Air Cooling Analysis for Oil-free CO2 Reciprocating Compressor

De Xiang Chen
Hefei General Machinery Research Institute;State Key Laboratory Of Compressor Technology, cdx97@tom.com

Air cooling analysis for oil-free CO2 reciprocating compressor

De Xiang Chen [1,2]*

[1] Hefei General Machinery Research Institute,
Hefei, Anhui Province, China
cdx97@tom.com

[2] State Key Laboratory of Compressor Technology,
Hefei, Anhui Province, China

* Corresponding Author

ABSTRACT

It is critical to limit cylinder wall temperature for oil-free reciprocating compressor to work properly and reliably. The purpose of this paper is to show how to obtain cylinder wall temperature in oil-free reciprocating compressor design. Temperature field of a two stage oil-free CO_2 reciprocation compressor with air cooling is computed numerically by two-way FSI simulation with conjugate heat transfer. Air flow around the compressor and heat conduction in the compressor are computed by 3D CFD code and FEA code respectively. Heat information is transferred between the two codes at FSI interface which is the compressor outside surface.

1. INTRODUCTION

Spaceflight application requires carbon dioxide (CO_2) reduction system to remove CO_2 and convert it into useful resources. CO_2 compressor is a key device for such system that is to provide vacuum for molecular sieve desorption as well as CO_2 compression for delivery to the Sabatier reactor (Richardson et al., 2015). Depending on the status of molecular sieve and Sabatier reactor, the work condition of this CO_2 compressor changes widely. Suction pressure ranges from 3kPa to 100kPa while discharge pressure ranges from 100kPa to 600kPa. Since the chemical reaction in Sabatier assembly cannot tolerate the existence of lubricant, the compressor need to be oil free.

We are designing an air-cooling oil-free reciprocating type compressor for such application. Due to the lack of lubricant, oil-free compressors usually use piston rings made of non-metallic materials with low friction coefficient and self-lubricating properties, such as Carbon- or graphite-filled PTFE and PEEK. In general, the wear rate of these materials increases with increasing temperature. Our preliminary tests showed that the wear of certain graphite-filled PTFE against titanium alloy cylinder increased by 45% when the temperature increased from 70°C to 115°C while the load and friction speed unchanged. During the operation of reciprocating compressor, cylinders typically generate considerable amounts of heat. The heat is due to the work of compression and the friction between the piston rings and the cylinder wall. Unless some of this heat is dissipated, undesirably high operating temperatures will occur. Therefore, limiting the cylinder wall temperature of an oil-free CO_2 reciprocating compressor is critical to its normal and reliable operation.

From the compressor design point of view, designers need to accurately assess the temperature of the cylinder wall to determine the amount of cooling air required. The dissipation of heat from cylinder to environment is a coupling process of heat conduction and air flow. The heat transfer process may be divided into three stages. The first stage is heat generation at cylinder wall. Heat generates by friction and gas compression and transfers to the inner wall of the cylinder by conduction and convection. The heat generated is depending on the suction and discharge pressure and the amount of gas delivered. The second stage is heat conduction in the cylinder liner and cylinder block. The heat transferred is depending on thermal conductivity of the material and thermal contact resistance between the cylinder liner and the cylinder block. The third stage is convection on the outer surface of the cylinder. The heat transferred is depending on the flow velocity and temperature of cooling air. Because of the complexity of the cylinder block

geometry, it is difficult to calculate the temperature distribution with a simple model if there is no test data for calibration.
In recent years, numerical tools like CFD and FEA are used more and more in compressor engineering to shorten compressor development time and reduce development costs. Posch et al. (2016) present a numerical model to predict the temperature field in a hermetic reciprocating compressor for household refrigeration appliances. The model combines a high resolution 3D heat conduction formulation of the compressor's solid parts, a 3D computational fluid dynamics (CFD) approach for the gas line domain and lumped formulations of the shell gas and the lubrication oil. Chen (2013) set up a numerical model to compute airflow field and temperature field according to the theory of fluid-solid coupled heat transfer, and solved the overheating of micro-compressor cylinder cover. Birari et al. (2006) use CFD tool to reduce physical experiments and the total development time of compressor, with R404A as an alternate refrigerant. They changed the existing compressor design based on the CFD simulation results, resulting in reduced number of prototypes for achieving intended results.
In this work, cylinder wall temperature of the oil-free CO_2 reciprocating compressor is calculated by two-way FSI simulation with conjugate heat transfer.

