Numerical study of unsteady flow in centrifugal cold compressor

Ning Zhang[1], Peng Zhang[2], Jihao Wu[1], Qing Li[1]*

1. Key Laboratory of Cryogenics, Technical Institute of Physics and Chemistry, Chinese Academy of Sciences, Beijing 100190, China
2. School of Energy and Power Engineering, Huazhong University of Science and Technology, Wuhan 430074, China

Background

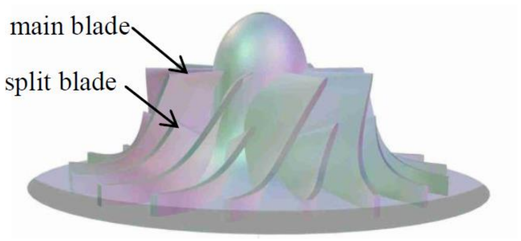
The large cryogenic helium refrigeration system is the supporting technology of advanced science and technology. With the development of accelerator technology and requirement of higher beam energy and luminosity, the cryogenic and superconducting technology is applied widely to the accelerator construction and lots of large helium cryogenic systems were established successively. The centrifugal cold compressor is the key equipment of the superfluid helium cryogenic system. The saturate helium bath is evacuated by the cold compressor, from 120 kPa (4.4 K) to 1.5 kPa (1.8 K).

Generally speaking, cold compressors enter the surge region in the case of low mass flow or high pressure ratio, while they enter the choke region in the case of high mass flow or low pressure ratio. Since a surge causes an intense oscillation of pressure, it is dangerous in particular.

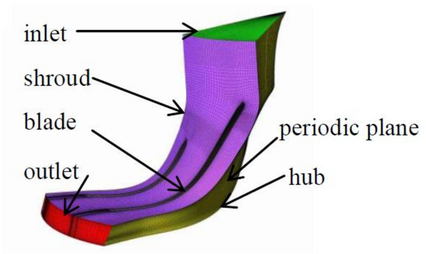
With the development of computer technology, numerical simulation technology has become an important and effective tool for studying the internal flow of centrifugal compressor. In order to obtain the installed range of cold compressor, unsteady flow in the case of low mass flow or high pressure ratio is investigated by the ANSYS CFX.

CFD model of cold compressor

In order to obtain high efficiency, backward-skewed blades consisting of main and split blade were used in the impeller of cold compressor such as shown in Fig. 1. The impeller diameter was 100 mm and the rotating speed was 44 krpm. Based on the ANSYS Workbench, 3D design and simulation of cold compressor was implemented. The 3D designing model was accomplished by Blade-Gen and computational grid was made by Turbo-Grid as shown in Fig. 2. The number of nodes was 300 000. After that, the CFD analysis was solved by ANSYS CFX software. In the CFX, the blades, hub and shroud were defined as adiabatic walls with the appropriate rotational velocity. The boundary condition at the inlet was always set as the total pressure and total temperature and the boundary at the outlet was set at the mass flow rate accordingly. The cryogenic helium below the atmospheric pressure was used as the refrigerant gas.



Model of impeller



Meshing of periodic channel

Conclusion

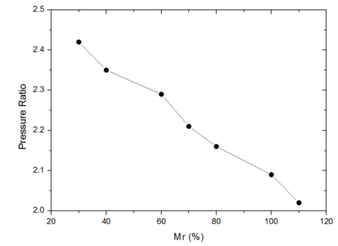
The numerical analysis has been done to understand the effect of mass flow on the performance of the centrifugal cold compressor. From the results of the numerical analysis, it can be deduced that the pressure ratio increases with the decrease in reduced mass flow. With the decrease of the reduced mass flow, backflow and vortex are intensified near the shroud of impeller. The unsteady flow will not only increase the flow loss, but also damage the compressor.

Result and Discussion

Simulation of the off-design performance of the centrifugal cold compressor was considered for seven off-design reduced mass flow: 30%, 40%, 60%, 70%, 80%, 100% and 110% under the design speed 44 krpm. The reduced mass flow is defined as follow:

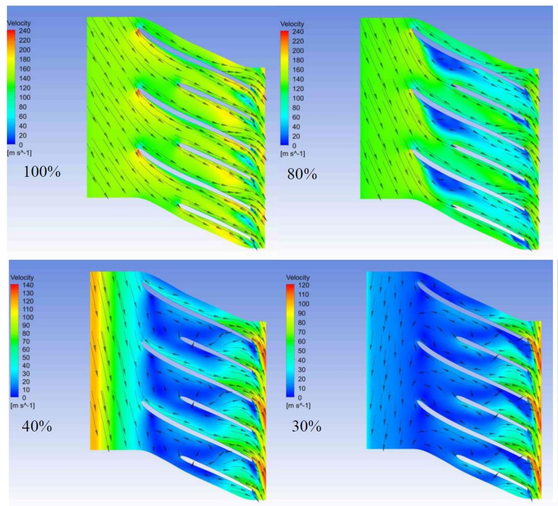
$$Mr = \frac{M}{M_{ref}} \cdot \frac{P_{ref}}{P} \cdot \sqrt{\frac{T}{T_{ref}}}$$

where M, P and T are mass flow, inlet pressure and inlet temperature of impeller and M_{ref}, P_{ref} and T_{ref} are corresponding parameters at design condition.

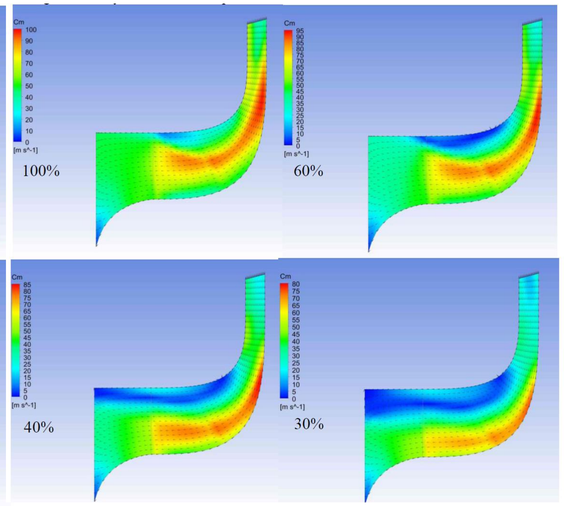


Total pressure ratio of impeller under the different reduced mass flow conditions:

When the centrifugal compressor is working in different conditions, the internal flow of impeller is different. At the design condition, the relative velocity of inlet is the same direction as the leading edge of the blade and helium stream enters the impeller passage smoothly. But when the inlet mass flow is less than the design mass flow, helium stream enters the impeller passage at the positive angles of attack and it causes flow separation on the back of the blade. For the inertia force, the flow separation is increased and it will not only increase the flow loss, but also damage the compressor. The surge occurs in the centrifugal compressor at a small flow rate. Such unsteady flows which often represent the backflow and vortex are intensified near the shroud of impeller.



Velocity vectors at 80% span under the different reduced mass flow conditions



Vector of area averaged Cm on meridional surface under the different reduced mass flow conditions