Research on Simulation and Prediction of Turbocharger Compressor Performance

Xiangling Liu[1*], Zhengwu Liu[2]

[1] College of Information and Mechanical and Electrical Engineering, Hunan International Economics University, Changsha, Hunan, 410205, China

[2] College of Mechanical and Electrical Engineering, Hunan Institute of Communications Engineering, Hengyang, Hunan, 421001, China

*corresponding author

Keywords: Compressor stage performance, Simulation, Prediction study

Abstract: Fine/Turbo software is used to perform numerical simulation and flow field analysis on the compressor stage of the H145 turbocharger. The modeling and analysis process of the turbocharger compressor stage are introduced, and the influence of different simulation methods on the final simulation results is analyzed. The results show that it is feasible to use CFD numerical simulation and flow field analysis to predict the performance of the compressor stage, which can meet the requirements of multi-scheme performance optimization and selection of turbochargers.

1. Introduction

Turbocharging technology is one of the technical means for modern engines to increase power and meet emission regulations. A good match between turbochargers and engines is an important link to give full play to the advantages of superchargers. In the process of matching the performance of the turbocharger and the engine, traditionally, the multi-scheme physical prototype and the engine are used to carry out the bench matching test, and the cost of the matching process is too high, which can no longer meet the requirements of the engine matching. In recent years, with the rapid development of CFD simulation technology, turbocharger performance prediction technology has made considerable progress. The turbocharger configuration process has changed from traditional single test verification to predictive analysis combined with test verification. In the turbocharger performance prediction, accurate compressor-level performance prediction is the key link. Among the various compressor performance prediction methods, the performance prediction of the compressor-level complete compressor based on Nume-ca software using computational fluid dynamics (CFD) methods can analyze the internal flow field distribution of the compressor, and perform matching the preliminary optimization of the turbocharger can improve the efficiency and success rate of the physical turbocharger configuration process.

2. Establishment of Simulation Model

The H145 turbo-compressor is selected as the research object. The compressor stage includes a compressor impeller, a vane diffuser and a compressor volute. The compressor impeller trim value is 0.524. Compared with the vaneless diffuser, the compressor stage simulation of the vane diffuser is increased by one level, and the grid processing is more complicated than the vaneless diffuser. The establishment of the 3D solid model adopts software such as Proe and UG. Through the combined use of multiple software, solid and curved surface models are established in the computer: the curved surface models of the compressor impeller long blades and splitter blades, the impeller hub and the wheel cover channel Model, curved surface model of vane diffuser, solid model of compressor volute cavity. Grid establishment steps: divide the long and short blades of the compressor impeller and the single-channel mesh model of the diffuser blade; divide the volute channel mesh; copy the single-channel mesh of the impeller blade and the diffuser blade into a full-

  DOI: 10.25236/iccem.2021.028

circle mesh model; The whole-week model mesh is merged with the volute mesh. The final three-dimensional grid of the compressor level is shown in Figure 1.

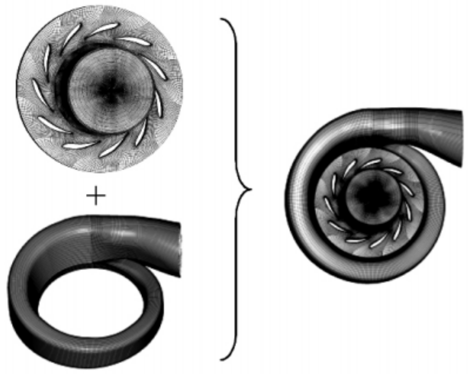


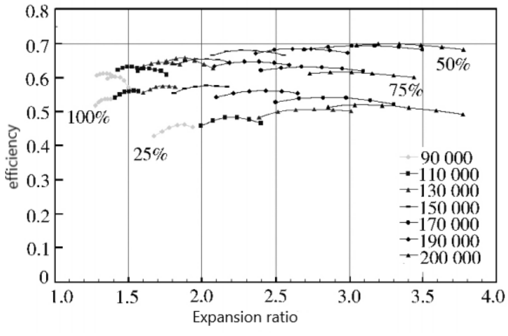
Fig.1 Turbine Stage 3d Grid Formation Process

