Research on the thermal flow characteristics of viscosity oil in hydrodynamic torque converter

Special Collection: Recent Advances in Fluid Dynamics and Its Applications

Jiahua Zhang (张嘉华); Qingdong Yan (闫清东); Boo Cheong Khoo; Cheng Liu (刘城); Zhifang Ke (柯志芳); Wei Wei (魏巍)

Physics of Fluids 36, 053112 (2024)
https://doi.org/10.1063/5.0211749
Research on the thermal flow characteristics of viscosity oil in hydrodynamic torque converter

Cite as: Phys. Fluids 36, 053112 (2024); doi: 10.1063/5.0211749
Submitted: 1 April 2024 · Accepted: 9 May 2024 ·
Published Online: 23 May 2024

Jiahua Zhang, Qingdong Yan, Boo Cheong Khoo, Cheng Liu, and Wei Wei

AFFILIATIONS
1 School of Mechanical Engineering, Beijing Institute of Technology, Beijing 100081, China
2 Department of Mechanical Engineering, National University of Singapore, Singapore 117575
3 Yangtze Delta Reg Acad, Beijing Institute of Technology, Jiaxing 314000, China
4 Advanced Technology Research Institute, Beijing Institute of Technology, Jinan 250307, China

Note: This paper is part of the Special Topic, Recent Advances in Fluid Dynamics and Its Applications.

Author to whom correspondence should be addressed: liucheng@bit.edu.cn

ABSTRACT

The increase in power density of hydrodynamic torque converters (HTCs) leads to a sharp rise in temperature within flow channels, affecting the reliability. In order to accurately predict the thermal effect and temperature distribution characteristics of the HTC internal viscosity oil, a multi-physics computational fluid dynamics (CFD) model is proposed. A specialized test bench was established, and the macro and internal flow temperature data were obtained. HTCs with different working conditions and wheel sets were studied. The results indicate that CFD model considering energy equation can accurately predict the overall hydrodynamic performance and the flow field temperature characteristics under different rotating conditions. The prediction error of the overall temperature rise is within 4.92%, and the flow field temperature prediction error of the stator is under 14.3%. The hydraulic characteristics is improved by 6.02%. The analysis of internal flow and energy exchange characteristics indicates the thermal effects and temperature distribution mechanisms caused by energy loss in the flow field within the HTC. The study provides an effective computational model for the prediction and control of the heat generation of the HTC and enhances the depth of research on the flow mechanism of inhomogeneous flow fields caused by thermal effects.

Published under an exclusive license by AIP Publishing. https://doi.org/10.1063/5.0211749

I. INTRODUCTION

The hydrodynamic torque converter (HTC) is extensively employed in vehicle transmissions due to its superior efficacy in power transfer. This sophisticated hydrodynamic transmission apparatus comprises a pump, turbine, and stator, each equipped with intricately twisted blades, as depicted in Fig. 1. The mechanism of power transfer within the HTC involves the movement of working oil between impellers, highlighting the pivotal role of the oil’s properties on the converter’s overall performance. The thermal efficiency of the HTC is significantly influenced by the characteristics of the working oil, with a majority of energy losses manifesting as alterations in the oil’s temperature. Elevated temperatures of the working oil can precipitate a decline in its viscosity, thereby impairing the transmission and lubrication capabilities of the system. Concurrently, heightened temperatures may accelerate the aging process of seals, increasing the propensity for leaks and adversely affecting the functionality of the system’s hydraulic components.

Recent advancements in the study of hydrodynamic machines have been marked by a notable increase in both numerical and experimental investigations. Liu et al. embarked on an exploration of the impact that various blade shape parameters have on performance, uncovering the fluid dynamics underlying the effects of oil charge conditions on cavitation behavior within HTCs, thereby offering valuable insights for enhancing performance. Guo et al. innovated by constructing a full passage geometry alongside a computational fluid dynamics (CFD) model, facilitating the analysis of cavitation flow behaviors under varying temperatures and pump rotational speeds within the HTC. Further contributing to the field, Liu et al. utilized four distinct scale-resolving simulation (SRS) approaches to simulate multiple hydraulic components across hydraulic couplings, retarders, and torque converters. Their research delved into the significant advantages of employing large eddy simulations for predicting flow fields. In an effort to mitigate stator cavitation, Ran et al. introduced an innovative method involving the slotting of one side of the stator...
suction side, effectively suppressing cavitation within the HTC. Li et al.\textsuperscript{9} applied the multi-scale resolution model to undertake a more comprehensive numerical analysis of the flow field within a retarder, taking into account the cavitating flow. Finally, Cui et al.\textsuperscript{10} conducted an in-depth study on the cavitation circulation mechanism of mining hydraulic couplers, through which they analyzed cavitating flows, contributing to a deeper understanding of these complex phenomena.

