heat exchanger modelling for generator ventilation systems

This paper discusses how the heat exchanger/cooler is modelled using CFD (Computational Fluid Dynamics) as a porous medium and the associated boundary conditions.

Dr. Ram K. Ganesh (Charlotte, NC, USA)

QuEST Global

Table of Contents

1 Summary

The cooling of the generator's stators and rotors is accomplished by circulating water, hydrogen or both through stator coils/core and rotor field windings. The gas coolant is circulated by means of a blower which is an integral part of the rotor assembly. This paper discusses how the heat exchanger/cooler is modelled using CFD (computational Fluid Dynamics) as a porous medium and the associated boundary conditions. The results are validated from the benchmark pressure drops observed in similar generators.

2 Modelling Geometry

There is an annular gap between the outer diameter of the case and the inner diameter of the frame through which the coolant flows axially towards the lead end and the non-lead end (see Figure 1). The coolant however enters this annulus radially across a portion of the inner surface of the case. The location where this portion ends is the boundary between the rotor and the end windings. As the coolant flow is symmetric, only half of the whole geometry is modelled. There are two vertical coolers on each end. The duct flow through the annulus has to make up to four right angle turns before going through the heat exchanger's rectangular inlet. In addition, the flow has to go through a constricted region, a sudden contraction. The coolant flow is highly three dimensional and any amount of hand calculations using standard formulae can only provide an estimate. However, a CFD simulation with the exact geometry can provide reasonable answers close to actual measurements.
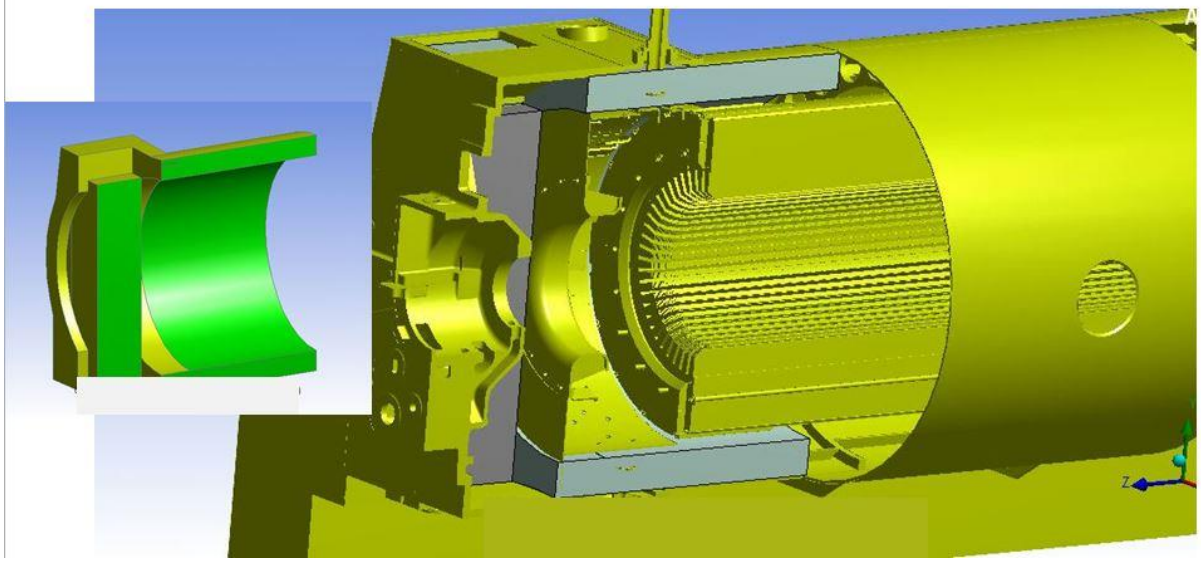
Figure 1: CFD Air Solid & Heat Exchanger

Even though, the air-solid portion of the CFD model is straight forward, the air-space between the rigid plates, the heat exchanger modeling needs special treatment using porous medium theory. The *FLUENT* engineering software has this modeling capability made available to the user.

