TEMPERATURE PREDICTION IN OIL-HYDRAULIC COMPONENTS AND CIRCUIT BY SYSTEM MODELING METHOD COUPLED WITH 3D-CFD SIMULATION

Fumio SHIMIZU*, Kazuhiro TANAKA*, Kohki TOMIOKA**, Hiroshi HIGO*

* Department of Mechanical Information Science and Technology
School 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)

** NITTO DENKO Co., Ltd., Engineering Center,
18 Hirayama Nakahara-cho Toyohashi Aichi, 441-3194, Japan
Department of Fluid Power System, Faculty of Engineering

ABSTRACT

Oil-hydraulic components and circuits become so compact and highly-pressurized that the temperatures occasionally rise so much. A demand to establish a method predicting the temperature rise in working oil and component housings has developed a new technique by coupling the results by three-dimensional computational fluid dynamics analysis with dynamic characteristics simulations by a lumped parameter system. In order to verify the precision of the proposed method, the method was applied to an actual hydraulic system and the predicted results were compared with the experimental measurement results. Both results agree well.

KEY WORDS

Oil-hydraulics, Temperature prediction, Experiments, Bondgraphs, CFD

NOMENCLATURE

- \( A \) [m\(^2\)]: heat dissipation surface area of cylinder
- \( a \) [m]: representative length of inlet port
- \( b \) [m]: length (cylinder end - inlet port center)
- \( c_p \) [J/kgK]: specific heat at constant pressure
- \( D \) [m]: cylinder diameter
- \( h \) [W/m\(^2\)K]: heat transfer coefficient
- \( L_o S \) [m]: characteristic distance of boundary between 3D and 1D flow pattern region
- \( N \) [s]: reciprocating motion cycle of piston
- \( P \) [Pa]: pressure
- \( Q \) [m\(^3\)/s]: volumetric flow rate
- \( q \) [W]: amount of heat inflow
- \( T \) [K]: temperature
- \( t \) [s]: time
- \( U \) [m/s]: velocity vector
- \( V \) [m\(^3\)]: volume of cylinder head chamber
- \( v \) [m/s]: velocity
- \( W \) [W]: power consumed for outside
- \( x \) [m]: axial location of piston head
- \( \delta \) [-]: Kronecker delta
- \( \lambda \) [W/mK]: thermal conductivity
- \( \mu \) [Pas]: viscous coefficient
- \( \nu \) [m\(^2\)/s]: kinematic viscous coefficient
- \( \rho \) [kg/m\(^3\)]: density

Suffix
- \( c \): cylinder
- \( in \): inflow
- \( p \): piston
- \( 1D, 3D \): one-dimensional, three-dimensional
**INTRODUCTION**

An oil-hydraulic system has become smaller and highly pressurized especially in aircrafts, where electric actuators, such as electro-hydrostatic actuators (EHA), are used for moving flight control surfaces. In the industrial field, the use of EHA is an integral part of the more-electric aircraft concept to replace inefficient centralized hydraulic systems with power-on-demand electrical systems [1]-[4]. Removing the centralized hydraulic system will, however, eliminates an effective heat transfer network, thus resulting in an aircraft with less overall heat to reject but with localized “hot spots”. Because of small heat capacity and small surface area for heat dissipation in EHA, the system is raised to high temperature while it works for long time under highly loaded condition. From this viewpoint, it is important to study a practical way to predict precisely system temperature rise.

Some studies have been performed to predict the temperature rise of an oil-hydraulic system considering heat generation and heat transfer in components. [2] [8]. An idea to predict the temperature rise of a hydraulic pipe was proposed by considering heat exchange between the working oil and the pipe-housing [3] [9]. In their studies, lumped parameter thermal models were developed to characterize component heat transfer within their operating environments by coupling with three-dimensional heat conduction analysis. The above idea was developed to predict time-variant temperature distributions in an oil-hydraulic cylinder by comparing with three-dimensional Computational Fluid Dynamics (hereafter, 3D-CFD) analysis [10] [11]. The study showed that in order to predict more precisely the temperature change inside the cylinder while the piston was moving, it was unsuitable to increase cell number merely and the optimal number of cells existed in modeling using lumped parameter modeling systems (hereafter, 1D-Model). It is because there were two characteristic regions, one-dimensional flow pattern region and three-dimensional flow pattern region, inside the cylinder. The latter flow pattern region should be modeled as one lumped cell.

In the present study, the proposed method predicting temperature rise was applied to an actual hydraulic system and was verified by measurement in the experiment. A commercial code as 3D-CFD simulations, ANSYS-CFX, was used to analyze flow patterns as well as temperature distribution time-dependently in a hydraulic pipe and cylinder [12]. Governing equations were the equation of continuity, Navier-Stokes equation [13], and the energy equation. Temperature distributions in housings of the pipe and cylinder could be calculated by use of a heat conduction equation.

