Preliminary Analysis of Supercritical CO_2 Compressor with CFD

Yongju Jeong, Jeong Ik Lee*
KAIST
**Corresponding author : Jeongiklee@kaist.ac.kr*

1. Introduction

A supercritical CO_2 Brayton (S-CO_2) cycle is a variation of gas Brayton cycle, which utilizes supercritical state CO_2 as a working fluid. One particular advantage of S-CO_2 cycle is that compression process occurs near the critical point, and this reduces compression work significantly compared to a conventional air Brayton cycle [1]. S-CO_2 cycle can be widely applicable to many heat sources such as concentrating solar power, fossil fuel, and nuclear. Especially, high density of S-CO_2 and simple layout of the system facilitate small to mid-scale power generation such as small modular reactor (SMR).

A compressor operating under S-CO_2 condition shows different behaviors with air compressor, and requires unconventional design and analysis methods. It is necessary to conduct an experiment for a compressor operating under S-CO_2 condition. However, due to high density of S-CO_2, the compressor tends to be considerably small, which makes it difficult to measure local flow variables. For this reason, CFD analysis should be applied. Previously, Cho designed and performed a compressor testing based on an 1D method [2]. This paper aims to analyze the fluid behavior inside the compressor.

2. Methods and Results

2.1 Geometry

Normally, a compressor consists of impeller, diffuser and volute. Compressor impeller mainly increase fluid velocity, and diffuser increase pressure by converting velocity effect to pressure effect. Volute is a connector between compressor and outlet pipe.

The target compressor is unshrouded compressor, so it has stationary casing wall and rotating impeller blade with a thin gap placed between them.

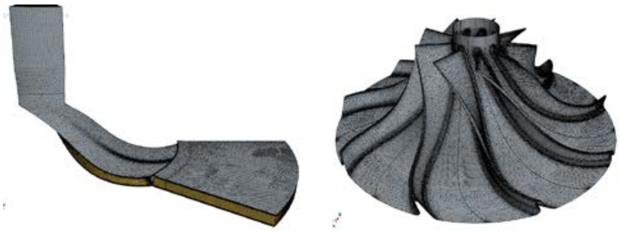

Fig. 1. Geometry of analysis domain (original / with multiplication)

In this study, impeller and diffuser are covered for the modeling. Because of the axi-symmetric nature of the compressor, it is possible to apply periodic boundary condition. The compressor has nine blades, so the analysis domain was divided into one-ninth of the compressor. Left side of Fig 1 shows one-ninth of the compressor, and right side shows periodically multiplicated geometry for impeller.

2.2 Model setup & Results

Table. 1. Summary of analysis model

	Type	Value
Mesh	Polyhedral	4E05
Turbulent model	k-w SST	
Inlet BC	Total pressure	80 bar / 40 ºC
Outlet BC	Static pressure	91 bar
Rotation	Moving reference frame	40,000 RPM

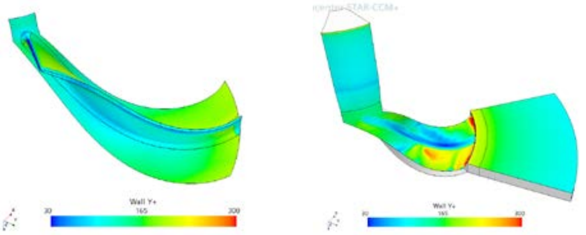

Fig. 2. y+ value of compressor

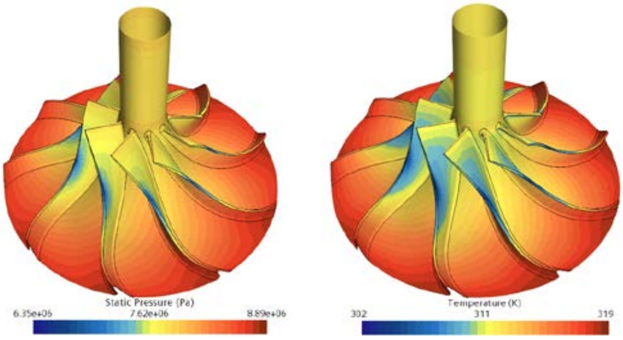

Fig. 3. Temperature and pressure of compressor

For the analysis, STAR-CCM+ [3] was used. Polyhedral mesh was utilized, and the number of mesh was 400K. For the turbulent model, k-w SST model was used. The model has become popular for turbomachinery application, because it has good prediction capability under adverse pressure gradients and separating flow [4]. In terms of boundary condition, total pressure condition was imposed at the inlet, and static pressure condition was prescribed at the outlet.

Rotating effect was modeled as moving reference frame. Summary of the prepared model is presented in Table 1.

After simulation was converged, y+ value was checked to predict the shear stress effect on the wall. As a result of mesh refinement, most of the wall had the y+ value between 30-300 as shown in Fig. 2, where wall function approach is available. Also, Fig 3 shows temperature and pressure field at the compressor impeller. Overall, pressure and temperature tend to rise from inlet to outlet. Blue colored field, indicating lower value of temperature, imply intense local acceleration of the flow.

3. Future work

In this paper, the target compressor geometry and physical model were prepared and reported. For an effective computation, the analysis domain was divided into one-ninth, and the wall function approach was adopted. As a future work, in order to confirm validity of the model, it is essential to produce a compressor performance map with respect to mass flow rate and rpm so that the produced data can be compared against experimental data. When the validity of model is guaranteed, it is planned to investigate specific design parameter effect and compressor operating limit.

4. Acknowledgement

This research was supported by Civil-Military Technology Cooperation Program (iCMTC) funded by the Agency for Defense Development – South Korea (17-CM-EN-04).

REFERENCES

[1] Ahn, Yoonhan, et al. "Review of supercritical CO2 power cycle technology and current status of research and development." Nuclear engineering and technology 47.6 (2015): 647-661.

[2] Cho, Seong Kuk, et al. "Optimum loss models for performance prediction of supercritical CO2 centrifugal compressor." Applied Thermal Engineering 184 (2021): 116255.

[3] Siemens, P. L. M. "Simcenter STAR-CCM." Academic Research, Release 12 (2017).

[4] Menter, Florianr. "Zonal two equation kw turbulence models for aerodynamic flows." 23rd fluid dynamics, plasmadynamics, and lasers conference. 1993.