2. CONSTRUCTION OF A CO_2 RICIPROCATING COMPRESSOR

The cross-section of a CO_2 compressor is shown in Figure 1. It is a reciprocating piston type compressor, two-stage compression, three vertical arranged cylinders, including two identical first stage pistons and one second stage piston with 20mm stroke. The whole machine was placed in a soundproof enclosure that has cooling air passage with a diameter of 80mm, as shown in Figure 2. The cooling air enters from one side and flows out from the other side. The heat dissipation fins are arranged on the cylinder block to enhance the convection heat transfer.

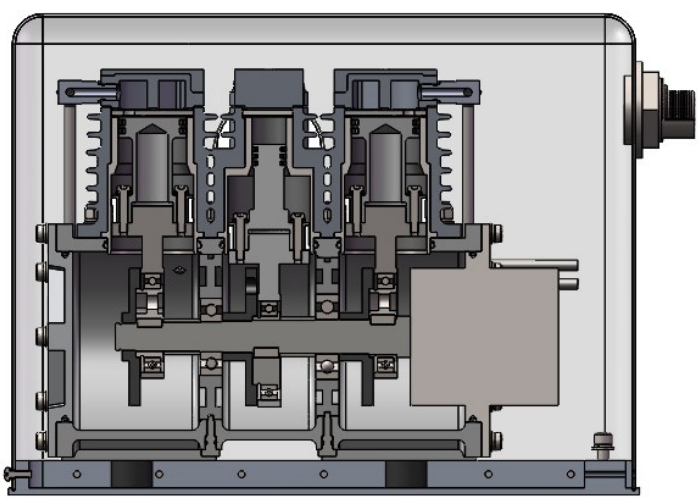


Figure 1: cross-section of a reciprocating compressor

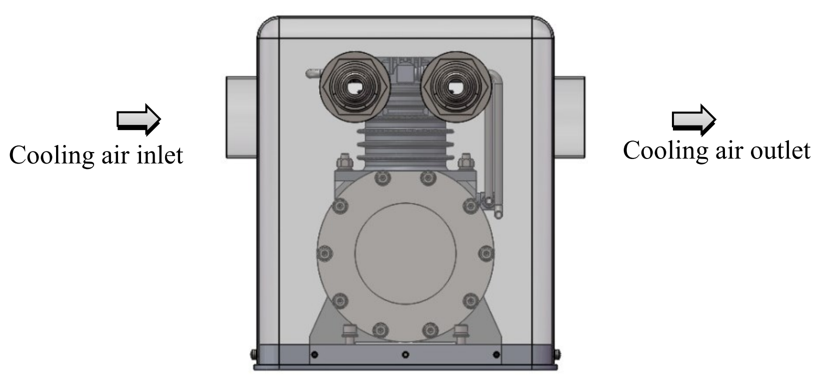


Figure 2: Schematic diagram of cooling air flow

3. ANALYSIS PRECEDURE

3.1 Numerical model

The geometry of the actual compressor is very complex. In order to reduce the difficulty of mesh generation and the consumption of computing resources, the geometry of the compressor is simplified, as shown in Figure 3. The simplified model removes details such as fillets, chamfers, etc., eliminating parts such as screws, washers, crankshafts, and moving parts. The simplified model consists of two domains: the flow domain and the solid heat transfer domain. The thermal coupling boundary between these two domains is the outer surface of the compressor. Figure 4 shows the image of the space mesh division. The mesh of fluid domain is refined near the surface of the compressor. The total number of fluid mesh is about 950,000, and the number of solid mesh is about 120,000.

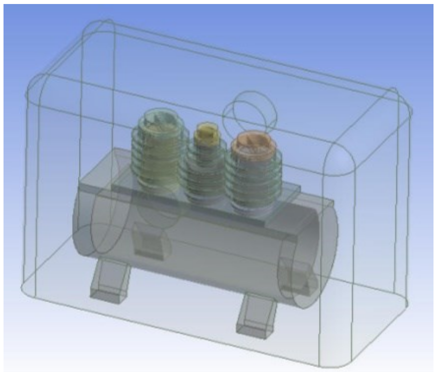
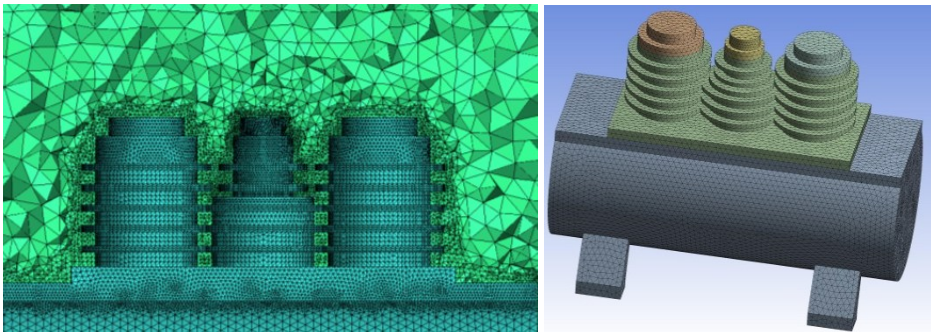