The time-dependent 3D-CFD calculations for system dynamics consume so much time that it is not practical despite giving precise results. Instead, it is expected useful to couple 3D-CFD calculations with 1D-Model. This software is used to validate 1D-Model.

**TEST CIRCUIT AND CONDITIONS**

Figure 1 shows a diagram of the test hydraulic circuit, which mainly consists of a pump, pipes, a control valve, a relief valve, a hydraulic cylinder, and a tank. The control valve is a closed center proportional solenoid valve. The experimental measurements of temperature are conducted using this equipment. The measurement location, such as A-port, B-port and the tank, is manifested in a symbol as TA, TB and TT, respectively.

![Figure 1 Test equipment of oil-hydraulic circuit](image)

**DYNAMIC CHARACTERISTICS SIMULATION**

Dynamic characteristics of the system are simulated by the Bondgraphs method based on a lumped parameter system [5-7]. The system model of Bondgraphs of an oil-hydraulic system is generally represented in Fig. 2 [6]. The R elements from R1- to R8-element mean energy losses and heat generation. R1, R2, R3, and R4 indicate resistance effect of a control valve. R5, R6, and R8 element indicate the pressure loss in the pipe from the control valve to the piston head chamber, from the valve to the piston rod chamber, and from the pump to the control valve, respectively. R7 element indicates the frictional force between the piston and the cylinder. The characteristic equations of main elements are shown on the right of the figure for reference.

**1D-MODEL SIMULATION METHOD**

1D-Model of Heat Flow for oil-hydraulic system

A schematic diagram for the 1D-Model for temperature analysis is shown in Fig. 3, where an arrow indicates...
heat flow direction. This figure is so similar to the system Bondgraphs shown in Fig. 2. The elements overlapping between Fig. 2 and Fig. 3 mean the same elements.

**Temperature rise in valve and tank**
Because a hydraulic valve generates high pressure loss among components in a hydraulic circuit, it is important to predict heat generation. When working oil flows through the throat part between valve body and housing, high pressure loss causes heat generation there. Heat balance in a valve is expressed in Eq. (1), where heat conduction term can be neglected because the working oil velocity is fast in the valve. This equation covers all types of valve [11].

\[
\frac{dT_i}{dt} = \frac{1}{c_p \rho V_i} \left[ \Delta p Q + c_p \rho Q(T_{in} - T_i) + h A_h(T_{wall} - T_i) \right] \tag{1}
\]

In the tank, the volume of working oil changes time-dependently due to the sum of inflow from T-port of a spool valve and outflow to a hydraulic pump.

Equation (2) consists of terms for heat flux flowing out and heat release by an oil cooler. In this equation, the heat release from the tank wall and the oil cooler is determined by the experimental measurement.

\[
\frac{dT_i}{dt} = \frac{1}{c_p \rho V_i} \left[ \left( c_p \rho Q_{in} T_{in} - c_p \rho Q_i T_i + h A_h(T_{wall} - T_i) \right) \right]
+ W_{\text{cooler}} - c_p \rho V_i \frac{dV_i}{dt} \tag{2}
\]

**Heat generation from working oil in a pipe**
The diagram of heat balance in a pipe is shown in Fig. 4, where suffix \( i \) is the cell index. Temperature of each pipe cell is simulated by heat balance among the heat generation in the cell, the heat transfer from the pipe wall to the internal oil and heat transportation between the neighboring cells of the pipe. Assuming that working oil is incompressible and does not work to outside of \( i \) th cell, the following Eq. (3) and (4) are derived [9]. Suffix \( n \) indicates computing time iteration. In this case, the inertial effect of oil column is ignored.

\[
\frac{dT_i}{dt} = \frac{1}{c_p \rho V_i} \left[ (P_{i+1} - P_i) Q^* + c_p \rho Q_i (T_{i+1} - T_i) + \dot{q}_i^* \right] \tag{3}
\]

\[
\dot{q}_i^* = h A_h(T_{wall} - T_i^*) + \lambda A_T \frac{T_{T+1} - 2T_i + T_{i-1}}{2l} \tag{4}
\]
Interface of heat transfer between pipe inside and housing