3 FLUENT CFD Modelling

FLUENT describes the physical models used to simulate the performance of heat exchangers that emphasizes the heat transfer part. However, modelling flow through porous media is the effective way to model the heat exchanger under the circumstances where time is critical and the desired result is only the pressure drop and not the heat transferred or lost. Therefore, the air-solid model geometry would consist of the annular duct region with a partial radial inlet, the space around the cooler (heat exchanger) with the constriction (minimum area), expansion into the cooler inlet rectangular area, the porous medium rectangular

prismatic cooler and a rectangular prismatic plenum to specify the exit boundary condition of zero gage pressure (see Figure 2). The density variation is assumed to be negligible, to be consistent with the generator integrated environment ventilation analysis program. Therefore, the inlet velocity is obtained based on the volumetric flow and the area over which the velocity boundary condition is being applied.
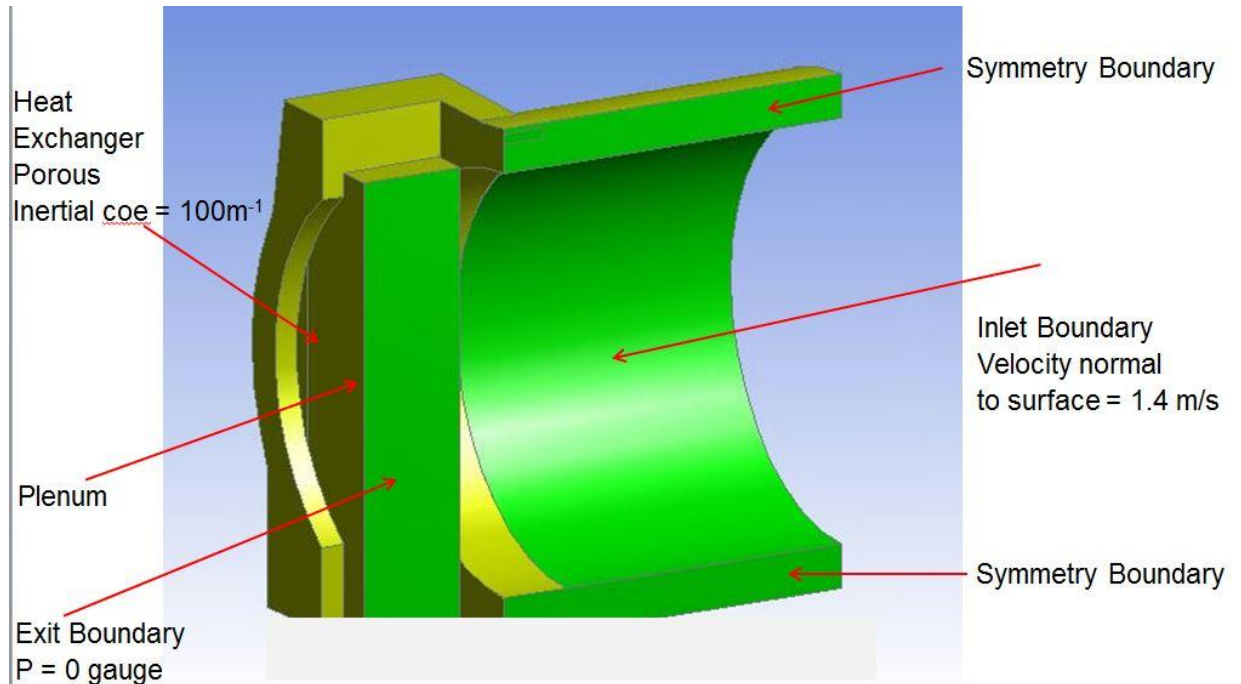

Figure 2: CFD Boundary Conditions

Knowing the volumetric flow, flow area, hydraulic diameter and kinematic viscosity, the Reynolds numbers is calculated. The flow is typically turbulent based on the high Reynolds number. To model the near wall turbulent boundary layer, viscous, realizable k-ε enhanced wall function is selected. The enhanced wall function treatment as recommended by *FLUENT* adequately resolves the boundary layer's viscous and turbulent behavior.

The cooler, the plenum and the remaining air-solid are three separate volumes in the model creation with interfaces and walls. The porous medium for the cooler has to be defined in the zone selection