Figure 3: Simplified geometry of compressor



(a) Fluid domain (b) Solid domain

Figure 4: Space Division of flow field and solid body

3.2 Analysis workflow

It is assumed that the compressor has reached a thermal equilibrium under certain operating conditions, and the heat generated is equal to the amount of heat taken by the cooling air. The overall simulation workflow is shown in Figure 5. The air flow in the fluid domain is computed by ANSYS FLUENT, while the temperature distribution in the solid domain is obtained by ANSYS steady state heat conduction module. Thermal coupling data is transferred between the two domains at the coupling interface. During the flow simulation, the coupling surface acts as a wall type boundary. The temperature value of the boundary is calculated from steady state thermal analysis via system coupling. At the end of the flow calculation, the convective heat transfer coefficient and the near wall temperature data are transferred to the thermal calculation program. In the steady state thermal analysis, the coupling surfaces act as the convection heat transfer boundary, the convection heat transfer coefficient and bulk temperature are calculated from the flow field. At the end of the thermal calculation, the coupling surface temperature is transferred to fluent.

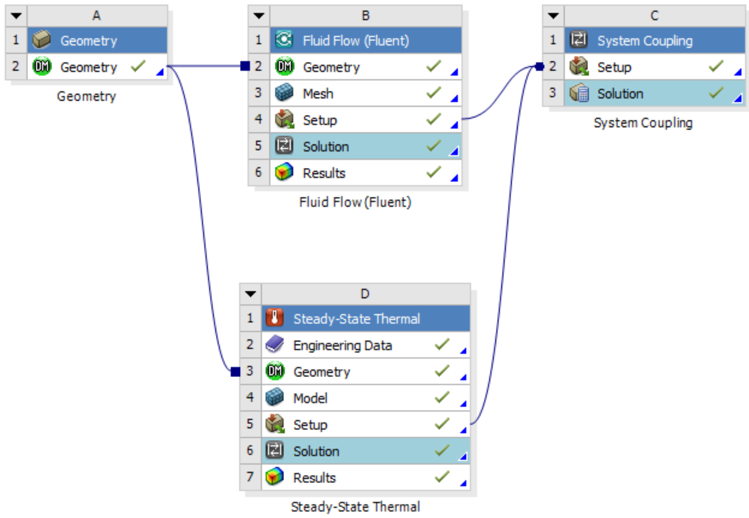


Figure 5: Air cooling analysis workflow

3.3 Analysis parameters

The crankcase, cylinder block and cylinder head are made of aluminum alloy. The thermal conductivity is 150W/m^{2}K. The cylinder liner and valve plate are made of titanium alloy and its thermal conductivity is 21.9W/m^{2}K. The inner walls of the cylinder liner, crankcase, and cylinder head are considered as heat flow boundary. The heating power is related to the inlet and outlet state of the compressor. The estimated range is 125W~250W. The heating power distribution at 250W is shown in Figure 6. Cooling air inlet temperature and air volume are design variables, temperature 25°C~30°C, air flow 60l/min~180l/min. A total of seven different cases were calculated during the simulation analysis, as shown in Table 1.

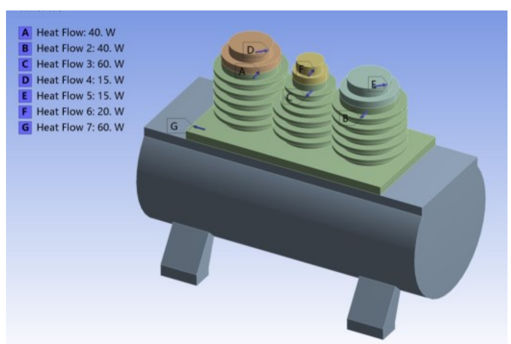


Figure 6: Heat flow boundary (250W)

