Analysis of internal temperature distribution and its total parasitic heat leak for a cryogenic cold Compressor

X B Dong¹², J Shang¹², H Su¹², C B Wei¹² and J H Wu¹²

¹ State Key Laboratory of Technologies in Space Cryogenic Propellants, Technical Institute of Physics and Chemistry, Chinese Academy of Sciences, Beijing, China
² University of Chinese Academy of Sciences, Beijing, China

E-mail: dongxinbo15@mails.ucas.edu.cn

Abstract. Cold compressors are the core component needed for the development of a facility for large-capacity refrigeration at superfluid helium temperatures. The compressor impeller is operated at extremely low temperature and low pressure which involves technical issues including material properties at low temperature, heat transfer, failures of the actuating motor resulting from overheating and performance penalty due to heat transfer from the warm side of the machine to the cold side. To address these issues, it is important to understand the temperature distribution inside the housing of the machine. To analyze the temperature distribution, an integrated numerical model is created on the basis of the model of the cold compressor structure and then simulated through computational fluid dynamics. The actual temperature rise of the cold compressor is measured at two temperature measuring points and compared with the temperatures predicted by the simulation to validate the accuracy of the model. The results show that the internal flow field around the motor and bearings of the cold compressor is completely turbulent and the total parasitic heat transfer to the cold gas stream is closely related to the rotating speed of the high-speed motor.

1. Introduction
The cold compressor (CC) is an established, economically viable means of obtaining superfluid helium temperatures in large scale cryogenic systems. The CC maintains vacuum pressure in a superfluid reservoir (normally 1.5 kPa at 1.8 K or 3.1 kPa at 2 K) by drawing a stream of extremely low temperature and low-pressure helium gas and returning it to the refrigerator at higher pressure. With the requirement of higher magnetic fields, it has been a trend to adopt cold compressors, based on cold compression (multistage cold compressors all the way up to atmospheric pressure) or mixed compression (a combination of cold compressors in series with ambient-temperature sub-atmospheric compressors) [1],
to obtain a larger capacity of superfluid helium. Several cold compressor systems have been successfully utilized in particle accelerators, such as LHC, CMTF, DESY and ESS.

As shown in reference [2], the CC is a high-speed centrifugal compressor which is normally driven by high-speed induction motor. The rotor is supported radially by two identical active magnetic bearings (AMB) and axially by a single degree of freedom (DOF) AMB thrust bearing. At present, due to the need for the electrical system to operate at ambient temperature, cold compressors are normally made up by two parts: the warm end with motor and bearings and the cold end with the compressor impeller and volute. Considering that the high-speed motor and active magnetic bearings generate large losses at high speed and high frequency, the warm end of the cold compressor will inevitably generate a large amount of heat and thus cause a considerable temperature rise. On the other hand, the cold end is operated under extremely low temperature such that an over 300 K temperature difference is formed between the two ends. Under running conditions, several technical issues are involved including material thermal contraction, failures of the actuating motor resulting from overheating and performance penalty due to heat transfer from the warm to the cold end. To address these issues, it is necessary to analyze the temperature distribution inside the cold compressor and develop methods to control the key temperatures. However, most of open source literatures about cold compressor only involve researches on impeller optimization [3], rotor dynamics design [4], performance testing [2] and system control [5].

In this paper, the design of a particular cold compressor which actuated by a high-speed motor is taken as a basic example. A CFD (computational fluid dynamics) method is adopted to calculate and analyze the temperature distribution and its heat transfer behavior. To validate the accuracy of the simulation model, two temperature sensors are fitted to the motor and AMB stator respectively, and then monitored on the test system in our lab. The influences of the rotating speed on the temperature rise and total heat transfer are simulated and discussed.

2. Set up for simulation

2.1. Physical model and meshing

Referring to related articles [6], the methods of thermal analysis of machines mainly involve the lumped parameter method, the finite element method and the CFD method. The lumped parameter model or the finite element methods were largely used in the past to describe the motor practical operation. However, the accuracy of these models largely depends on how much effort the developers take in defining the heat transfer paths and in assessing the applicability of empirical correlations used in the calculations. To avoid these limitations, the CFD method is adopted in this paper to identify the parasitic heat leak, taking advantage of its capabilities of computing a detailed internal flow characteristic and temperature distribution.

