Study on the Effect of Branching Outlet Pipe using CFD Method

Jonghark Park [a*], Heetaek Chae [a], Heonil Kim [a], Cheol Park [a]
[a] *Korea Atomic Energy Research Institute, Daedeokdaero 1045, Yuseong Daejeon, Korea*
[*] *Corresponding author: pjh@kaeri.re.kr*

1. Introduction

For a downward flow type research reactor immersed in a pool, coolant passes through the upper guide structure, cools down fuel assemblies in the core, and return to primary cooling loop through outlet nozzle of lower plenum. If the pressure drop of reactor core is too small, flow rates of fuel assemblies in the core may be affected by the flow pattern in the outlet plenum, which depends on the location and number of outlet nozzles.

A uniform distribution of coolant flow in the reactor core is required to ensure a thermal-hydraulic safety. If outlet nozzle number could affect the flow distribution in the reactor core, it should be evaluated and fed into the design of reactor structure.

This is a numerical study to figure out the effect of branching outlet pipe. CFD analyses for two types of outlet plenum are conducted and the predicted results of flow distribution and fluctuation are compared with each other.

2. CFD Modeling and Calculation

2.1Modeling for CFD analysis

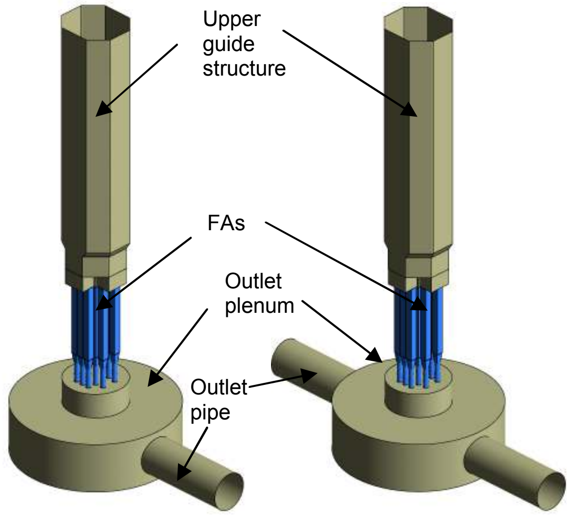
Two types of geometric models for the reactor structure are created as shown in Fig. 1, which were used to produce the mesh models for numerical predictions using a commercial CFD code, CFX-11.

To reduce the effort on the calculation and for simplicity of calculation model, the fuel assemblies in the reactor core are treated as simple rectangular channels which can simulate the pressure drop with change of flow velocity in the rectangular channel. The rectangular channel is treated as a porous region to produce the velocity increment effect by decrease of flow area due to fuel plates. A volumetric heat source is implemented to this region to give an effect of heat generation from the fuel plates. This modeling method was effective to create a CFD model for a large integrated system such as a nuclear reactor [1, 2].

2.2Boundary condition set-up

A research reactor is immersed in the 9 m water pool, and its upper guide structure is open to pool, so the CFD model needs to cover the pool as well as the reactor structure. The pool can be removed from the CFD model, however, by applying a hydrostatic pressure boundary at the top of the upper guide structure.

Flow rate in range of 61.4 kg/s to 429.9 kg/s is



(a) single outlet (b) double outlet

Fig. 1 Geometric models for CFD analysis

applied to the outlet boundary condition. For two outlet plenum model, a half flow rate is given to each outlet face. Under the condition of immersed in the deep water pool, buoyancy and hydrostatic head is significant factor to be considered in the CFD model. RSM (Reynolds Stress Model) is adopted for simulating turbulence, which is known for its good capability to predict the flows with strong swirl, rotating flow and vortex flow.

2.3 Calculation procedure

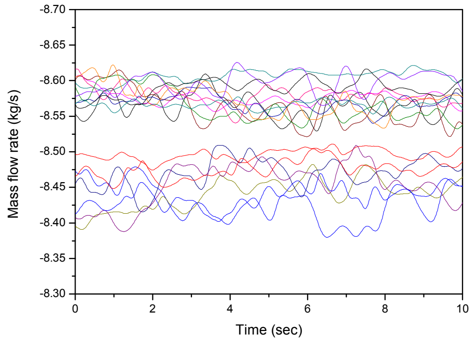
The major purpose of this study is to investigate the effect of branching outlet pipe on the core flow distribution which would be fluctuating. In order to find out this feature, CFD analysis should be carried out by transient calculation. The first calculation for steady state condition is done, and then second calculation for transient condition is conducted with the result of first calculation as an initial condition, for 10 seconds of simulation time with 0.005 seconds of time step.