Here it is proposed how to couple the temperature prediction 1D-Model for the working oil with 3D-CFD for heat conduction in a pipe housing. The temperature data are exchanged between the 1D-Model and 3D-CFD at an interface such as the pipe inner wall surface. This technique is illustrated in Fig. 5 [9], in which a room partitioned by solid lines at the bottom side corresponds to i th cell in the 1D-Model of the pipe inside. Top side indicates the pipe housing and is divided into many grids for the 3D-CFD calculation. Generally the number of grids is much more than that in the 1D-Model. In exchanging data on the interface, the mean value of temperature distributed along the whole length of i th cell in the 1D-Model, , is given to the 1D-Model as an additional input temperature into the inside. Next, the temperature is newly calculated in i th cell through Eq. (4). From the new temperature, heat flux is calculated and given to 3D-CFD program as a boundary condition. Because time constant is quite different between the 1D-Model and 3D-CFD simulations, however, the data-exchange timing will be studied in the following section. By repeating this calculation cycle, system dynamic characteristics inside the pipe as well as temperature distributions in the pipe housing are calculated time-dependently.

The calculated results on time-dependent temperature change based on 1D-Model were compared with calculated results using 3D-CFD. Both results agree well, especially as the number of cells increases. This means that the Bondgraphs method is valid in prediction of heat generation and heat transfer. On coupling 1D-Model with 3D-CFD, CFD simulation can be performed once a 10 iterations of BG simulation to ensure precise coupling.

Temperature rise prediction based on heat balance in a cylinder

Equation (5) is derived in the cylinder chamber [11].

\[
\frac{dT_t}{dt} = \frac{1}{c_p \rho V_t} \left( c_p \rho Q_i + h A_s (T_{wall} - T_t) - c_p \rho Q_o \frac{dV}{dt} \right)
\]  

(5)

In the above equation, the first term of right hand side indicates the inflow or outflow amount of heat with piston motion, the second the amount of heat transfer between the cylinder chamber and housing, and the third the effect of cylinder volume change. The temperature change in the cylinder chamber can be calculated by solving Eq. (5).

Effect of cell number in 1D-Model of cylinder [11]

Figure 6 shows a 3D-CFD result of streamlines through the inlet port in the cylinder head chamber when the piston is moving with the maximum speed at the neutral position, \( x=0 \text{[mm]} \), from the maximum displacement, \( x=-100 \text{[mm]} \). Three arrows are described at intervals of \( 100 \text{[mm]} \) from the cylinder head end wall. 3D vortex flow pattern and parallel flow pattern are observed near the inlet port and near the piston, respectively. The figure shows that a transition region between 3D and 1D flow pattern can be seen near \( x=200 \text{[mm]} \).

![Figure 6 Streamlines at cylinder chamber](image)

It is important to separate the flow field into two parts, such as 3D vortex flow pattern region and 1D parallel flow pattern region, in making 1D-Model of the cylinder with high precision. 3D vortex flow pattern region should be modeled as one cell and 1D parallel flow pattern region can be divided into some cells.

How to determine LoS

The location of separating the flow patterns (hereafter, LoS) is a key parameter in making 1D-Model of the cylinder. LoS was determined by dimensional analyses considering directional dependency against any cylinder with a different size. As a result, the non-dimensional length of LoS is expressed by \( x/a \) is a function of Reynolds number expressed by \( \rho D v_m/\mu \) with representative velocity \( v_m \), the product of Reynolds number with representative velocity \( v_p \) and \( D/a \), and the non-dimensional length \( h/a \). After determining unknown parameter values so that LoS is the same as the result of 3D-CFD, the following characteristic equations are
derived crossing the dividing line of $\rho D v in/\mu = 2.3 \times 10^4$.

LoS \([x/a]\):

(i) $\rho D v in/\mu \geq 2.3 \times 10^4$

\[
x/a = 0.015 \left( \frac{D}{a} \frac{\rho D v in}{\mu} \right)^{-0.69} \left( \frac{\rho D v in}{\mu} \right)^{0.96} \left( \frac{b}{a} \right)^{0.098}
\]  \( (6) \)

(ii) $\rho D v in/\mu < 2.3 \times 10^4$

\[
x/a = 1.44 \left( \frac{D}{a} \frac{\rho D v in}{\mu} \right)^{-0.014} \left( \frac{\rho D v in}{\mu} \right)^{0.19} \left( \frac{b}{a} \right)^{0.27}
\]  \( (7) \)

Figure 7 shows the comparison of LoS obtained between the results through Eq. (6) & (7) and the 3D-CFD results. The abscissa and ordinate indicate Reynolds number with representative velocity $v_{in}$ and the non-dimensional length of LoS \([x/a]\), respectively. The maximum difference is 14% and the both results agree well. It is verified that the suitable LoS can be calculated through Eq. (6) & (7).