The simulation domains are depicted in detail in figure 1. This model couples fluid-solid behavior and contains several solid domains and fluid domains. These domains are modeled individually and then assembled together. During the process, some unimportant structure features are neglected including void volumes, feed-throughs and other minor details. To reduce the heat transfer between the driving motor end and the cryogenic end, two pieces of polyurethane foam are designed here, taking its excellent characteristics of thermal insulating performance. A liquid nitrogen cooling jacket is designed between flange and volute, locating on the middle of the support portion, to absorb heat transfer from warm end.
The meshing job is completed with the help of commercial software package ANSYS Meshing while the computational model of flow and heat transfer process in the machine is set up by ANSYS CFX. The SST model is used as the turbulence model for the helium domain as it was designed to give highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients [7]. Figure 1 additionally displays parts of the structured grids. To get a relatively accurate result while avoiding an excessive number of cells in the overall mesh, most grids are generated by subdividing the domain in small volumes and adopting hexahedral mesh grids. The total mesh number is approximately 8.22 million, with over 3.4 million for the helium gas domain while the rest are in the solid domains.

![Figure 1. Physical model of CC and part of its schematic of computational grid](image)

2.2. Boundary conditions

The cold compressor is installed on the cold box flange, with the housing of the warm end placed in the atmosphere. During operation, all of the motor and AMB losses, including iron hysteresis and eddy-current (iron loss), copper wire resistance, magnetic hysteresis, and windage, are transferred as heat to the cold helium, cooling water and ambient surroundings.

The iron loss and copper loss of the machines are calculated by the motor manufactory by using Motor-CAD. The detailed parameters are given in Table 1. Among these, the rotor losses mainly come from the air gap magnetic resistance change which is caused by the space-harmonics, the time-harmonic of winding armature and magnetic field. The method of calculating eddy current losses in the sleeve, protective jacket and PM can be found in reference [8]. The AMB heat losses are calculated based on reference [9]. In the simulation analysis, these losses are modeled as heat sources and are converted to heat per unit of domain [10].

| Table 1. The Iron loss and copper loss of the machine generated by the motor and AMB |
|--------------------------------|--------------------------------|--------------------------------|--------------------------------|--------------------------------|
| Rotor                          | PM motor at 60 kpm P/watt     | single radial AMB P/watt      | axial AMB P/watt               |                               |
| magnetic hysteresis            | 13.8                          | magnetic hysteresis           | 3.15                          | thrust disk eddy current 1.2 |
| Protective jacket iron         | 0.85                          | eddy-current 8.7              |                               |                               |
| Stator                         | copper iron 6.8               | copper iron 6.8               | copper iron 23.9              | copper 5.8                    |
|                                | 37.8                          |                                |                               |                               |

According to empirical correlations obtained from large space natural convection measurements, an approximate heat transfer coefficient (7.5 W·m⁻²·K⁻¹) is applied to the outside surface of the housing.
wall [11], while the ambient temperature is set to a fixed value. The other boundary conditions are set as follows: the temperature of the impeller surface is 3.4 K; the inlet temperature of cooling water is 288 K; the cooling water flow rate is 6 liter/min; The liquid nitrogen cooling boundary is set as a constant temperature. A surface-to-surface radiation model based on the discrete transfer model (DTM) method is utilized to calculate heat radiation transfer [12].

2.3. Material properties
The temperature gradient between the warm and cold ends of the machine is relatively great, and involves the shaft, heat insulation material, a copper thermal sink, the impeller and other structures. Of special importance, the thermal conductivity of these materials changes greatly from ambient temperature to cryogenic temperature, so temperature dependent thermal conductivities are employed to obtain an accurate temperature distribution. The thermal conductivity values used in the CFD simulation are shown in Figure 2. Temperature gradients within the warm end are relatively small, so that fixed thermal conductivity is used for the structural material. The warm end values are shown in Table 2.

