CFD Analysis for Advanced Integrated Head Assembly

Won Ho Jo [a*], Tae Kyo Kang [a], Yeon Ho Cho [a], Hyun Min Kim [a]
[a]*KEPCO E&C, 111 Daedeok-daero 989 beon-gil, Yuseong-gu, Daejeon, 34057, Korea*
[*]*Corresponding author: dderts@kepco-enc.com*

1. Introduction

The Integrated Head Assembly (IHA) is permanently installed on the reactor vessel closure head during the normal plant operation and refueling operation. It consists of a number of systems and components such as the head lifting system, seismic support system, Control Element Drive Mechanism (CEDM) cooling system, cable support system, cooling shroud assemblies.

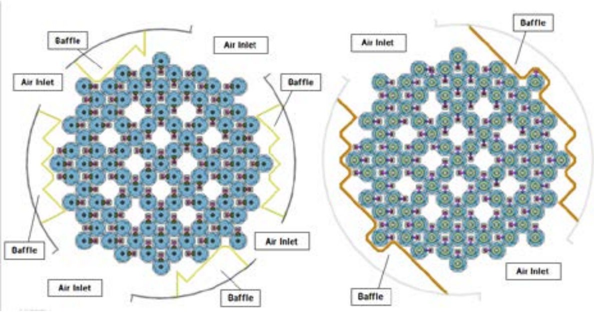
With the operating experiences of the IHA, the needs for the design change to the current APR1400 IHA arouse to improve the seismic resistance and to accommodate the convenient maintenance. So, the Advanced IHA (AIHA) design has been developed. We have started from the design concepts of the APR1400 IHA, and decided to change the cooling air passage to provide more space inside the IHA for the cable routing and to simplify the design by reducing the baffle regions and inlet air openings.

In this paper, the effects of the design changes were rigorously studied for the various sizes of the inlet openings to assure the proper cooling of the CEDMs. And the system pressure differentials and required flow rate for the CEDM cooling fan were analyzed regarding the various operating conditions for determining the capacity of the fan.

2. Methods and Results

2.1 Cooling air passage and design requirements

The cooling air passage inside the AIHA was changed from the APR1400 IHA design. The air inlet openings and baffles induce the CEDM cooling air from containment atmosphere and provide the path for the heated cooling air. Comparing to the APR1400 IHA, the number of baffles and air inlet openings is reduced to two (2) regions from four (4) regions as shown in Figure 1. The baffle regions were expanded to the maximum possible spaces while avoiding the interference with other components, especially the CEDMs, inside the AIHA to compensate for the reduced number of the baffle regions. Other than the baffle regions, the CEDM cooling air passage around CEDM coil stacks is identical to the APR1400 IHA. The CEDM cooling air flows through the air openings located on the middle cooling shroud assembly, and then flows down through the CEDM cooling shrouds and CEDM plenum plate. Then it flows crossing the RV head regions, and then it goes upward through the baffle to the upper air plenum assembly. Finally, the cooling air exits through the cooling fan to the containment. Figure 2 shows the cooling air passage from the air inlets to the CEDM cooling fans.



