure differential of 70 bar. There are multiple reasons for that. On the one hand, the determination of the flow angles from CFD simulations is more error-prone for the spool CG compared to other spool geometries, because the fluid is strongly confined by the cylindrical grooves. Consequently, the area-averaging of the flow velocities, respectively the flow angles may show larger deviations compared to the spool geometries without grooves. On the other hand, the spool could rotate slightly during the
measurement, which could impact the magnitude of the measured flow force. Especially at larger spool strokes, the pressure losses and the flow patterns inside the valve could change notably when the spool rotates.

CONCLUSION

Based on the conservation of momentum, an enhanced analytical model for the estimation of the flow forces was derived for the square-edged spool valve. The model was validated with the help of measurements and CFD simulations. It was shown that the analytical model can also be used for other spool geometries with some limitations. Except for the spool CG, the accuracy of the prediction of the flow force is very good for engineering purposes.

Compared to the determination of the flow force from measurements or CFD simulations, the flow force can be quickly estimated by using the analytical model when knowing the values \( \alpha_p, \epsilon_1 \), and \( \epsilon_2 \). Even in case that the absolute values of the flow force should deviate from the exact one, the analytical model can be used to estimate the trends when changing the variables. In this way, the analytical model can be used to carry out a preliminary optimisation of the spool valve.

Nevertheless, the necessity of the estimation of the flow angles and the discharge coefficients still remains a significant limitation of the analytical calculation since there are no generally applicable values. For some spool geometries like the spool BE, it can be assumed that the flow angles approximately equal the bevel angles. In this case, the analytical model can still deliver good results.

In conclusion, the estimation of the flow force with the analytical model represents a good compromise between the accuracy of the estimated flow force and the necessary resources. The accuracy of the analytical estimation could be raised if the flow angles and the discharge coefficients would be estimated accurately.

ACKNOWLEDGMENTS

The IGF research project 18569 N / 1 of the research association Forschungskuratorium Maschinenbau e. V. – FKM, Lyoner Straße 18, 60528 Frankfurt am Main was supported from the budget of the Federal Ministry of Economic Affairs through the AiF within the scope of a program to support industrial community research and development (IGF) based on a decision of the German Bundestag.

REFERENCES

### NOMENCLATURE

<table>
<thead>
<tr>
<th>Designation</th>
<th>Denotation</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a_D$</td>
<td>Discharge coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Jet angle at vena contracta</td>
<td>°</td>
</tr>
<tr>
<td>$\varepsilon_{1/2}$</td>
<td>Inflow/Outflow angle</td>
<td>°</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Fluid density</td>
<td>kg/m$^3$</td>
</tr>
<tr>
<td>$A$</td>
<td>Area</td>
<td>m$^2$</td>
</tr>
<tr>
<td>$A_{1/2}$</td>
<td>Inflow/Outflow area</td>
<td>m$^2$</td>
</tr>
<tr>
<td>$C_1$</td>
<td>Coefficient 1</td>
<td>N</td>
</tr>
<tr>
<td>$C_2$</td>
<td>Coefficient 2</td>
<td>-</td>
</tr>
<tr>
<td>$C_v$</td>
<td>Velocity coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$D$</td>
<td>Spool diameter</td>
<td>m</td>
</tr>
<tr>
<td>$E_R$</td>
<td>Relative error</td>
<td>%</td>
</tr>
<tr>
<td>$F_{fl}$</td>
<td>Steady-state flow force for the inlet throttling</td>
<td>N</td>
</tr>
<tr>
<td>$F_{sys}$</td>
<td>Force acting on a system</td>
<td>N</td>
</tr>
<tr>
<td>$F_{sys,x}$</td>
<td>Axial force acting on a system</td>
<td>N</td>
</tr>
<tr>
<td>$I$</td>
<td>Momentum</td>
<td>kg.m/s</td>
</tr>
<tr>
<td>$L$</td>
<td>Length of the spool chamber</td>
<td>m</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass</td>
<td>kg</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass flow rate</td>
<td>kg/s</td>
</tr>
<tr>
<td>$n$</td>
<td>Normal unit vector</td>
<td>-</td>
</tr>
<tr>
<td>$Q$</td>
<td>Flow rate</td>
<td>m$^3$/s</td>
</tr>
<tr>
<td>$v$</td>
<td>Velocity</td>
<td>m/s</td>
</tr>
<tr>
<td>$v_{1/2}$</td>
<td>Inflow/Outflow velocity</td>
<td>m/s</td>
</tr>
<tr>
<td>$v_{1/2,x}$</td>
<td>Average inflow/outflow radial/axial velocity</td>
<td>m/s</td>
</tr>
<tr>
<td>$w$</td>
<td>Area gradient</td>
<td>l/s</td>
</tr>
<tr>
<td>$x$</td>
<td>Spool stroke</td>
<td>m</td>
</tr>
<tr>
<td>$x_1$</td>
<td>Spool stroke of the investigated valve</td>
<td>m</td>
</tr>
<tr>
<td>$x_2$</td>
<td>Width of the outlet groove of the investigated valve</td>
<td>m</td>
</tr>
<tr>
<td>$\Delta p$</td>
<td>Pressure differential</td>
<td>Pa</td>
</tr>
<tr>
<td>$\Delta r$</td>
<td>The height of the radial clearance</td>
<td>m</td>
</tr>
</tbody>
</table>