Contemporary research on the thermal dynamics within turbomachinery primarily focuses on aerospace turbines, examining aspects such as the thermal effects of impeller airflow\textsuperscript{11} and the behavior of various seals\textsuperscript{2,13} under thermal stress. Investigations also extend to the thermal-induced power losses in the mechanical frameworks of rotating machinery.\textsuperscript{14} CFD technology has also been widely applied in related flow analysis.\textsuperscript{15,16} However, studies on HTCs are scant due to their complex flow field structures and operational conditions. Yang et al.\textsuperscript{17} conducted a comparative analysis on the temperature distribution within HTCs using different turbulence models, but their approach, limited to instantaneous temperature predictions, lacked experimental validation for a comprehensive understanding of thermal effects. Robinette and Blogh\textsuperscript{18} devised a transient model to estimate the temperature of the fluid within automotive HTCs, relying on an energy balance and convective heat transfer correlations. Despite its utility, this model did not delve into the intricate mechanisms of thermal effects within the flow field.

Quantifying multiple physical parameters in the flow fields of rotating machinery is complex, primarily due to the intrusive nature of traditional probe-based measurement techniques.\textsuperscript{19} In the context of turbomachinery, these methods can significantly disturb the flow field, affecting the accuracy of pressure, temperature, and velocity readings. However, the adoption of wall-mounted miniature transducers presents a less intrusive alternative,\textsuperscript{20} potentially minimizing the impact on the flow field and enhancing the precision of test outcomes.

The complexity of multiphysics computational models, particularly for turbomachinery with intricate blade geometries, poses significant challenges in achieving convergence. This complexity necessitates a high-quality mesh. Research by Ali and Baccar\textsuperscript{21} involved a detailed three-dimensional analysis using CFD to explore the dynamics of Bingham fluids. Banas and Badur\textsuperscript{22} conducted a thermal-fluid structure interaction (FSI) analysis on turbine guide vanes, examining the effects of turbulence models on heat transfer and stress distribution. Jogee and Anupindi\textsuperscript{23} work with large-eddy simulation (LES) provided insights into the heating mechanisms of fluids through pinfin arrays. Liu et al.\textsuperscript{24} introduced a virtual wall thickness method to enhance the simulation of temperature fields in turbine blades with thermal barrier coatings, improving both the modeling process and efficiency.

To deepen the understanding of the thermal effects of viscosity oil in HTC and achieve accurate predictions of the thermal flow characteristics, this study employed CFD simulation techniques. It takes into account the oil viscosity and density changes of the inhomogeneous flow field and thermal behavior during operation. A full 3D non-enclosed internal flow domain model was established, discretized using high-quality hexahedral meshes. Variable viscosity fluid thermal effect analysis equation were developed, with a focus on eddy viscosity through partial differential equations.\textsuperscript{25} In addition, a macroscopic thermal effect and microscopic flow field test system for HTC were constructed, and the thermal flow characteristics were experimentally analyzed. This quantitatively verified the accuracy of the HTC thermal effect model. This method helps in predicting the thermal effects and internal flow mechanisms of HTC, laying a theoretical foundation for performance optimization, proper oil charging conditions, and structural design assessment.

II. COMPUTATIONAL MODEL

A. Turbulence and thermal model

The shear stress transport (SST) $k-\omega$ model is instrumental in turbulence modeling, particularly for predicting the occurrence of flow separation under different inflow conditions.\textsuperscript{26,27} This model integrates the transport mechanism of turbulent shear stress, enhancing the accuracy of flow separation predictions. It employs specific equations for turbulence kinetic energy ($k$) and the rate of dissipation ($\omega$), which are central to its analytical framework and are given as follows:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i} \right] + P_k - \beta \rho \omega \omega - \beta_r \rho \omega^2$$

$$+ 2(1 - F_1) \rho \sigma_s \omega^2 \frac{\partial k}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right]$$

$$+ \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] - \beta_r \rho \omega^2$$

$$+ 2(1 - F_1) \rho \sigma_s \omega^2 \frac{\partial k}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] + \frac{\partial k}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right]$$

$$+ \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] - \beta_r \rho \omega^2.$$  \hfill (1)

$$+ 2(1 - F_1) \rho \sigma_s \omega^2 \frac{\partial k}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] + \frac{\partial k}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right]$$

$$+ \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] - \beta_r \rho \omega^2.$$  \hfill (2)