(a) APR1400 IHA (b) AIHA

Fig. 1. Baffle and Air Inlet Opening Regions

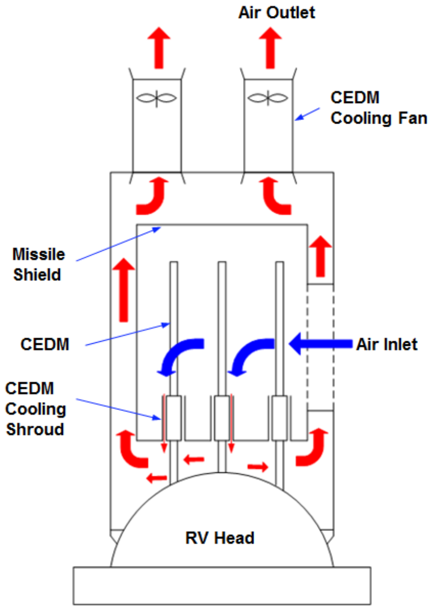


Fig. 2. IHA Cooling Air Passage

There are three (3) CEDM cooling fans on the top of the IHA. Each fan is capable of providing 50% of the total cooling air flow. During the normal operation, 2 out of 3 cooling fans are operated, and the rest one is in the standby condition. The CEDM cooling fan shall be designed to provide sufficient cooling air to keep the CEDM coils below the design temperature of 350 °F. Each CEDM is designed to require 800 Standard Cubic Feet per Minute (SCFM) ± 10% for the proper cooling. The inlet air temperature of the CEDM cooling air is controlled to be ranged from 60 °F to 120 °F same as the containment air condition.

2.2 Finite Volume Model

The fluid regions were modeled first using the 3D CAD program, Creo 2.0 [2], by subtracting the AIHA volume from the confined surrounding volume. Then the fluid region inside the upper cooling shroud assembly was removed because the air flow in this region is negligible. The final fluid region is shown in Figure 3.

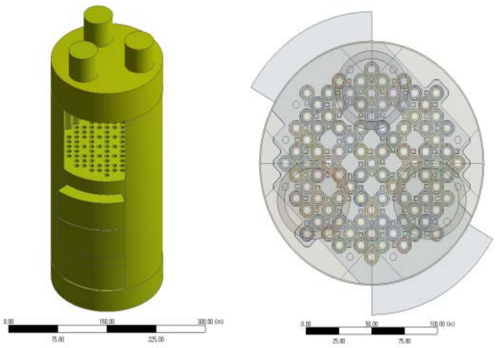


Fig. 3. AIHA Fluid Region Model

Then the modeled fluid region and solid models of CEDMs and RV head nozzles from the 3D CAD program were exported to ANSYS Workbench [3] for the surface mesh and grouping boundaries. The CEDM cooling shrouds and the inlet bells of the cooling fans are made of thin plate with the thickness less than 0.5 inch. So, they were modeled as surfaces. Then the Workbench model is transferred to ANSYS Fluent. [4]

2.3 Turbulence Model and Boundary Conditions

The most popular turbulence models for the commercial purpose are the K-ε and K-ω family models. In this research, the Realizable K-ε model [5] with the scalable wall function is applied to the cooling air analysis. The standard K-ε model is a well-established model capable of resolving through the boundary layer. [6] However, the realizable model exhibits superior performance compared to the standard model for the flow involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation.

It is generally known that the Realizable K-ε model is more economical than the K-ω SST model [7] since the SST model requires a quality initial condition. The quality initial conditions can be established by other converged solutions from an analysis. A case study was performed to compare the results from the Realizable K-ε model with those from the K-ω SST model for the added inlet case. The cooling air distribution to the CEDMs from the Realizable K-ε model is ranged from 718 SCFM to 882 SCFM. The same result from the K-ω SST model is ranged from 732 to 876 SCFM. So, the Realizable K-ε model is selected for this study because one of the main purposes for this study is to decide the proper size of the inlet openings to assure the cooling performance of the cooling fans in a conservative way.

Thermal and mass flow rate conditions are applied to the boundaries of outlets and solid parts. The temperature of CEDMs is given as 400 °F which is the maximum initial thermal condition at the start of CEDM motion. And the temperature of the RV head nozzles is applied as 507.5 °F which is the mean temperature of the reactor coolant inside the head nozzle. The mass flow rate boundary condition at the fan outlet was decided considering the number of CEDMs and flow distribution requirement of 800 SCFM for each CEDM. The initial condition was 48.3 lbm/s per one fan, and it was changed to 48 lbm/s after reviewing the analysis results. The boundaries of the air inlets and fan outlets are described in Figure 4 including the arbitrarily decided fan numbers for the analysis.

Fig. 4. Inlet Openings and Fan Outlets

2.4 Results

The CFD analyses were performed to determine the proper size and location of the air inlet openings in order to assure the proper cooling air distribution for each CEDM. Then the case analysis has been made for the different fan operating conditions. For all the analyzed cases, the fan numbers shown on Figure 4 are referred.