Table1: Analysis parameters

Case	**Air temperature ℃**	**Air flow l/min**	**Heating power W**
1	25	180	250
2	30	180	250
3	30	180	200
4	30	90	250
5	30	60	250
6	30	60	200
7	30	60	125

4. RESULTS AND DISCUSSION

Of all the 7 cases computed, case 5 gives the worst condition, and the corresponding cylinder liner wall temperature should be the highest. Table 2 lists the maximum temperature of the inner wall surface of the cylinder liner under different calculation parameters. The result of case 5 is the largest, which is consistent with the prediction.
Figure 7 shows the trend of the max cylinder liner wall temperature with cooling air flow at a heating power of 250 W and a cooling air temperature of 30°C. This figure can be used for the selection of cooling fan, controlling the temperature of cylinder liner wall to not exceed the limit value. Figure 8 shows the relationship between the max cylinder liner wall temperature and the heating power at a cooling air temperature of 30°C and a cooling air flow rate of 60 l/min. This graph can help to predict the max temperature at the cylinder liner wall.

Table2: Max temperature at cylinder liner wall

case	1	2	3	4	5	6	7
stage 1 °C	52.5	57.5	52.0	68.7	77.9	68.4	53.9
stage 2 °C	60.7	65.7	58.6	77.5	85.7	74.7	58.0

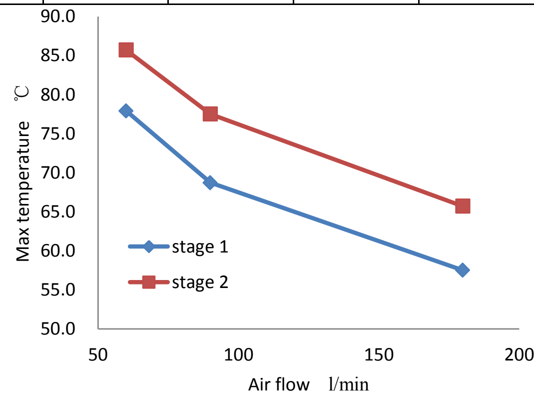


Figure 7: Cylinder liner wall temperature vs air flow (air temperature 30℃, heating power 250W)

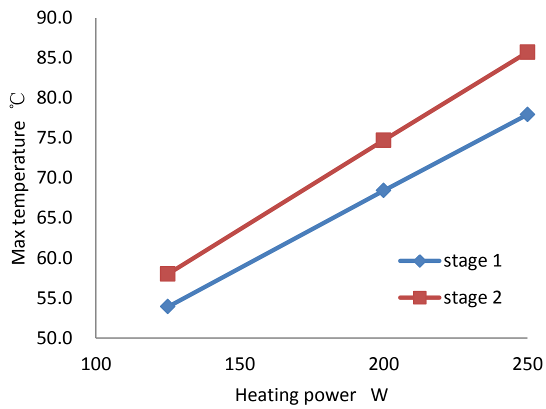
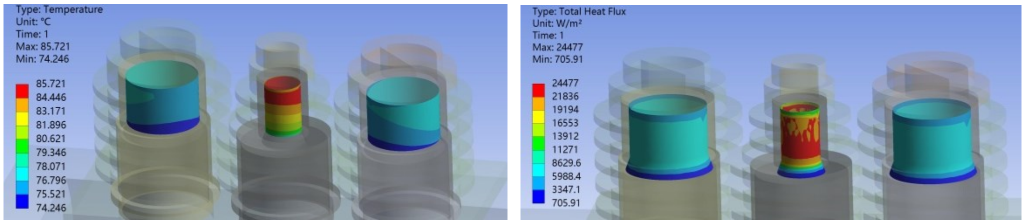


Figure 8: Cylinder liner wall temperature vs heating power (air temperature 30℃, air flow 60l/min)

