# Simulation of Pressure Drop for Different Venturi Diameters

Dae-Won Cho[a], In-Cheol Chu[a], Dong Jin Euh[a*]
[a]*Thermal Hydraulics Safety Research Division, KAERI, Yuseong-gu, Daejeon, Republic of Korea*
[*]*Corresponding author: dwcho@kaeri.re.kr*

## 1. Introduction

Korea Atomic Energy Research Institute (KAERI) is developing Sodium-cooled Fast Reactor (SFR) and trying to obtain the approval about SFR design. As the type of SFR is different from the other reactors, it is expected that the hydraulics characteristics such as, pressure drop and velocity distributions should be different. Additionally, it is essential to maintain the pressure drop for various reactor core groups (1-9) which have different flow rates.

Normally, the orifice and venturi can be used to control the pressure drop in the hydraulics system and the diameter of venturi is very sensitive for the pressure drop. Therefore, it is required to maintain the pressure drop between venturi inlet and venturi throat firstly. This study calculates the pressure distribution by computational fluid dynamics (CFD) for various reactor core groups and maintains the pressure drop between venturi inlet and venturi throat within 5% errors. The commercial software Star-CCM+ was used to calculate the pressure distributions.

## 2. Model description

### *2.1 Geometry*

Fig. 1 shows the schematic of venturi used in the simulations where the diameter of inlet (D1), outlet (D1), converge and diverge angles (α, β) are constant. The variable which is different along the cases is the throat diameter of venturi.

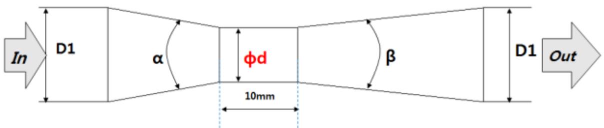



Fig. 1. Schematic of venturi used in the simulations

### *2.2 Meshing scheme*

This study uses 3-D models to calculate the pressure distributions. Polyhedral, thin and prism layer meshes are applied to describe the mass flow inlet, pressure outlet and wall boundaries.

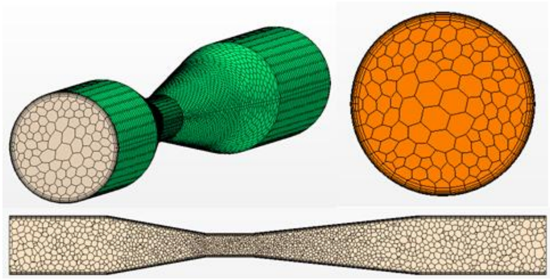


Fig. 2. Shape of mesh used in simulations (overall, transverse and longitudinal sections)

### *2.3 Boundary and initial conditions*

The circle type inlet and outlet faces in Fig. 2 are defined as mass flow inlet boundary and pressure boundary (0 pa) while the cylinder type boundary is the wall boundary. The fluid used in the simulation is water with a density of 983.2 kg/m$^3$ and a viscosity value of 4.67 x 10$^{-4}$ m Pas at 60℃. The mass flow rates for each group are listed in Table 1

Table 1 Mass flow rates used in simulations for various groups

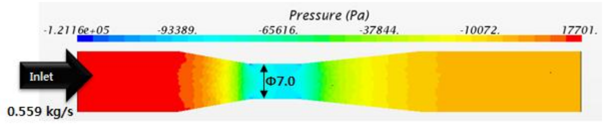
| | Flow rate (kg/s) | | Flow rate (kg/s) | | Flow rate (kg/s) |
|---|---|---|---|---|---|
| Group 1 | 0.559 | Group 4 | 0.454 | Group 7 | 0.327 |
| Group 2 | 0.525 | Group 5 | 0.375 | Group 8 | 0.29 |
| Group 3 | 0.500 | Group 6 | 0.359 | Group 9 | 0.258 |

### *2.4 Simulation models*

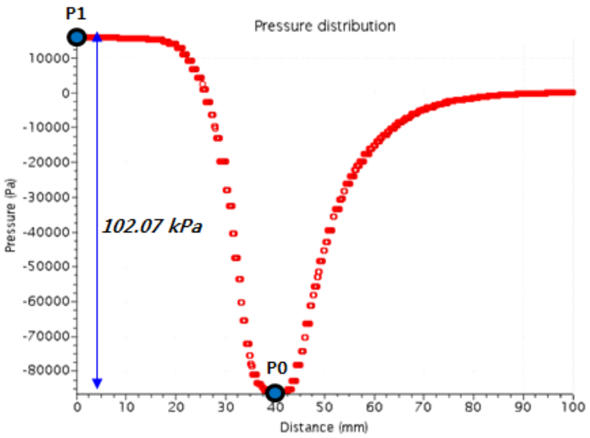
The flow simulations are carried out using a steady state and segregated solver. In this method, Navier-Stokes equations are solved were sequentially using iterative methods until the residual values are less than 1x10$^{-5}$. Among the various turbulence modes, this study uses k-ε turbulence models [1, 2]

## 3. Simulation results

The target of pressure drop between the inlet face boundary and the center of venturi throat is 100 kPa. As the mass flow rates are different along the various groups, it is necessary to control the diameter of throat precisely. Fig. 3 shows the 2D pressure distribution and 1D pressure plot (venturi center) for group 1.




(a) 2D pressure distribution




(b) 1D pressure plot

Fig. 4. Simulation results of group 1 (a) 2D pressure distribution (b) 1D pressure plot

With an effort to match the pressure drop precisely within the small ranges (max. 5% error), it is possible to get the desirable diameter of venturi and the corresponding pressure drop for 9 groups as shown in Table 2

Table 2 Simulation results of pressure drop for various groups

| | Flow rate (kg/s) | Diameter of venturi throat (mm) | Pressure drop, P1-P0 (kPa) |
|---|---|---|---|
| Group 1 | 0.559 | 7.0 | 102.07 |
| Group 2 | 0.525 | 6.8 | 100.41 |
| Group 3 | 0.500 | 6.7 | 99.22 |
| Group 4 | 0.454 | 6.4 | 99.88 |
| Group 5 | 0.375 | 5.9 | 97.17 |
| Group 6 | 0.359 | 5.7 | 102.71 |
| Group 7 | 0.327 | 5.5 | 99.42 |
| Group 8 | 0.290 | 5.2 | 99.10 |
| Group 9 | 0.258 | 4.9 | 100.29 |

## 4. Conclusions

CFD simulation techniques are used to calculate pressure distributions for various groups. Thus, this study finally succeeds to match the pressure drop within the given ranges. The results of the simulations can be used to design the orifices in the next step.

## ACKNOWLEDGEMENT

The authors gratefully acknowledge the support of the Korea government (MSIP) (No. 2012M2A8A2025635).

## REFERENCES

[1] W. K. In, D. S. Oh, and T. H. Chun, CFD Analysis of Turbulent Flow in a Nuclear Fuel Bundle With Mixing Vane, ASME 2002 Pressure Vessels and Piping Conference , American Society of Mechanical Engineers, p. 317-323, 2002

[2] A. M. Vaidya, N. K. Maheshwari, P. K. Vijayan, and D. Saha, Computational study of moderator flow and temperature fields in the calandria vessel of a heavy water reactor using the PHOENICS code, Kerntechnik, Vol. 73, p. 33-40, 2008