1. Abstract

The HANARO research reactor has been operating since 1995. It was designed by KAERI and its performance has been verified for the last 10 years. CFD techniques have been continually applied as a useful tool to supplement the thermal-hydraulic safety analyses of HANARO. This helped us design and/or verify the hot water layer system, flow straigtener(FS), inlet plenum and so on. This paper summarizes some of the representative CFD results performed at the HANARO.

2. CFD applications at the HANARO

2.1 Analysis of flow in the HANARO reactor pool[1]

2.1.1 General

HANARO reactor pool is cylindrical and its size is 4m in diameter and 12.2m in depth. The reactor core sits at the bottom. Ten percent of the total flow is fed into the bottom of the pool and thus the same amount of pool water is being sucked through the chimney on top of the core; this is called the bypass flow. At the top of the reactor pool, a hot water layer is formed. This prevents any activated reactor pool water from reaching the top of the pool surface by a buoyant force, where workers are present. Using CFD we wanted to simulate how the bypass flow behaves and the hot water layer is affected.

As in a real operation, a HWL is formed before the bypass flow enters the reactor pool. In the calculation of the pool behavior, a HWL is formed by circulating 50°C water for 2 hours and then the bypass flow is dumped into the pool. The case of a hotter bypass flow than the pool is also studied to observe the effect of the bypass flow on the HWL.

2.1.2 Calculation method

Since the calculation domain is the reactor pool, the reactor structures are simply modeled as cylinders. The spaces between those cylinders and the pool liner are turned into calculation spaces. The bypass flow enters the domain at the bottom of the pool. The hot water from and to the HWL heater system is assumed as boundary conditions. The flow rate of the HWL and bypass (1.4kg/s and 77kg/s) were inputted by using the boundary conditions. For a convergence of the problem, 40 seconds of a fixed time step (time step 0.2s) for a calculation was implemented and then an adaptive time step calculation was resumed by using the previous fixed time step calculation results as the initial conditions.

2.1.3 Results and discussions

To simulate the formation of a HWL, hot water of 50°C was assumed to circulate at a rate of 1.4kg/s for 2 hours which resulted in a HWL of 43°C at the pool top. If the temperature of the bypass flow is large, it may increase due to a natural convection and it may affect the HWL at the pool top. The calculation for this case was performed to analyze the flow behavior. The initial conditions are the same except that the temperature of the bypass flow is larger than that of the pool by 5°C. Figure 1 shows the temperature and velocity field at t=120s and 300s after the hot and normal bypass flow enters respectively. In the case of the normal bypass flow, i.e., same bypass temperature as that of the pool (right figure), the HWL is maintained. While, in the case of the hot bypass flow (left figures), some upward motion is clearly seen at the bottom right of the HWL at the same time(120s). As a result, the bottom right of the HWL starts to collapse. At t=300s the overall pool temperature starts to change because of a mixing of the hot bypass flow and the cold pool water. This tells us that the temperature of the bypass flow should not be higher than that of the pool to guarantee a stable operation of the HWL system. And the thicker the HWL is the less the chance of a collapse of the HWL. As a design application, the upgrade of the heater power of the HWL system was conducted.
2.2 Design of a flow straightener (FS)[2]

2.2.1 General
One of a few drawbacks of our reactor was the flow-induced vibration of the fuel bundles which, we believe, was due to a large swirl in the inlet plenum caused by an asymmetric flow entering the inlet plenum. The swirl in the inlet plenum affected the flow inside the fuel channels. We had to change some of the mechanical parts of the fuel bundles several times to solve this problem temporarily. Final approach to cope with this was to change the flow pattern inside the fuel channels and thus suppress a fuel vibration. So we designed a FS which is being installed just underneath the fuel channel to make the flow smooth inside the fuel channel. Another function of the FS is to screen debris from falling down to the inlet plenum or coming up to the fuel channel. To install the FS into the reactor core, the pressure drop across the core must be equal to the design
pressure drop to guarantee the same flow-rate in the core before and after the installation.