Here, $F_1$ is defined as the following blending function:

```plaintext
F_1 = \frac{1}{C_20/C_21} \left( 1 + \frac{\partial \omega}{\partial x_i} \right) \left( 1 + \frac{\partial \omega}{\partial x_i} \right) \left( 1 + \frac{\partial \omega}{\partial x_i} \right)
```
\[
\begin{align*}
F_1 &= \tan \left\{ \min \left[ \max \left( \frac{\sqrt{k}}{\beta' \omega y' \omega} , 500 \right) \frac{4 \rho \sigma_{\omega \omega}}{CD_{\text{vis}}} \right]^4 \right\} \\
CD_{\text{vis}} &= \max \left( 2 \rho \sigma_{\omega \omega} \frac{1}{\alpha \omega} \frac{\partial k}{\partial x_1} \frac{\partial \omega}{\partial x_1} \right) 10^{-10}.
\end{align*}
\]

The \(k-\varepsilon\) model is employed in the boundary layer away from the wall. When \(F_1 = 0\), the \(k-\omega\) model is applied in the inner layer. The turbulent viscosity \(\mu_t\) is defined by

\[
\mu_t = \frac{a_1 \kappa}{\max(a_1 \alpha, SP_2)} ,
\]

where the strain rate invariant measure is \(S\). Then, \(F_2\) is defined as the second blending function and is given as follows:

\[
F_2 = \tanh \left[ \max \left( 2 \frac{\sqrt{k}}{\beta' \omega y' \omega} \right)^2 \right] .
\]

In the stagnation zone, the SST \(k-\omega\) turbulence model defines a production limiter to prevent turbulence generation, which is defined as

\[
P_h = \mu_t \frac{\partial u_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) .
\]

In the complex flow field of HTC, the transformation of fluid kinetic energy into internal energy, influenced by wall kinetic forces and turbulence, significantly contributes to temperature increases. This conversion process is encapsulated in the energy equation, which quantitatively describes the relationship between the fluid’s kinetic energy and its resultant thermal effects within the system. The energy equation is given as follows:

\[
\frac{\partial (\rho E)}{\partial t} + \frac{\partial}{\partial x_j} [\rho u_i (\rho E + p)] = \frac{\partial}{\partial x_j} \left( \lambda \frac{\partial T}{\partial x_j} - \sum_j h_{ij} \tau_{ij} \right) + S_h .
\]

Here, \(T\) is the temperature, \(u_i\) is the velocity component, \(t\) is the time, \(\mu\) is the dynamic viscosity, \(E\) is the unit mass energy, \(\lambda\) and \(h_{ij}\) are the thermal conductivity and enthalpy component, and \(\tau_{ij}\) and \(S_h\) are the viscous stress tensor and viscous dissipation term.

Due to the significant temperature influence on the properties of oil, changes in temperature will greatly affect the operating performance of equipment that uses oil as the working medium,\(^{30,31}\) including HTC that use 15W40 viscosity lubricating oil. Temperature variations in the flow significantly alter the fluid’s viscosity, leading to inhomogeneous viscosity and density profiles.\(^{2,33}\) This change in viscosity, in turn, influences the kinetic energy within the flow, establishing a reciprocal relationship between viscous forces and the flow’s kinetic energy. This two-way coupling underscores the complex interplay between thermal effects and fluid dynamics in systems like HTCs. According to the test results of viscosity oil at different temperatures, the relationship between dynamic viscosity \(\mu(T)\), oil density \(\rho(T)\), and temperature \(T\) is shown in the following equations and Fig. 2:

\[
\mu(T) = 116.421 \times T^{-1.95} ,
\]

\[
\rho(T) = 888.65 - 0.64 \times T .
\]
determined to be 1.68%, while the deviation in the temperature rise ($\Delta T$) was observed to be 1.13%, both satisfying the stipulated criterion for mesh independence.