3. Simulation of Compressor-Level Model

The compressor-level full flow field calculation adopts Numeca's Fine/Turbo software. Two calculation models are designed and the models in the two states are calculated. The calculation domain is from the inlet of the compressor impeller to the outlet of the diffuser, and then to the outlet of the volute. The calculation process gives the boundary conditions of the compressor inlet and outlet. The inlet is given a total temperature of 298K, a total pressure of 100kPa and an absolute airflow angle, and the outlet is given a compressor outlet back pressure. The solid wall adopts the boundary conditions of impermeability, non-slip and adiabatic. In the calculation process, according to the characteristics of the compressor flow characteristic curve and the actual design parameters of the compressor, the outlet static pressure is first set at each speed, so that the compressor is in a blocking condition, and then the outlet back pressure is gradually increased, which is close to the surge. The situation is changed to a given compressor outlet flow rate, which gradually approaches the surge boundary.

4. Analysis of Calculation Results

In this example, the simulation of the turbine end with 4 openings of the adjustable nozzle blades was mainly carried out. At each opening, the calculations were made for 6 speed working conditions (90×103 r/min, 110×103 r /Min, 130×103r/min, 150×103r/min, 190×103r/min, 200×103r/min) flow characteristics. Each calculation model uses the same inlet and outlet boundary setting conditions, and a series of different flow points are calculated according to the preset inlet pressure under each speed working condition. According to the data calculated by the simulation, the expansion ratio-similar flow characteristic curve and the expansion ratio-efficiency characteristic curve of the turbine under different opening degrees are drawn (as Figure 2).



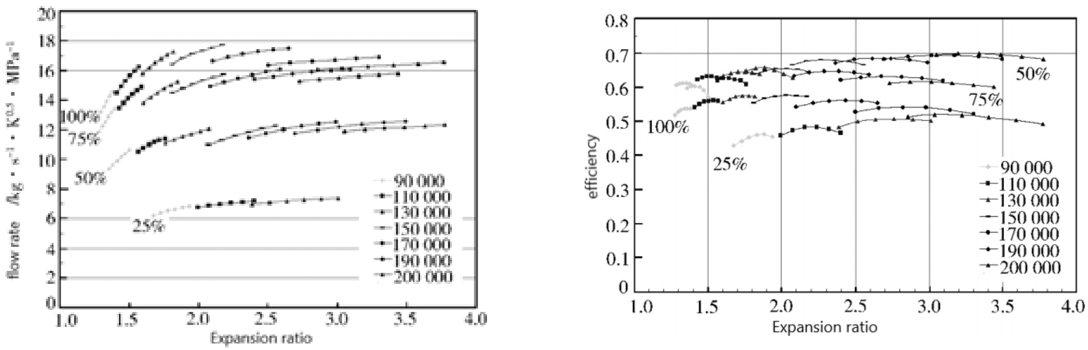


Fig.2 Turbine Simulation Map Diagram

It can be seen from the flow comparison graph that there is an error between the numerical simulation result and the experimental test result, and the measured flow is smaller than the calculated result, but the change law of the two is the same, and both show linear changes. For the difference in flow, there are two main reasons: First, the simplification of the simulation model turbine hub, the manufacturing error of the turbine box channel, the modeling error of the simulation turbine box channel, the modeling and manufacturing errors of the turbine blades and nozzle blades, etc. It will cause differences between the simulation data and the test data; secondly, the theoretical value is used for the opening in the simulation calculation, and there is an error in the control and calibration of the nozzle vane opening during the test. Since the error is linear, appropriate corrections can be made to make the simulation results meet the quantitative analysis requirements in terms of flow. It can be seen from the efficiency comparison graph that the error between the numerical simulation results and the experimental test results is relatively small at high speeds, and as the speed decreases, the errors gradually increase. It can also be seen from the efficiency graph that the efficiency varies non-linearly. For the difference in efficiency, the main reasons are the following: the simulation calculation result is the theoretical efficiency of the turbine, and the experimental test data is the efficiency of the turbine output to the compressor, which reduces the transmission of the turbine through the shaft to the compressor compared to the theoretical efficiency. The mechanical efficiency does not consider the heat transfer loss of the turbine to the oil during the test. At the same time, the turbocharger has a large mechanical transmission loss at low speed, which shows that the simulation efficiency and the measured efficiency at low speed have a large error; on the other hand, simulation Necessary model simplifications were carried out during the calculation, such as the assumption of impermeable, non-slip, and adiabatic boundaries, which resulted in the simulation efficiency being greater than the measured efficiency.