\[ hL = \frac{Nu}{PrRe^{6.21}} \]  \( (8) \)

\[ \sqrt{\frac{2}{c_f}} = 2.50 \ln(Re c_f) + 2.40 \]  \( (9) \)

$\Pi(In) = 12.5 Pr^{2/3} + 2.12 \ln Pr - 9.24$ : $Pr \geq 0.7$  \( (10) \)

On the other hand, the heat transfer coefficient in the 1D-flow pattern region is obtained by the following equation as laminar heat transfer.

\[ Nu = 1.62 \left( \frac{Re_d Pr^{\frac{a}{x}}}{\lambda} \right)^{\frac{1}{7}}, \quad Nu = \frac{hL}{\lambda} \]  \( (11) \)

Table 1 and Fig. 9 show the values of the heat transfer coefficients around the cylinder and their definitions, respectively. The values in Table 1 are 1/3 of the results calculated in Eq. (8) and (11). Table 11 shows the analysis conditions and values of the properties. Kinematic viscosity of the working oil is defined as a function of temperature. Heat exchange between the cylinder inside and the housing is performed similarly to the case of the pipe and pipe housing, which is introduced in Fig. 5.

**Heat Transfer Coefficient for Temperature Analysis**

Figure 8 shows the numerical grids for 3D-CFD calculation on temperature distributions of the cylinder housing. The number of numerical grid is 14,000. Heat transfer coefficient in the cylinder head and rod chamber should be given previously before calculating heat exchange between the 1D-Model and 3D-CFD analysis. To determine the heat transfer coefficients is especially important in the interfaces between 1D- and 3D-thermal analysis. It is necessary to consider separately between 3D- and 1D-flow pattern region, as discussed previously, because the flow can be considered turbulent in the 3D-flow pattern region and a formula of turbulent heat transfer is applicable there. On the other hand, the flow can be considered laminar in the 1D-flow pattern region and a formula of laminar heat transfer is applicable there.

However, the inlet flow velocity as well as the flow condition varies temporally as the piston moves sinusoidally in this experiment. So the heat transfer coefficient can also be considered to vary temporally. In the present study in order to simplify the model in the 3D-flow pattern region, 1/3 of the heat transfer coefficient in case of the maximum piston velocity is given as time-averaged heat transfer coefficient. This value, 1/3, is obtained previously as the best one in comparison between the experiment and the 3D-CFD simulation. In this regard, the heat transfer coefficient is calculated by the formula of flat-plate turbulent heat transfer [14], shown in the following equation.

\[ Nu = \frac{1}{2} \frac{PrRe}{0.85 + \Pi(Pr) \sqrt{\frac{c_f}{2}}} \]  \( (8) \)

\[ \sqrt{\frac{2}{c_f}} = 2.50 \ln(Re c_f) + 2.40 \]  \( (9) \)

$\Pi(Pr) = 12.5 Pr^{2/3} + 2.12 \ln Pr - 9.24$ : $Pr \geq 0.7$  \( (10) \)
Table 1 Heat transfer coefficient at a LoS

<table>
<thead>
<tr>
<th>LoS [mm]</th>
<th>( h_{3D} ) [W/m(^2)K]</th>
<th>( h_{1D} ) [W/m(^2)K]</th>
<th>( h_r ) [W/m(^2)K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>94</td>
<td>118</td>
<td>75.4</td>
<td>76</td>
</tr>
</tbody>
</table>

3D flow region (\( h_{3D} \))

1D flow region (\( h_{1D} \))

Table 2 Experimental conditions

<table>
<thead>
<tr>
<th>Motion of Piston</th>
<th>Frequency, 0.1, 0.2, 1.0 Hz</th>
<th>Amplitude, 50 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Supply Pressure</td>
<td>10 MPa</td>
<td></td>
</tr>
<tr>
<td>Experimental Time</td>
<td>1200 s</td>
<td></td>
</tr>
<tr>
<td>Working Fluid</td>
<td>ISO-VG32</td>
<td></td>
</tr>
<tr>
<td>Ambient Temperature</td>
<td>23 deg.</td>
<td></td>
</tr>
</tbody>
</table>

RESULTS AND DISCUSSIONS

Figure 10 shows temperature measurement locations on the surface of the experimental test cylinder which is a widely-used type. This measurement was conducted under unloaded condition. The measurement was performed at three locations on the cylinder outer surface, which are shown as the number from Point 1 to 3 in Fig. 10. The test conditions are shown in Table 2. In the test, the temperature changes are measured when the piston is moving with constant oscillation amplitude and frequency. In this experiment, heat transfer coefficient was given as 10[W/m\(^2\)K] from each component to the air and as 170[W/m\(^2\)K] in the cooler, which was calculated reversely from the experimental condition.
Temperature change at tank

Figure 11 shows the comparison on the temperature of working oil in the tank in case that the frequency of piston motion is 0.2[Hz] and the amplitude is 50[mm]. At t=1200[sec], the tank temperature increases up to T=45[deg], which is lower than the temperature at A-port, T=50[deg], shown in Fig. 12 (a). This is because the tank volume is much larger than that of the other component. The calculated results agree well with the experimental results and it means that it becomes possible to predict precisely the tank temperature when the heat transfer coefficient from the tank can be calculated in the experiment.