C. Simulation settings

In a complex flow regime, achieving convergence of thermal effects presents a notable challenge, compounded by the gradual increase in outlet temperature within high-viscosity flow fields. Consequently, this study adopts a three-stage approach to the numerical calculation process. Initially, leveraging upwind numerical scheme, the study conducts an initial computation of the HTC isothermal flow field, aiming to delineate the circulation flow pattern. Subsequently, building upon the outcomes of the preliminary calculation with lower precision, a high-resolution advection scheme is employed, and a smaller time step size ($1 \times 10^{-4}$ s) is applied to achieve better convergence. This scheme facilitates a detailed analysis of isothermal flow turbulent structure, employing the SST model to ensure the attainment of precise turbulent analysis results. Simultaneously, mindful of the imperative for stability and computational efficiency in thermal flow calculations, the study incorporates the frozen rotor boundary to facilitate data exchange between different domains. Finally, building upon the refined isothermal flow field results, the study implements a total energy model to establish the steady-state thermal effect within the inner flow field. The setting details are shown in Table I. Considering that the hydraulic HTC is installed in a closed enclosure and the testing duration is relatively short, adiabatic thermodynamic boundary conditions are adopted.

III. HYDRODYNAMIC TORQUE CONVERTER TEST RIG

A. Macro performance test of HTC

The test rig is shown in Fig. 6 encompasses an oil compensation system, which includes components such as a heat exchanger and pump station, the primary test apparatus comprising a drive motor, transducers, and a dynamometer, and the control and data acquisition framework based in signal acquisition systems. The operational parameters of the HTC are modulated through the synergistic function of the drive motor, with a rated power of 250 kW, and the dynamometer, with a rated power of 315 kW. To accurately gauge the requisite torque across varying speed ratios, torque and speed transducers are employed (7). The input rotating velocity is maintained constant, whereas the output rotating speed is incrementally elevated to a certain value. The state of oil situation is continuously monitored during the testing phase by analyzing the inlet oil’s temperature and pressure,
utilizing transducers labeled as 4, with the inlet oil temperature being regulated at 70 °C by managing the flow of oil filling and the operation of the heat exchanger. Furthermore, the temperature of the exiting oil is kept below 120 °C under the monitoring of outlet pressure and temperature sensors. Parameter recording is initiated simultaneously once the system reaches a steady state, which is ensured after a duration of 5 s. The precision of the measuring instruments employed in the experimental framework is calibrated to align with the spectrum of the measurement parameters required, detailed in Table II. The diameter of HTC is 315 mm, and the configuration of the base impeller and blade structures (PIT1S1) is shown as Figs. 7 and 8.

As shown in Fig. 9(a), temperature transducers are installed at the inlet and outlet pipe joints of the HTC housing to detect the temperature of the circulating oil, ensuring that the oil does not degrade during operation while also testing the temperature rise characteristics. By controlling the liquid filling flow and radiator power, the inlet oil temperature is maintained at 70 °C. Taking the stall condition with the maximum temperature rise ($SR = 0$) as an example, the collection

<table>
<thead>
<tr>
<th>Analysis step</th>
<th>I. No thermal</th>
<th>II. No thermal</th>
<th>III. Thermal</th>
</tr>
</thead>
<tbody>
<tr>
<td>Advection scheme</td>
<td>Upwind</td>
<td>High resolution</td>
<td>High resolution</td>
</tr>
<tr>
<td>Interface model</td>
<td>Frozen rotor</td>
<td>Frozen rotor</td>
<td>Frozen rotor</td>
</tr>
<tr>
<td>Thermal model</td>
<td>None</td>
<td>None</td>
<td>Total energy</td>
</tr>
<tr>
<td>Time step size</td>
<td>$1 \times 10^{-3}$ s</td>
<td>$1 \times 10^{-4}$ s</td>
<td>300</td>
</tr>
<tr>
<td>Step number</td>
<td>300</td>
<td>RMS $1 \times 10^{-5}$</td>
<td>RMS $1 \times 10^{-5}$</td>
</tr>
<tr>
<td>Convergence target</td>
<td>RMS $1 \times 10^{-4}$</td>
<td>RMS $1 \times 10^{-5}$</td>
<td>RMS $1 \times 10^{-5}$</td>
</tr>
<tr>
<td>Pump status</td>
<td>$NP = 1800$ rpm</td>
<td>$NP = 0$–1620 rpm</td>
<td>$NP = 0$ rpm</td>
</tr>
<tr>
<td>Turbine status</td>
<td>$NT = 0$ rpm</td>
<td>$NT = 0$ rpm</td>
<td>$NT = 0$ rpm</td>
</tr>
<tr>
<td>Stator status</td>
<td>$NS = 0$ rpm</td>
<td>$NS = 0$ rpm</td>
<td>$NS = 0$ rpm</td>
</tr>
<tr>
<td>Inlet details</td>
<td>$P_{inlet} = 0.9$ MPa</td>
<td>$P_{inlet} = 0.9$ MPa</td>
<td>$P_{inlet} = 0.4$ MPa</td>
</tr>
<tr>
<td>Outlet details</td>
<td>$P_{outlet} = 0.4$ MPa</td>
<td>$P_{outlet} = 0.4$ MPa</td>
<td>$P_{outlet} = 0.4$ MPa</td>
</tr>
<tr>
<td>Boundary details</td>
<td>No-slip and smooth wall</td>
<td>No-slip and smooth wall</td>
<td>Adiabatic</td>
</tr>
<tr>
<td>Thermodynamic boundary conditions</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