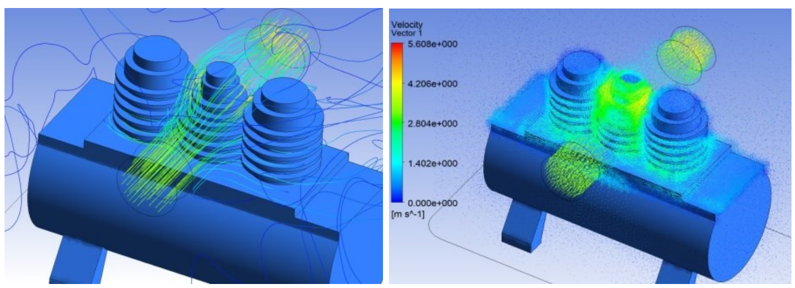
Figure 9 shows the temperature and heat flux distribution at inner wall of cylinder liners for case5. It can be told from the figure that the temperature of the stage-2 cylinder liner is higher than that of the stage-1 cylinder liner, which is due to the small cylinder bore and small heat dissipation area of the stage-2 cylinder. The temperature distribution of the stage-2 cylinder liner is symmetrical bilaterally, but there is a temperature difference of about 1℃ in the flow direction of the cooling air. The windward surface temperature is lower, while the leeward surface temperature is higher. The temperature distributions of the two cylinder liners are similar, with the lower side close to the secondary being lower and the other higher.



(a) Temperature (b) Heat flux

Figure 9: Temperature and heat flux at cylinder liner wall

Figure 10 shows the stream line and velocity around the cylinder block for case 5. Figure 11 shows the distribution of convective heat transfer coefficient on the outer surface of the compressor for the same case. The stage-2 cylinder is exposed to cooling air directly, and scoured intensively by air. Hence the convection heat transfer capacity is stronger than that of the two stage-2 cylinders. Figure12 shows the distribution of air temperature. Because the air outlet is arranged where the stage-2 cylinder is facing, the air velocity away from the cylinder is relative low, and the convection heat transfer intensity is less intensive. There is an eddy zone in the air flow at both ends of the compressor, which may lead to an increase in the temperature inside the enclosure.



(a)stream line (b)velocity

Figure 10: Flow field

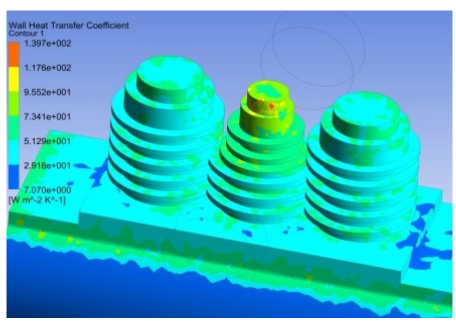


Figure 11: convective heat transfer coefficient

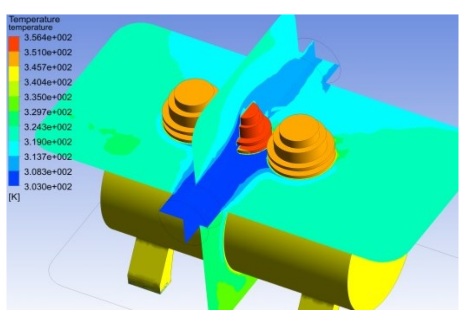


Figure 12: Air temperature

5. CONCLUSION

We are designing an air-cooled, oil-free compressor for a CO_2 reduction system. FSI analysis is applied to obtain the temperature distribution inside the cylinder liner. Air flow around the compressor and heat conduction in the compressor are computed by 3D CFD code and FEA code respectively. Since the design is still at early stage, the model is simplified in the calculation. In spite of this, the trend between the maximum inner wall temperature and cooling air temperature, flow rate, and heating power was obtained in the analysis, which provided useful help for the thermal design of the compressor when no experiment data available. In the future, air cooling analysis should be carried out based on detailed model, and verified with prototype.

REFERENCES

Birari, Y.V., Gosavi, S.S., Jorwekar, P.P. (2006). Use of CFD in Design and Development of R404A Reciprocating Compressor. *Proceedings of the International Compressor Engineering Conference at Purdue,* Purdue University, IN, USA, paper C072.

Chen, J. (2013). Micro-compressor Air-cooling System Based on Fluid-solid Thermal Coupled Simulation. Journal of Southeast University, 43(1), 65-70.

Posch, S., Hopfgartner, J., Heimel, M., Berger, E., Almbauer, R., Stangl, S. (2016). Thermal Analysis of a Hermetic Reciprocating Compressor Using Numerical Methods. P P*roceedings of the International Compressor Engineering Conference at Purdue*, Purdue University, IN, USA, paper 1215.

Richardson, T.M.J., Jan, D., Hogan, J., Bohrer, R., Marson, D.K. (2015). Carbon Dioxide Compression, Storage, and Delivery Trade Assessment. *The 45th International Conference on Environmental Systems*, Bellevue, WA, paper065.

ACKNOWLEDGEMENT

The authors would like to acknowledge the Anhui Natural Science Foundation for their financial support with project No: 1708085ME128.