Temperature change at A/B port

Figure 12 (a) shows the results on the time-dependent temperature change of working oil at A- and B-port in case of the piston motion with the piston frequency, 0.2[Hz], and the amplitude, 50[mm]. In the figure, “TA” and “TB” indicate the temperature at A-port and B-port, respectively. In the both results, the temperature at A-port is higher than that at B-port. This is because A- and B-port are connected to the cylinder head and rod chamber, respectively, and the cross-section of the head chamber is larger than that of the rod chamber. When the piston moves from the head to rod side, the flow rate into the head chamber becomes more than that out of the rod chamber and the pressure loss became larger at A-port. The calculated results agree well with the experimental results and the difference between both results is less than 1[deg]. This fact shows that the temperature at A- and B-port behind the control valve is well predicted.

The good agreement between the both results is obtained in other cases. Figure 12 (b) shows the results at A- and B-port in the second case of the piston motion with 1[Hz] frequency and 10[mm] amplitude. The both results have the same tendency. Figure 12 (c) shows the results on the time-dependent temperature change of working oil at A- and B-port in the last case of the piston motion with the frequency, 0.1[Hz], and the amplitude, 150[mm]. The both results have the same tendency.

Temperature change in piston head chamber

Figure 13 (a) shows the calculated and measured results on the temperature change at Point 1, 2 and 3 which are shown in Fig. 10. The temperatures increase with time at all locations because the temperature rise calculated according to 1D-Model is exchanged to the housing calculated through 3D-CFD. The temperature at Point 2 is the highest among the three measurement locations. This is because Point 2 is located just in the inlet port, where the working oil with high temperature is supplied there continuously. The temperature at Point 1 is lower than that at Point 2, because the housing wall thickness is different between Point 1 and 2, as show in in Fig. 10. The wall thickness is 25[mm] thick at Point 1 and 10[mm] thick at Point 2. In any case, the calculated

![Figure 13](image)

(a) Temp. at piston-head side (0.2[Hz], 50[mm])

(b) Temp. at piston-head side (1[Hz], 10[mm])

(c) Temp. at piston-head side (0.1[Hz], 150[mm])

Figure 13 Temperature change at piston-head side

Table 3 Calculation time

<table>
<thead>
<tr>
<th>Exp. Time</th>
<th>Calculation Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>1200 sec.</td>
<td>ANSYS-CFX (Estimation)</td>
</tr>
<tr>
<td></td>
<td>Proposed method</td>
</tr>
<tr>
<td></td>
<td>PCI(Power of CPU: Pentium Xeon 3.6GHz)</td>
</tr>
</tbody>
</table>
results agree well with the experimental results and the difference between both results is less than 1[deg]. Figure 13 (b) shows the results at Point 1, 2 and 3 in the second case of the piston motion with the frequency, 1[Hz], and the amplitude, 10[mm]. The calculated results agree well with the experimental results. Figure 13 (c) shows the results at Point 1, 2 and 3 in the last case of the piston motion with the frequency, 0.1[Hz], and the amplitude, 150[mm]. Also in this case, the calculated results agree well with the experimental results.

Calculation time
Table 3 shows the comparison on the time necessary for calculating temperature rise of the test hydraulic circuit between through the proposed method and through a commercial CFD code. To calculate the dynamics and temperature changes in the circuit for 1200[sec], the proposed method takes 92[min] and the CFD code needs 264 days, which is estimated from the fact that it needs $1.9 \times 10^8$[sec] to calculate temperature rise for 20[sec] in the actual time. The result shows that the proposed method gives very good performance for prediction of temperature rise.

CONCLUSION
In order to establish a method predicting temperature rise in the working oil and component housings in an oil-hydraulic circuit, a new idea of coupling 3D-CFD results with 1D-Model calculation is proposed in this study. The 1D-Model of the system is based on a lumped parameter system by the Bondgraphs. The proposed method is applied to an actual system and good prediction of the temperature rise is obtained. The proposed method can predict the temperature rise in an oil-hydraulic circuit precisely and smoothly.

REFERENCES