FIG. 6. Rig test for the HTC.
frequency of the inlet and outlet temperature sensors is set to 50 Hz. Temperature collection starts when the pump wheel speed reaches 1800 ± 3 r/min, with a collection duration of 5 s, as shown in Fig. 9(b). It can be seen that the inlet and outlet temperatures enter a stable state 2 s after the start of collection, with the inlet temperature fluctuating less than 0.8°C, and the outlet temperature fluctuating less than 0.7°C. The study selected the temperature data from the third to the fifth second of the collection results for time-averaging processing, and the results serve as the inlet and outlet temperatures of the HTC.

B. Multi-physics field test system of HTC

The multi-physical field test system for circulating oil relies on the HTC hydraulic performance test rig (Fig. 6). Ultra-miniature dynamic pressure sensors and their wires are installed in the grooves on the blade surfaces and fixed with metal sheets to achieve stable installation under high-speed and high-temperature oil flow conditions. The pressure sensors used have a measurement accuracy higher than ±0.1%. The micro temperature transducers are mounted on the shroud of the stator near the blade leading and tail, respectively. The measurement accuracy of the sensor is ±0.5%. The transducers installation locations are shown in Fig. 7.

IV. RESULTS AND ANALYSIS

A. The overall thermal effect of the flow field

According to the quantitative analysis, the temperature rise results between the inlet and outlet of the flow field (Fig. 10). The inlet temperature is kept stable at 70 ± 1°C. Both the numerical model and test data indicate that the maximum temperature rise occurs at the stall condition \((SR = 0)\), meaning the largest temperature difference between the inlet and outlet is 45.13°C. As the \(SR\) increases, there is a corresponding decrement in the circulation velocity of the oil within the system, the outlet oil temperature also decreases, corresponding to a decline in temperature rise. Although the heat exchange between the ambient air and HTC is somewhat suppressed by the casing, there is still some energy exchange with the external environment under high temperatures, leading to a relatively larger error in CFD results under high-temperature rise conditions at low \(SR\). The maximum relative error is 4.92% \((SR = 0)\). When the \(SR\) exceeds 0.6, the temperature rise observed experimentally surpasses that predicted by calculations. This discrepancy arises due to a lesser temperature differential between the internal flow field of the HTC and its surrounding environment, compounded by the heat generated within the internal bearings and losses from disk friction. Therefore, the multiphysics numerical model can accurately predict the working temperature rise characteristics of the fluid in the HTC with high precision.

B. Hydrodynamic characteristics analysis

To assess the hydrodynamic performance of the HTC and ensure its compatibility with various transmission systems, it is essential to first establish performance evaluation criteria. The HTC based on the \(SR\) has three performance evaluation criteria, torque ratio \((TR)\), capacity constant \((CC)\), and efficiency \((\eta)\),

\[
SR = \frac{N_T}{N_P},
\]

where

- \(N_T\) is the monopole torque of the turbine
- \(N_P\) is the pump torque
The incorporation of thermal effects, as depicted in Fig. 11, markedly enhances the predictive accuracy of hydraulic performance. Particularly noteworthy is the significant improvement in the prediction accuracy of \( CC \). Notably, at a speed ratio of 0.4, the maximum accuracy of \( CC \) prediction is achieved, exhibiting a reduction in error by 6.02%. Moreover, enhancements in prediction accuracy are also observed for \( TR \) and \( g \), with respective improvements of 1.65% and 1.74%. Notably, the consideration of thermal effects leads to a more precise depiction of the viscosity–density properties within the flow field compared to isothermal conditions. Consequently, the incorporation of thermal effects results in a reduction in the average viscosity of the flow field, consequently mitigating the circulating mass flow rate, particularly evident at lower \( SR \) where increased oil flow rates yield larger \( CC \) predictions, as illustrated in Fig. 12.