2.2.2 Selection of FSs and experiment

Three types of FSs were studied, i.e., muffler, honeycomb, and star shape, which are shown in the following figures. The total pressure drop of each FS type is listed in Table 1. The pressure drop of the muffler-type FS was too big to adopt, even though its straining function was the best. The star type had the lowest pressure drop and so was decided to use it. To guarantee the same core flow after an installation of the FS, an experiment must follow.

![Image of flow straighteners](image1)

**Figure 2. Types of the flow straighteners**

<table>
<thead>
<tr>
<th>FS Type</th>
<th>Muffler</th>
<th>Honeycomb</th>
<th>Star</th>
<th>No FS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total Pressure Drop [kPa]</td>
<td>140 (w/ 3mm holes)</td>
<td>90</td>
<td>70</td>
<td>40</td>
</tr>
<tr>
<td></td>
<td>100 (w/ 5mm holes)</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*Table 1. Pressure drop for several flow straighteners*

In the experimental loop, a magnetic flow-meter is installed. Since the flow-meter only measures a volumetric flow, some compensation was applied to correct the density change with the coolant temperature, which results rather accurate mass-flow. The mass-flow of from 50 to 110% of the design flow-rate were investigated to measure the pressure drop.

2.2.3 Results and discussions

The final dimensions of the FSs were determined by a trial & error approach through experiments. With the final FSs, the pressure drop across the cylindrical channel was 208 kPa at a design flow of 12.7 kg/sec and the pressure drop across the hexagonal channel was 207 kPa at a design flow of 19.6 kg/sec, respectively. Both match the design requirement well, 209 kPa ± 5%.

2.3 Design of flow-skirt[3]

2.3.1 General

Based on the 10 years experience of HANARO, KAERI is now designing a new research reactor called AHRR, advanced HANARO research reactor. Among the by-products of this project, design of a new inlet plenum with a flow-skirt to make the inlet flow symmetric was studied, which is intended to get rid of the flow induced vibration as in the current HANARO design. The results of a wide flow-skirt and two different narrow flow-skirts are presented. The effect of the inlet pipe location with respect to the flow-skirt was also studied.

2.3.2 Calculations
First, a wide flow-skirt was studied (Fig. 3). Many holes at the bottom part of the skirt were modeled precisely. The purpose of this study was to make sure that a large swirl inside the inlet plenum disappears and the flow distribution in the fuel channels becomes uniform. The result indicates that some swirl still exist inside the inlet plenum. And the maximum channel flow difference is 3% compared to 12% without the flow-skirt.

![Figure 3. Wide flow-skirt](image)

To improve this wide flow-skirt, two different narrow flow-skirts are investigated. Fig. 4 shows the perspective views of the inlet plenum with these narrow flow-skirts. The inlet plenum type-I injects the cooling water directly to the perforated region of the flow-skirt. In type-II, the inlet location is moved to an upper part where the skirt has a smooth surface. Instead of treating the flow-skirt as a simple porous media, every hole in the flow-skirt has been modeled with an appropriate mesh in the calculation to achieve a comprehensive result. A total mesh number is about 1.6 million. RNG k-ε turbulent model known to be suitable for a swirl flow is employed.

![Fig. 4 Perspective views of the narrow-skirt inlet plenum](image)

### 2.3.3 Results and discussions

Since the injected cooling water has a high flow velocity, the dynamic pressure distribution at the perforated region for type-I must be considerably more uneven than type-II, which is illustrated in Fig. 5. This may cause an uneven velocity distribution near the entrance region of the flow in type-I compared to the type-II.

![Fig. 5 Dynamic pressure contours on the flow-skirt](image)

Fig. 6 shows the velocity contours at the outlet of each flow tube. The velocity contours of type-I are

![Fig. 6 Velocity contours at the outlet](image)
distorted for all the flow tubes, which means that there may exist small strong vortices affecting pressure fluctuation. But type-II shows centralizing and relatively even distribution of flows.

![Image of velocity contours at the outlet of each flow tube](image)

**Fig. 6 Velocity contours at the outlet of each flow tube**

According to the CFD analysis results, the inlet should be located far from the perforated region of the flow-skirt to avoid a direct bumping into the holes of the flow-skirt. Narrow type flow-skirt can make the flow-rate of each tube more even than the wide type one.