All the analyses were performed at the steady state condition. For the initial design of the AIHA, air inlets were sized with 45 degrees width at two opposite directions in middle cooling shroud shell. The analysis results did not meet the flow distribution requirement of 800 SCFM ±10%. Several cases were attempted to decide the sizes of the inlet air openings for the AIHA. The air inlet locations were lowered by 5 inches in the first test case, and the additional small air inlet were added to the initial model below the existing air inlets for another test case. However, those cases were failed to meet the flow distribution requirement for each CEDM. Finally, the size of air inlets was increased as much as 60 degrees width and the fan out flow was reduced from 48.3 lbm/s to 48 lbm/s to meet the requirement. The analysis results for the test cases are summarized in Table I. Figure 5 shows the typical air flow from the air inlet to the CEDM cooling shroud inlet.

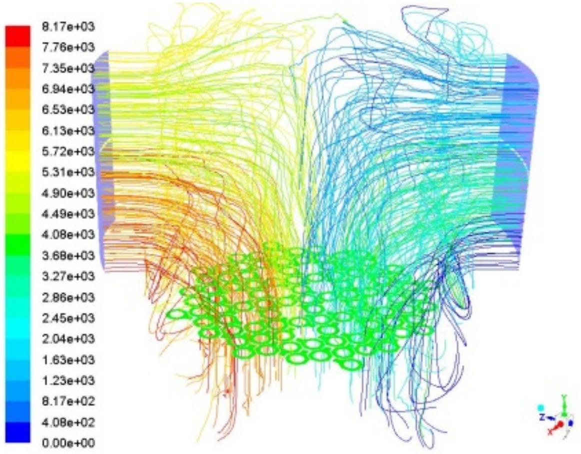


Fig. 5. Air Flow from Inlet to CEDM Cooling Shroud

Table I: Cooling Air Distribution

Inlet Case	Out flow per fan (lbm/s)	Min. (SCFM)	Max. (SCFM)
Initial inlets	48.3	715.9	894.2
Lowered (5”)	48.3	716	895
Inlets added	48.3	718	882
Final inlets #1	48.3	734	882
Final inlets #2	48.0	724	878

Note: 60 °F (Air), Fan 1&2 Operated.

After establishing the proper air inlet size and mass flow rate from the test cases, subsequent analyses were performed to estimate the cooling air temperature around CEDM coil stacks, system pressure differential, and the cooling air conditions in AIHA. With the final model, the CFD analyses were performed for the following conditions:

Condition 1: 60 °F (Inlet air), Fan 1 &2 in operation.
Condition 2: 60 °F (Inlet air), Fan 1 &3 in operation.
Condition 3: 120 °F (Inlet air), Fan 1 &2 in operation.
Condition 4: 120 °F (Inlet air), Fan 1 &3 in operation.

As a design requirement, the CEDM coils stacks shall be operated below 350 °F. The contour of the air temperature around CEDM coil stacks are shown in Figure 6 which indicates the air temperature at the outmost surface of the CEDMs in Condition 3. According to this result, the outmost surface temperature is below 350 °F except for a few small local regions. It proves that CEDMs are in a safe operating condition within the range of the coil stack operating temperature.

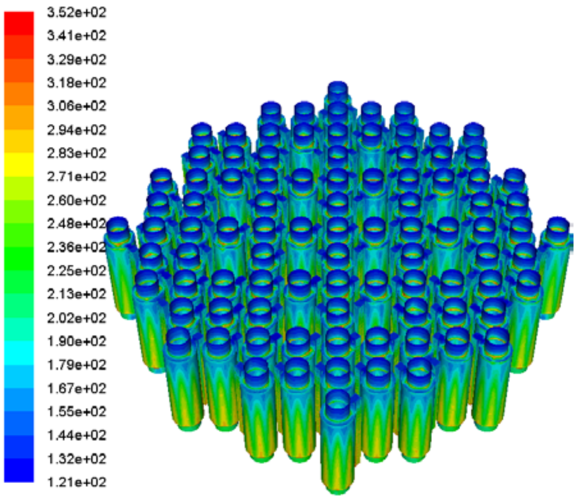


Fig. 6. Outmost CEDM Surface Temperature (°F) in Condition 3

The system pressure differential and required mass flow rate per fan are the input parameters for determining the capacity of the fan. The required mass flow rate per fan is analyzed as 36,954 SCFM. The system pressure differentials and average static air temperature at the operating fan outlet are described for all the conditions in Table II.

