Transactions of the Korean Nuclear Society Spring Meeting
Gwangju, Korea, May 30-31, 2013

CFD Analysis of a Centrifugal Pump with Supercritical Carbon Dioxide as a Working Fluid

Seong Gu Kim[a*], Jeong Ik Lee[a], Yoonhan Ahn[a], Jekyoung Lee[a], Jae Eun Cha[b], Yacine Addad[c]
[a]*Dept. Nuclear & Quantum Eng., KAIST, 373-1, Guseong-dong, Yuseong-gu, Daejeon, 305-701, Republic of Korea*
[b]*Fast Reactor Technology Development Division, KAERI, 305-353, DukJin-Dong 150, Yuseong-gu, Daejeon, Korea*
[c]*Dept. Nuclear Engineering, KUSTAR, PO.box:127788, Abu Dhabi, United Arab Emirates*
**Corresponding author : skim07@kaist.ac.kr*

1. Introduction

The supercritical carbon dioxide (S-CO_2) Brayton cycle is considered as an effective power generation cycle for the next generation nuclear systems. Comparing to power generation systems with water or helium, the S-CO_2 cycle occupies much less volume due to small component size.

The research team is conducting a S-CO_2 pump experiment to obtain fundamental data for the advanced pump design and measure the overall performance of the pump near the critical point. The S-CO_2 pump testing loop configuration is similar to SNL and JAEA testing loop while the operating conditions and focus of experiment are different from other test facilities.

This paper presents the methodology of a 3-dimensional flow analysis for the S-CO_2 pump by using the commercial CFD code.

2. CFD Analysis

2.1 Geometry and meshing

Pump impeller and diffuser geometry provided by the manufacturer was utilized to perform CFD analysis. Fluid domain and grid were generated with ICEM CFD software. This closed impeller has rotating circular shroud and hub on either side of the flatted shape blades. The imported solid model was divided into rotor and stator part and converted into fluid domain. Then, the pump geometry and analysis domain were exported to ICME CFD software and structured to the meshing elements. The geometry is shown in the Figure 1(a).

2.2 Fluid property

To simulate a nonlinear behavior of S-CO_2, a table format properties were generated and coupled to the CFD code. The fluid properties were imported from NIST REFPROP and RGP format was used to implement the properties table into ANSYS CFX solver version 14.5. In this study, RGP tables generated in the range of 233~400K and 1~50MPa with a grid size of 500 by 500 were used for the simulation.

The CFX solver calculates properties by using bilinear interpolation of pressure and temperature. The error between the real property from NIST and calculated values are compared and summarized in Table 1. The point locations were selected in streamwise flow direction as shown in Figure 1(b). Larger errors appeared near the critical point as expected. The results of density, dynamic viscosity and constant heat capacity properties reported relatively high error range up to 6.9%. And the other properties reported low error less than 3.6%.

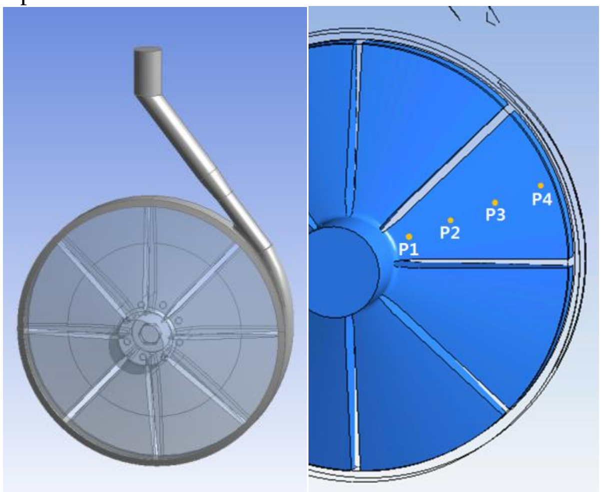

Fig.1.(a)The geometry of S-CO_2 pump with diffuser. (b)Point locations for material assessment.

Table I: Calculated error of S-CO_2 property comparison to real property from NIST REFPROP.

	Density [kg/m^3]	Enthalpy [kJ/kg]	Cp [kJ/kg-K]	Thermal conductivity	Dynamic viscosity
Point i	**311.10K**	**8.363MPa**		[W/m-K]	[Pa-s]
CFX	406.317	361.048	15.152	68.047	2.9186E-05
NIST	387.56	365.71	14.17	65.645	2.7991E-05
% Error	4.840%	1.275%	6.930%	3.659%	4.269%
Point ii	**312.17K**	**8.538MPa**			
CFX	411.725	361.262	13.528	66.444	2.9587E-05
NIST	395.05	365.27	13.043	64.728	2.8501E-05
% Error	4.221%	1.097%	3.721%	2.650%	3.810%
Point iii	**313.03K**	**8.681MPa**			
CFX	415.583	361.53	12.467	65.366	2.9871E-05
NIST	400.65	365.06	12.197	63.998	2.8889E-05
% Error	3.727%	0.9670%	2.214%	2.138%	3.396%
Point iv	**313.83K**	**8.812MPa**			
CFX	418.49	361.97	11.565	64.354	3.0100E-05
NIST	404.79	365.13	11.445	65.356	2.9189E-05
% Error	3.383%	0.8651%	1.051%	1.736%	3.122%

2.2 Problem setup

To predict the turbulent flow, standard k-e model is applied. And the total energy equation with viscous work effect is used for heat transfer model. The boundary conditions were selected to be equal of the test conditions. CFD results are compared to the obtained test data for the validation. The analyses were performed to obtain a characteristic curve of the S-CO_2 compressor.