### C. Measurement point pressure analysis

The performance and simulation accuracy of fluid machinery can be evaluated by flow channel pressure characteristics. The test data at the pressure measurement points after the system has stabilized are analyzed, noise reduction on the dynamic pressure results is performed, and the time-averaged absolute pressure results from the P1 and P2 pressure sensors are obtained for comparison and analysis. As shown in Fig. 13, the numerical model accurately obtains the trend of pressure with regard to \( SR \). The pressure results obtained at the measurement points after considering thermal effects are slightly higher than no thermal results, and the maximum error is reduced from 14.61% to 14.25%. The consideration of the thermal effect contributes to the construction of a more accurate blades surface pressure field.

### D. Temperature analysis of measuring points

The temperature results of the measurement points at the inlet and outlet of the stator shroud are shown in Fig. 14. The temperature calculation error is higher at high \( SR \), and the maximum error appears at the T2 measurement point (\( SR = 0.8 \)), which is 14.3%. According to the experiment, the temperature of the oil increases as it flows through the stator channel, and the temperature difference is inversely

\[
TR = \frac{T_r}{T_p},
\]

\[
\eta = \frac{N_p \times T_p}{N_r \times T_r} = SR \times TR,
\]

\[
CC = \frac{T_p}{N_r^2 D^3}.
\]
proportional to the SR, and the maximum temperature difference is 2.61 °C. The CFD does not specify the temperature difference at the temperature measurement points well, presumably because the steady-state model fails to capture the temperature gradient distribution due to the structure evolution details of the vortex of oil flow.

V. FLOW FIELD DESCRIPTION

A. Overall temperature field analysis

Unlike the closed-loop HTC model, the thermal effect requires calculation of flow characteristics across the entire cycle domain, the cross section of the circulation speed vector field is shown in Fig. 15. The internal oil is supplied by the pump station and injected at a higher pressure from the test jig inlet into the gap between the pump wheel and the stator. Due to the high rotational speed of the pump wheel, a low-pressure area is formed at the inlet, drawing the oil into the pump, where it undergoes the main circulation through the three impellers. During the circulation process, part of the circulating oil also leaks into the central leakage area. Part of the oil flows out of the wheel set through the gaps between the turbine and the stator and returns to the pump station from the outlet. The overall circulation of
the internal flow field ensures a reasonable flow field pressure level to inhibit the occurrence of cavitation and also carries out the internal high-temperature oil through internal circulation, the temperature of the flow field is stabilized.

The overall thermal effect result of the flow field is shown in Fig. 16(a). Because of the oil circulation inside the impeller generates a large amount of heat, the oil temperature at outlet is significantly higher than the inlet, among which there is a local low temperature area in the flow field in the wheel set except for the gap between the pump and the stator. Together with the average temperature of the inlet, outlet, and wheel set field in Fig. 16(b), the overall temperature level of the circulation flow field in the wheel set is close to the flow field of the outlet flow channel.

B. Wall thermal characteristics analysis

The impeller temperature distribution characteristics under the stall condition ($SR = 0$) and the flow field circulation characteristics are provided in Fig. 17. Since the low-temperature oil from the inlet enters the pump wheel at the gap between the pump and stator, the overall temperature distribution gradient of the pump is larger, and there is an obvious low-temperature region at the shroud near the inlet. The oil temperature rises rapidly after entering the circulating flow field of the wheel set, so the overall temperature distribution of the turbine and stator walls is relatively uniform.
Due to the large degree of blade twist of the turbine and stator, the high flow velocity of oil in the pump results in its thermal effects being stronger than those in the flow fields within other impellers, while the blade surface produces a temperature gradient significantly higher than that of shroud and hub. Wall viscous thermal effect tends generate a lot of energy into heat, hence resulting in the blade tail heat accumulation is significantly greater than that at the head; in the blade head of the direct fluid impact position, there is a local low temperature distribution.