Table II: System Pressure Differential & Average Air temperature at Operating Fan Outlet (static)

Condition	System Pressure Differential (in-H_2O)	Average Air temperature (°F)
1	6.8	130.3
2	6.7	130.3
3	7.3	180
4	7.2	180

The static temperature and static pressure results at the cross section passing the AIHA center of the fan 1 in Condition 4 are plotted in Figure 7.

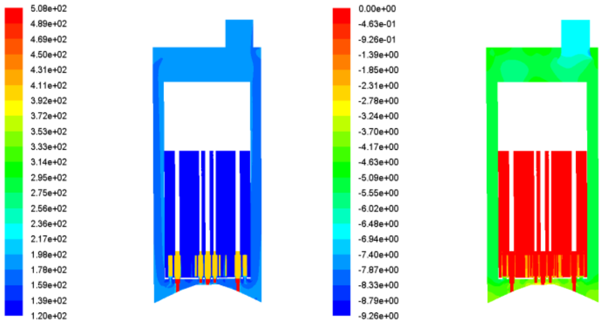


Fig. 7. Static Temperature (°F) &Static Pressure (in-H_2O) of AIHA in Condition 4 (Fan 1)

3. Conclusions

As a part of the design process of the AIHA, the number of air inlets and baffle regions are reduced by simplifying the design of the APR1400 IHA. The design change of the baffle regions has been made such that the maximum possible space are occupied inside the IHA cooling shroud shell while avoiding the interference with CEDMs. So, only the air inlet opening was studied for the design change to supply a sufficient cooling air flow for each CEDM. The size and location of the air inlets in middle cooling shroud assembly were determined by the CFD analyses of the AIHA. And the case CFD analyses were performed depending on the ambient air temperature and fan operating conditions.

The size of the air inlet openings is increased by comparison with the initial AIHA design, and it is confirmed that the cooling air flow rate for each CEDM meet the design requirement of 800 SCFM ± 10% with the increased air inlets. At the initial analysis, the fan outlet flow rate was assumed as 48.3 lbm/s, but the result revealed that the less outflow rate at the fan is enough to meet the design requirement. In the subsequent analysis results show that the temperature of the CEDM coil stack is maintained below 350 °F.

It is recognized that the cooling air passage design for the AIHA is better in that the system pressure differential of the AIHA is decreased as much as about 25% compared to that of the APR1400 IHA design. However, the required flow rate per fan is almost the same or a little less than the APR1400 IHA. Therefore it is expected that the size of the CEDM cooling fan can be decreased in the AIHA design comparatively. The decreased weight of a fan might be helpful to reduce the IHA design loads under the normal operating and accidental conditions.

The information from the CFD analysis will be used as the baseline in the detailed design and structural analysis of the AIHA for the future application. Iterative design and analysis process will be performed to complete the AIHA design.

REFERENCES

[1] T.K. Kang, W.H. Jo, Y.H. Cho, S.G. Chang, and D.H. Lee, The Application of an Integrated Head Assembly for Advanced Power Reactor 1400, ICONE24, 2016.
[2] C. F. Sikora, Creo Parametric 2.0 Advanced, 2013.
[3] ANSYS Workbench User’s Guide for Release 17.0, 2016.
[4] ANSYS Fluent User’s Guide for Release 17.0, 2016.
[5] T.H. Shih, W.W. Liou, A. Shabbir, Z. Yang, and J. Zhu, A New K-ε Eddy Viscosity Model for High Reynolds Number Turbulent Flows – Model Development and Validation, Computers Fluids, 24(3): 227-238, 1995.
[6] D.C. Wilcox, Turbulence Modeling for CFD, 2nd Edition, DCW Industries, 2006.
[7] F. R. Menter, Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications, AIAA Journal, Vol. 32, August 1994.