3. Results

Results of total pressure and velocity at the whole fluid domain in stationary frame are showed in the Figure 3. To obtain an efficiency characteristic curve, mass flow rate at outlet is controlled while other conditions remain constant. Also authors performed grid independence study with variation in the number of grid size. The results shown in Table 2 reported a pump efficiency variations of less than 1.77% and it shows that the value is converging.

To compare the obtained results to the test data, the efficiency curve from the pump manufacturer tested with water and the S-CO_2 pump testing data were marked on the Figure 2 together for comparison.

Table II: Results of grid independence study.

Nodes	Elements	Pressure Ratio	Mass flow rate	Compressor efficiency
94,337	478,876	1.159	2.19kg/s	74.38%
148,199	755,965	1.159	2.19kg/s	75.05%
184,399 (reference mesh system)	920,184	1.158	2.19kg/s	76.15%
244,302	1,287,495	1.156	2.19kg/s	76.73%

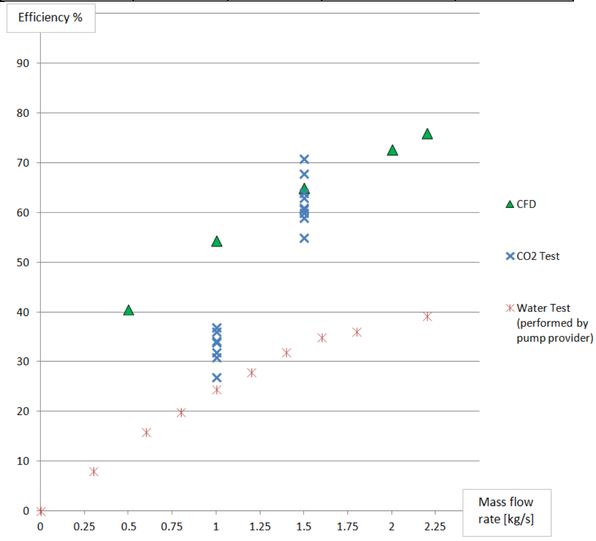

Fig.2. Results of pump efficiency versus mass flow rate.

4. Conclusions

In Figure 2, the results at the 1.5kg/s mass flow rate seems to be close agreement between the CFD efficiency and S-CO_2 test results. In the low mass flow rate of 1.0kg/s, CFD predicted 17~25% higher efficiency than the test result. In the real test facility, the steel structure of pump is not an adiabatic wall and also the mechanical losses such as suction, blade loading and leakage exist in the pump. The reason why CFD analysis showed higher pump efficiency at the low mass flow is the abovementioned losses were excluded from the model. However, as the mass flow rate increases these have less effect on the efficiency. If the heat transfer through the structure and pump losses are applied in the analysis, other losses can be estimated.

From the S-CO_2 pump experiment, more data will be obtained and compared to the CFD analyses under the methodology presented in this paper. After the fluid behavior in the pump are well understood, these analysis results will be used for optimizing impeller for advanced S-CO_2 compressor design in the future. However, it is very encouraging that even at very small mass flow rate the efficiency of S-CO2 pump near the critical point operation is very high compared to the manufacturer water test. The reason behind such phenomenon will be more carefully studied in the future.

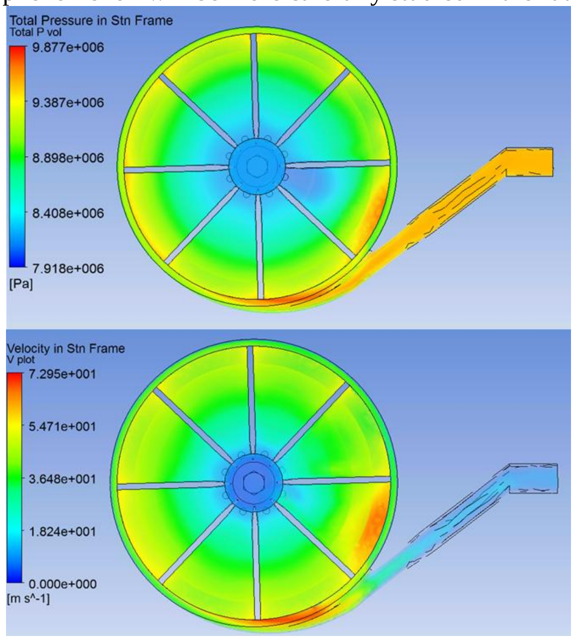

Fig.3. Contour plot of total pressure and velocity.

ACKNOWLEDGEMENT

Authors gratefully acknowledge that this research is supported by the National Research Foundation of Korea(NRF) and funded by the Korean Ministry of Science, ICT and Future Planning.

REFERENCES

[1] Tae W. Kim, Nam H. Kim, Kune Y. Suh, Seong O. Kim, *Engineering Thermo-fluid Dynamics Analysis of Supercritical Carbon Dioxide Turbomachinery for Kalimer-600*, AICFM, 2007.

[2] Jekyoung Lee, Jeong Ik Lee, Yoonhan Ahn, Hojoon Yoon, KAIST, KUSTAR, *Design Methodology of Supercritical CO2 Brayton Cycle Turbomachineries*, Proceedings of ASME Turbo Expo 2012, 2012.

[3] T.A. Munroe, M.A. Zaccaria, W.H. Flaspohler, R.J. Pelton, K.D. Wygant, O.B. Dubitsky, Knolls Atomic Power Laboratory, Concepts NREC, *Fluent CFD Steady State Predictions of a Single Stage Centrifugal Compressor with Supercritical CO2 Working Fluid*, Proceedings of the S-CO2 Power Cycle Symposium 2009, 2009.

[4] ANSYS Inc., *ANSYS CFX-Solver Modeling Guide Release 14.0*, ANSYS Technical paper, 2011.