![Figure 2. The thermal conductivity of different materials](image)

| Material name | coefficient of thermal conductivity $W/(m\cdot K)$ | application | Material name | coefficient of thermal conductivity $W/(m\cdot K)$ | application |
|---------------|---------------------------------|-------------|---------------|---------------------------------|-------------|
| XG832H carbon fiber | 9.8 | permanent magnet | copper | 20.5 | winding |
|                | 0.7 | projective jacket | silicon steel | 22.6 | stator |
| 42CrMo insulating paper | 44 | shaft | steel 304 | 12 | sleeve |
|                | 0.19 | insulation | |

3. Experimental verification
In order to verify the accuracy of the CFD model, a performance test was conducted with the designed cold compressor. The cold compressor is mounted to the specially designed cold box which is intended to provide the desired steady-state and transient operating conditions. During tests at different conditions, the temperatures at certain positions inside the cold compressor are recorded. Space constraints inside
the housing limit the number of temperature sensors. Four PT-100 sensors and two Cenox-1050 sensors are installed as illustrated in Figure 3.

The stator winding accounts for most of the heat generation, so one PT-100 sensor (accuracy: ±0.1 K) is located inside the axial AMB stator winding (point 1). Another PT-100 (point 2) is placed on the wall of the motor stator in the upper portion of a silicon steel lamination. To monitor the temperature of the cold end, two Cenox-1050 sensors (accuracy: ±0.05 K) are installed in the inlet pipe (point 4) and outlet pipe (point 5) respectively. The other two PT-100 sensors (accuracy: ±0.1 K, points 3 and 6) monitor the cooling water and liquid nitrogen temperatures.

During the test, the motor is accelerated to 32,000 rpm while the temperatures are monitored. The final result of point 1 and point 2 is obtained after the stable operation of the cold compressor without large temperature fluctuation (within 1K). From Figure 4, it can be seen that the temperature of the motor stator and the AMB stator agree well with the simulation model. The motor and AMB temperature increases associated with increase of ambient temperature are well captured. Over all five conditions, the difference between the component temperature and ambient is also displayed and the maximum difference between measurement and simulation is less than 5 K. These comparisons fully demonstrate the accuracy of the numerical model, so that the following section of results and discussions focuses on insights from the numerical model.

![Figure 3. View of the test rig and the measuring points](image1)

![Figure 4. Comparison of simulated results and experimental data](image2)
4. Result and discussion

4.1. Temperature distribution and flow characteristic

Figure 5 illustrates the simulated temperature distribution of the entire solid structure in the plane of the center section with an ambient temperature of 293.5K. From the contour of temperature distribution, the highest temperature is 317.5 K, located in the motor rotor, near the upper end of the motor stator. The entire rotors and stators of the motor, upper AMB and thrust bearing, as well as the top cover of the housing, are close to this temperature. Along the vertical direction, an over 300 K temperature difference exists between the warm and cold ends.

Figure 6 illustrates the flow characteristic of the helium gas. The internal flow is entirely turbulent with strong swirl driven by rotor rotation. Helium gas in the narrow air gaps reaches an average speed of 80 m/s in the swirling direction. The high-speed rotation enhances the heat convection transfer between rotor and stator elements, so as to carry most heat source losses away from the gaps. However, due to viscosity effects and the blockage of fixed stators, the swirl speed decreases considerably near the housing. These two effects, along with gravity and buoyancy force effect are the three main reasons why the heat transfer from the rotor to the housing and cooling water at the warm end is not good.

4.2. Heat transfer to cryogenic end

According to the previous research in the literature, heat transfer from the warm end to the cold end significantly affects the compressor performance. In terms of quantity, approximately a 20 W heat is equivalent to 1% decrease in isentropic efficiency for the compressor size being considered [2]. To address this, it is necessary to evaluate the heat transfer rate between the warm section and the cold section portion under different work conditions. For this purpose, off-design conditions are simulated in the model with the rotating speed ranging from 0 to 40,000 rpm.

Furthermore, to evaluate the heat transfer to the cold end, a heat balance model is applied to the numerical CFD data. The model can be presented in the following form:

\[ Q_{\text{total}} = Q_{\text{volute}} + Q_{\text{shaft}} + Q_{\text{impeller}} \]  

(1)
where $Q_{\text{total}}$ represents the total amount of heat transfer to the cryogenic helium; $Q_{\text{volute}}$ is the heat transferred through the volute; $Q_{\text{shaft}}$ is the heat conduction of the contact surface between the shaft and impeller. $Q_{\text{impeller}}$ represents the heat flow through the back wall of impeller, including heat convection, heat radiation and heat conduction.