### 2.4 Design of a tube-type fuel[4]

#### 2.4.1 General

Tubular fuel was considered as one candidate for AHRR. It has several concentric fuel plates separated by small gaps for coolant passages. Since these coolant channels are independent, the flow must be distributed well so as not to cause any thermal-hydraulic problems. We intended to design the gaps with equal flow velocities. With the help of a CFD tool, it was possible to determine the geometries to produce even flows in the coolant channels.

#### 2.4.2 Calculations

The calculation was done for two geometry models, basic and detailed. The basic model is the conceptual design of a tubular fuel with a rather simpler geometry. The detailed model is for a suggested geometry after analyzing the result of the basic one. For both models, the standard k-e model was applied for the turbulence and a wall function was assumed for the near-wall flows. Sufficient numbers of nodes in the narrow gaps were selected for an accurate estimation of the wall pressure drop. For the 2nd case, a complete inlet structure model was added while for the 1st one only the gap regions were accurately modeled so as to concentrate only on the flow distributions for the coolant channels.

#### 2.4.3 Results and discussions

From the calculated velocities for the 1st model, the velocities for the inner-most(1st) and the outer-most(7th) channel are much lower than the average, which is shown in the following Table 2.

<table>
<thead>
<tr>
<th>channel</th>
<th>velocity(m/s)</th>
<th>normalized velocity</th>
<th>$D_h$(mm)</th>
<th>normalized $D_h$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1st</td>
<td>8.40</td>
<td><strong>0.941</strong></td>
<td>4.377</td>
<td>0.937</td>
</tr>
<tr>
<td>2nd</td>
<td>9.25</td>
<td>1.037</td>
<td>4.557</td>
<td>0.976</td>
</tr>
<tr>
<td>3rd</td>
<td>9.33</td>
<td>1.046</td>
<td>4.656</td>
<td>0.997</td>
</tr>
<tr>
<td>4th</td>
<td>9.21</td>
<td>1.032</td>
<td>4.719</td>
<td>1.011</td>
</tr>
<tr>
<td>5th</td>
<td>8.95</td>
<td>1.003</td>
<td>4.763</td>
<td>1.020</td>
</tr>
<tr>
<td>6th</td>
<td>8.45</td>
<td>0.947</td>
<td>4.795</td>
<td>1.027</td>
</tr>
<tr>
<td>7th</td>
<td>7.61</td>
<td><strong>0.853</strong></td>
<td>4.819</td>
<td>1.032</td>
</tr>
<tr>
<td>Average</td>
<td>8.923</td>
<td>1</td>
<td>4.669</td>
<td>1</td>
</tr>
</tbody>
</table>

**Table 2. Average velocities and hydraulic diameters for each channel for basic model**

This is because the incoming flow itself is small for those channels due to a wall friction effect, i.e., walls of
the center rod and the diffuser. This can be seen from the velocity contour of the inlet region shown in the Fig. 7.

![Velocity contours of the inlet region of the tubular fuel assembly](image)

Fig. 7 Velocity contours of the inlet region of the tubular fuel assembly

The flow separation occurring near the diffuser section makes the outer-most channel flow the smallest. In order not to be affected by the inlet structures, in the detailed model, the fuel plates were moved to 60mm in down-flow direction with an addition of complete inlet structures. The flow distribution became much flatter than in the basic model and the comparison is shown in the next figure.

![Improvement of an average velocity distribution of the flow channels](image)

Fig 8. Improvement of an average velocity distribution of the flow channels

### 3. Conclusion

Some representative applications of the CFD tool to the HANARO research reactor are presented. It has been used for analyzing the flow characteristics of the reactor pool with a hot water layer, design of flow straightening devices, i.e., flow straightener and flow-skirt, design of a new tubular fuel geometry, etc. The importance and usefulness of the CFD technique in the estimation of a thermal-hydraulic behavior are becoming greater with the rapid advances in hardware and software development. The CFD tools are already inevitable one for design and analysis in this field. The application of CFD to a research reactor design is also very useful. Since the 1st introduction of this tool to the HANARO team its use has become very popular.

### 4. References