3. Results and Discussions

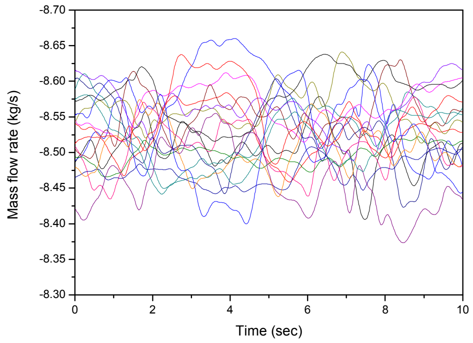
3.1Effect of branching

Mass flow rate variations of all fuel assemblies in the core are monitored throughout the transient calculation as shown in Fig. 2. It can be seen that flow rates of all fuel assemblies are fluctuating with time.

In Fig.2, several distinct differences between single outlet and double outlet can be found out. The flow

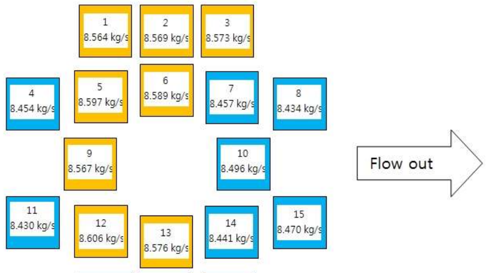


(a) single outlet plenum

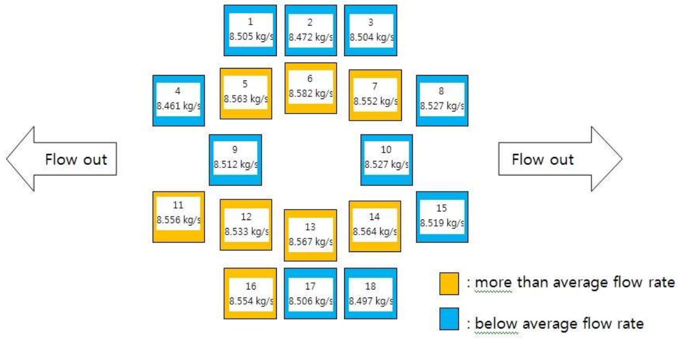


(b) double outlet plenum

Fig. 2 Fluctuation of mass flow rate on each FA in the core (153.5 kg/s)



(a) single outlet plenum



(b) double outlet plenum

Fig. 3 Flow distribution of average mass flow rate (153.5 kg/s)

rates of single outlet plenum are divided into two groups: greater or smaller than the average flow rate (8.53 kg/s). Fig. 3 shows the flow rates of each fuel assembly. Orange color means greater than average mass flow rate, and cyan color means smaller than average flow rate. For the single outlet plenum, the fuel assemblies with smaller than average flow rate are located near the outlet pipe and opposite side. Double outlet plenum shows notably different results. The grouping of flow rates by location seen in the single outlet plenum can not be seen here. This can be explained by connecting to the magnitude of fluctuation. In the single outlet plenum, flow rates can be seen separately, since they vary within a narrow band. But more violent fluctuation of double outlet plenum in wide range makes grouping of graph lines difficult. In spite of that, whole range of fluctuation for both case are nearly the same. The most violent fluctuation is located on the nearest side of the both outlets.

The flow rate deviations from the average value do not exceed ± 3% in all fuel assemblies, independent of plenum type.

3.2Effect of flow rate

Decrease or increase in flow rate directly affects the flow deviation and feature of fluctuating flow. It is the most remarkable thing that the grouping of graph lines of flow rates disappear in low flow rate. As the flow rate increases, such a grouping shows up more apparently, and fluctuation become more violent and its period shorter.

Although the flow rate increases or decreases, the flow deviation from the average value keep ± 3%. The location of greater or smaller flow rate seems to be independent of flow rate change.

4. Conclusions

According to CFD results, the branching of outlet pipe influences the flow distribution and fluctuation of flow rate in each fuel assembly. Flow fluctuation for double outlet plenum is more violent than that for single outlet plenum. Increase of flow rate make fluctuation more violent, but do not change the flow distribution noticeably. Flow deviation from the average value maintains ± 3% regardless of flow rate change.

REFERENCES

[1] Jongharк Park et al, Study on the modeling method for computational fluid dynamics to predict the coolant flow behavior in a tank-in-pool type research reactor, KAREI/TR-3858/2009, 2009.

[2] Jongharк Park et al, Study on the Coolant Behavior in a Tank-in-pool type Research Reactor using a CFD Technique, Proceedings of KPVP-2009, pp. 379-380. 2009.