The specific value of $Q_{\text{volute}}, Q_{\text{shaft}}, Q_{\text{impeller}}$ at different rotating speeds are shown in Figure 7. It can be found that the total amount of heat transfer increases with the increase of rotating speed. When the design speed is 32000rpm, the total heat transfer is about 32.9 W. Observing each bar in Figure 7, the amount of $Q_{\text{volute}}$ and $Q_{\text{shaft}}$ takes up almost 24 W and it almost not varies with the speed change. That is due to $Q_{\text{volute}}$ is independent of speed when ignoring the small increase in warm end temperature from drag losses relative to the overall warm-cold temperature difference. For $Q_{\text{shaft}}$, the difference between the maximum value and the minimum value of heat conduction is also merely 1W. The reason for this phenomenon is largely due to the existence of the hollow part, which plays an important role in increasing the thermal resistance of the shaft, making the heat conduction of the shaft not sensitive to the temperature difference between warm and cold end.

The $Q_{\text{impeller}}$ varies greatly with the increase of rotation speed. Especially in the case of 36000 rpm, $Q_{\text{impeller}}$ accounts for almost 32 percent of $Q_{\text{shaft}}$. This is mainly caused by the forced convection between the insulation material bottom wall and impeller back plate wall. Considering the existence of viscous force when the impeller rotates at high speed, the low temperature helium near the back wall of impeller is dragged to rotate. As the limitation of air gap space, helium gas causes a strong vortex flow here and forms a strong forced convection between the two sides, thus carrying more heat to the impeller.

![Figure 7. The total heat flow varying with rotating speed](image)

**5. Conclusion**

In this paper, the temperature distribution of a cryogenic cold compressor is studied by CFD simulation analysis and experimental measurement. The main conclusions that can be drawn from this work are:

A coupled fluid-solid CFD model is utilized to simulate the temperature field of cold compressor. The maximum difference in key temperatures at the warm end between experiment and simulation is less than 5 K. The overall temperature distribution and helium gas flow characteristics are obtained, revealing an over 300 K temperature gradient along the shaft. The flow in the narrow gaps between rotors and stators of the motor and magnetic bearings at the warm end is turbulent, and the swirl speed decreases considerably along the radial direction outward from the narrow gaps. The total heat transfer...
to the cold end is about 32.9 W at the design speed. The influence of rotational speed on the temperature rise at the warm end and on the total heat transfer from the warm end to the cold end are analyzed.

6. References
[1] Philippe L and Laurent. T 2014 CERN Yellow Report Cooling with Superfluid pp 453-476.
[2] Yoshinaga S, Honda T, Shimba T and Ozaki S 2005 IHI Engineering Review 38 pp 45-50.
[3] Geng M F, Song Y T, Cheng A Y, Feng H S, Zhang Q Y and Jiang Q F 2018 FUSION ENG DES 131 pp 84-89.
[4] Kameno H, Kubo A, Takahata R and Ueyama H 2000 KOYO Engineering 158 pp 16-20.
[5] Asakura H and Kündig A 2002 Proceedings of ICEC19 (Grenoble, France) pp 681-686.
[6] Boglietti A, Cavagnino A, Staton D, Shanel M, Mueller M and Mejuto C 2009 IEEE T IND ELECTRON 56(3) pp 871-882.
[7] Bardina J E, Huang P G and Coakley T J. 1997 28th Fluid Dynamics Conference pp 2121.
[8] Huang Z Y, Fang J C, Liu X Q and Han B C 2016 IEEE T IND ELECTRON 63(4) pp 2027-2035.
[9] Zhang Y, McLoone S, Cao W and Qiu F Y 2017. IEEE T ENERGY CONVER 32(4) pp 1468-1478.
[10] Lamghari-Jamal M I, Fouladgar J, Zaim E H and Trichet D 2006 IEEE T MAGN 42(4) pp 1271-1274.
[11] ANSYS CFX Help, Computational Fluid Dynamics, Release 15.0, ANSYS Inc.
[12] Karanth K V, Manjunath M S, Sharma N Y 2011 World Congress on Engineering pp 2355-2360.

Acknowledgments
The authors would like to thank and acknowledge Prof. Junjie Li and Xiangdong Xu for their supports of establishing the test platform and providing constructive comments and advice.