C. Analysis of fluid heat flow effects

The evolution of the flow field between blades is shown in Fig. 18. As shown in Fig. 18(a), it can be observed that at low SR, the high-temperature, low-viscosity oil exhibits high flow velocity, leading to various degrees of flow separation on the suction sides of both pump and turbine blades. As the SR increases and the oil viscosity decreases, the flow separation on the turbine blade suction side disappears, and small-scale flow separation appears on the stator pressure side and the pump wheel suction side. This indicates that the temperature changes in the flow field caused by thermal effects have a significant impact on the velocity field.

The eddy viscosity distribution is shown in Fig. 18(b). Due to the large N, there is a large oil kinetic energy out of the pump, and the eddy viscosity is larger near the inlet after entering the turbine. After the oil passes through the vortex area, certain dissipation occurs, resulting in lower eddy viscosity. Corresponding to the high eddy viscosity area leads to a larger heat conversion rate and transfer to the downstream of the blade, so the turbine blade downstream surface temperature is higher; the eddy viscosity value characterizes the turbulent heat conversion efficiency. As the SR increases, the global value of oil eddy viscosity decreases, so the overall thermal conversion efficiency decreases and the overall temperature rise of the flow field decreases. The torque of blades is generated by the pressure difference between the suction and pressure sides of the blades. The higher pressure difference in turbine blades compared to pump is an intrinsic mechanism for their increased torque. Considering thermal effects has improved the computational accuracy of the flow field between blades, leading to more precise pressure difference results, thereby enhancing
FIG. 17. Wall temperature distribution ($sr = 0$).

FIG. 18. Inter-blade flow field visualization results.

(a) Relative velocity vector  (b) Eddy viscosity distribution  (c) Pressure distribution
the predictive capability for hydraulic performance. The pressure difference also decreases with the increase in SR, resulting in a lower overall impeller torque, which is also consistent with the trend of the hydrodynamic characteristics (Fig. 11).

Figure 19 shows the dynamic viscosity and shear distribution at a blade surface of turbine wall. The viscosity effect between the boundary layer oil close to the blade surface and the blade wall lead to the generation of wall shear stresses. A high viscosity region exists at the hub side for different operating conditions, which also leads to a reduction in shear in this region, creating the localized low temperature region in Fig. 17. The increase in SR suppresses the global flow velocity of the oil (Fig. 18), which further leads to a reduction in the dynamic viscosity of the blade wall surface. Consequently, the viscous resistance between the oil and the blade wall decreases, ultimately resulting in a reduction in the overall energy loss in the flow field. The analysis of shear force indicates the main areas where thermal effects occur on the blade surfaces, providing a theoretical basis for research on low-energy-consumption torque converters based on blade surface structure design.

In addition, Fig. 20 shows the contours of local static entropy production at different SR. At the analyzed working conditions, the static entropy near the blade suction surface and at the pump and stator is larger. When the SR is 0 and 0.4, the global static entropy distribution is basically the same. Nevertheless, when the SR is 0.8, the global static entropy decreases significantly, while the overall enthalpy of the flow field is relatively evenly distributed. The previous analysis shows that the decrease in SR affects the energy loss area. The above-mentioned research results also help to explain the trend of the overall thermal effect of the flow field.

In the context of fluid dynamics within a workspace, the medium is subject to the effects of inertial forces, viscous forces, frictional forces, centrifugal forces, and Coriolis forces, leading to secondary flow such as breakage and vortices. The spatial characteristics of eddy generated by turbulence are characterized by vortex structure, enabling the characterization of the spatiotemporal evolution process of internal turbulent transition. In this study, the Q criterion is applied to capture the vortex core, and its definition is as follows:

$$Q = \frac{1}{2} (||\Omega||^2 - ||S||^2).$$

(14)

To capture a better resolution of the vortex structure, the isosurface of $Q = 2.2 \times 10^6 \text{s}^{-2}$ is established. Figure 21 illustrates the distribution of vortex structures within the workspace, highlighting the SST
model's capability to capture complex vortex structures. The primary motion in the HTC is the main circulation between the turbine and pump, which is also the working principle of the HTC as a hydraulic device. To better study the flow mechanisms, the workspace in Fig. 20 is magnified. Overall, the pump rotating speed and the constraints of the blades lead to rotational effects in the flow structure. Unstable flow phenomena, such as separation vortices at the blade head and secondary vortices in the wheel chamber, are more clearly captured, describing the circulation between the turbine and pump. Around the pump blades, the phenomenon is more complex, manifesting in a spiral motion, which also leads to the generation of fluid kinetic energy loss within the pump wheel, resulting in a rapid temperature rise. Within the turbine and stator, mainly large-scale vortices are present, leading to thorough mixing of the internal fluid, and hence, a more even distribution of flow field temperature.