Because the mechanical efficiency and heat conduction energy loss of the same series of turbochargers are similar, the simulation data is corrected based on the measured data of the same series of turbochargers. The modified turbine end performance simulation results can be used as engine simulation calculation input to meet the scheme. Assess the needs. Combined with the evaluation of the optimization schemes of the turbine and nozzle blades, the multi-scheme performance optimization prediction design of the turbine end of the supercharger was finally completed. Another important function of the numerical results of the turbine performance prediction is that it can be used as the basis for the optimization of the three-dimensional performance design of the turbocharger. The result shows the comparison of the relative Mach number cloud diagrams at the turbine end under different opening and different speed conditions. It can be seen from the cloud diagrams of each section that the calculated values of the compressor stage's static temperature, static pressure, airflow relative to the Mach number, etc. all follow the process of increasing the amount along the airflow direction. The temperature field distribution conforms to the principle of distribution along the flow channel. The temperature increases along the gas flow compression direction, and the volute exit temperature is the highest; the static pressure distribution shows that the pressure gradually increases along the volute radius, the pressure distribution is uniform, and there is no local sudden change; the air flow path is relative to the Mach The number distribution is uniform, the airflow speed of the impeller basically works at subsonic speed, there is no vortex, no shock wave is generated in the channel, and the airflow operation requirements of the flow field are met. The comparative analysis of the calculation points near the surge point, the highest efficiency point, and the near clogging point can well reflect the change process of the internal flow field of the turbocharger with the flow rate.

5. Conclusion

In this study, CFD numerical simulation was used as a means, and the H145 turbocharger was taken as an example to predict the compressor stage characteristics under different working conditions and conduct a preliminary analysis of its internal flow field. The results show that the CFD analysis method based on Numeca on the compressor-level full ternary flow can clearly show the internal flow, that is, the distribution of fluid pressure, temperature and velocity. Comparing the single-channel simulation results with the full-cycle compressor-level simulation simulation, the analysis shows that when the compressor-level model is simulated and calculated, in order to shorten the calculation time, the single-channel model can be calculated to predict the compressor performance of the complete model, which is much more convenient. The optimization of this kind of plan, after the plan is confirmed, the compressor-level flow field calculation can be carried out for the whole week. The research results show that this method of CFD numerical simulation and flow field analysis of the compressor stage is feasible as a compressor stage performance prediction, and it can meet the design requirements of turbocharger compressor stage performance optimization prediction.

Acknowledgment

Outstanding youth project of Hunan Education Department: Research on flow field simulation and key parts design optimization of turbocharger of gasoline engine(17B150).

References

[1] Chen Gang, Yan Jianfei, Zhang Jindong. Research on simulation and prediction of turbocharger compressor stage performance. Automotive Engine, 2011, 000(006): 67-70.

[2] Zhang Haifeng, Wu Zhenglin. Numerical simulation of casting process of turbocharger casing. Commodity and Quality: Academic Observation, 2013, 000(007): 300-300.

[3] Ban Haibo, Gao Bo, Huang Jiawei. Simulation analysis of turbocharger compressor impeller cooling. Railway Locomotives and EMUs, 2020, No.555(05): 5+20-21+35.

[4] Zhang Zhen. Numerical simulation study of turbocharger dynamic performance. Automotive Engine, 1995, 000(006): 48-52.

[5] Xiao Xiong, Liu Xiangling, Wu Tong, et al. CFD analysis of gasoline engine turbocharger compressor based on NUMECA. Light Industry Science and Technology, 2017(4): 63-64.

[6] Jiang Guohe, Li Taotao. Fluent-based turbocharger compressor flow field simulation and noise analysis. China Water Transport (second half of the month), 2011, 11(007): 78-80.