FIG. 20. Static entropy under different SR.

FIG. 21. Evolution of vortex structures in the working chamber.
VI. VALIDATION

The reliability of the multi-physics field numerical model is verified by test research, and impellers with different blade shapes were designed and manufactured (Fig. 22). The hydraulic performance and thermal characteristics of another two wheel sets (P2, T2, S1 & P3, T2, S1) were analyzed numerically and tested experimentally. The analysis results are shown in Fig. 23. The accuracy of the characteristic prediction results of different wheel sets is high, among which the highest error of CC are 4.72% (P2T2S2) and 3.95% (P3T2S2) at stall operation, and the highest error of heat flow characteristic prediction are 10.2% (P2T2S2, SR = 0.3) and 12.2% (P3T2S2, SR = 0.3).

The temperature rise of a HTC under the same working conditions is positively correlated with its energy capacity. Moreover, as the energy capacity increases, the internal oil circulation speed of the HTC increases, leading to the generation of complex secondary flows, which results in an increase in the error of thermal effect analysis. Meanwhile, comparing with Fig. 10, the temperature rise prediction results of the low-CC HTC are lower than the experimental results at higher SR working conditions (for P2T2S1: SR > 0.5; for P3T2S1: SR > 0.4). This is because the proportion of the thermal effect of bearings and clearance flow fields in the low-energy HTC with a smaller temperature rise increases. Therefore, the study shows that the calculation model in this paper has higher accuracy in predicting the thermal effects of low-energy HTC. At the same time, the following aspects should be considered to enhance the accuracy of the model: (1) consider the thermal effects generated by the disk friction losses and structural friction heat of the HTC and (2) consider using large eddy simulation model to capture vortex structure, constructing a more accurate flow field.

VII. CONCLUSIONS

This paper proposed a multi-physics field numerical calculation model for the thermal effect of viscosity oil flow field in HTC and investigated the internal flow field characteristics under various operating conditions and validated the numerical model with experimental data. The main conclusions are as follows:

(1) The macroscopic thermal effects and micro measurement points temperature of the numerical model were calculated and validated. The results show that as the speed ratio increases, the temperature rise at the outlet of the flow field decreases, with the maximum calculation error being 9.79%. The maximum error of the measurement points temperature is 14.3%. The highest temperature rise is 45.13 °C. From the perspective of the flow field temperature distribution results, the outlet temperature is closer to the temperature of the circulating oil.

(2) The consideration of flow field thermal effect constructs a more accurate CFD model with variable viscosity, which effectively improves the prediction accuracy of hydraulic performance. The prediction accuracy for capacity constant is improved by 6.02%. The maximum absolute pressure error at the blade measurement point is reduced to 14.25%.

(3) Energy dissipation caused by wall losses mainly occurs in the near shroud wall and the head of the blades, leading to high wall shear stresses in the corresponding areas and the formation of downstream high-temperature regions. The study provides a theoretical basis for the design of drag reduction mechanisms.

(4) The dissipation of kinetic energy caused by turbulence in the internal flow field is also a reason for the occurrence of thermal effects. The consideration of thermal effects in constructing inhomogeneous flow field captures a more accurate three-dimensional flow mechanism, explaining the formation mechanism of flow field thermal effects, to enhance the depth of research on related flow mechanisms.

ACKNOWLEDGMENTS

This study was supported by China Scholarship Council (Grant No. 202306030181), the National Natural Science Foundation of China (Grant No. 51805027), Beijing Institute of Technology Research Fund Program for young Scholars (Grant No.
The authors have no conflicts to disclose.


\textsuperscript{1}C. Hong, S. Qiao, and A. A. Shah, "Influence of oil temperature on pressure distribution and flow force of valve core," \textit{Int. J. Hydromechatronics} \textbf{6}, 342 (2023).


\textsuperscript{27}X. Deng, "A mixed zero-equation and one-equation turbulence model in fluid-disk thrust bearings," \textit{J. Tribol.} \textbf{146}, 034101 (2024).


32. J.-C. He, Y. Bao, and X. Chen, "Turbulent boundary layers in thermal convection at moderately high Rayleigh numbers," Phys. Fluids 36, 025140 (2024).