4

process. The inlet boundary surface has to be picked and the inlet velocity boundary condition has to be specified. Similarly, the exit surface of the plenum is picked and the exit zero gauge pressure boundary condition is applied. The cooler's inlet and the outlet surfaces, the air-solid surface in contact with the cooler's inlet surface and the plenum surface in contact with the cooler's outlet surface have to be identified as interfaces. The remaining surfaces are treated by *FLUENT* as walls by default. The air-solid surface that is in contact with the cooler's inlet surface and the cooler's inlet surface are separately linked as matching surfaces. The cooler's outlet surface and the plenum's inlet surface are also separately linked as matching surfaces. This is called matching 'non-conformal interfaces'. This modeling technique makes the meshing in the three volumes, air-solid, cooler and the plenum independent (see Figure 3 and Figure 4). While the air-solid due to its multi-faceted nature is meshed with tetrahedral elements, the rectangular prismatic cooler and the plenum are meshed with hexahedral elements. This improves accuracy in the pressure drop calculations in the porous cooler.
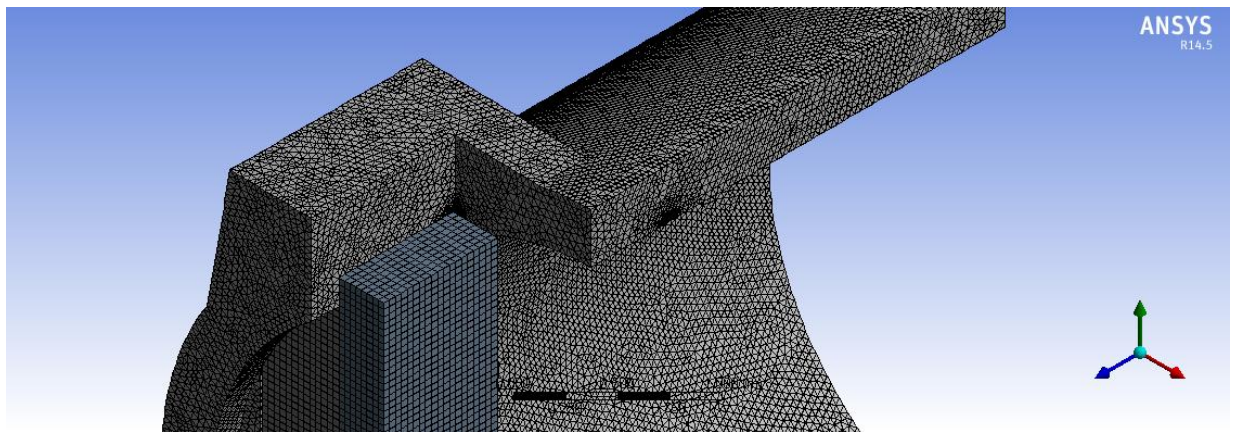

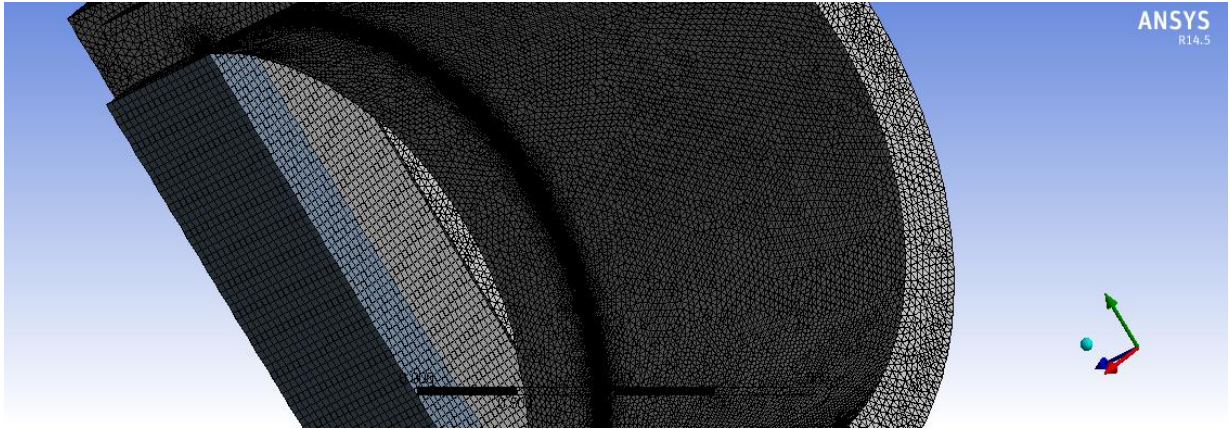**Figure 3: Mesh around the Heat Exchanger**

Figure 4: Mesh for the Heat Exchanger & Plenum

4 Porous Medium Modelling

In *Fluent*, cell zones consist of fluids and solids. Porous zones are treated as fluid zones. The pressure loss in the porous medium is determined using the inputs described in the momentum equations for porous media. The porous media model incorporates an empirically determined flow resistance in a region of the model defined as "porous". In essence, the porous media model adds a momentum sink in the governing momentum equations. The volume blockage is not physically represented in the model by default which is a limitation and the appropriate corrections should be made to obtain the desired results. Porous media are modeled by the addition of a source term to the standard fluid flow equations. The source term is composed of two parts: a viscous loss term and an inertial loss term.

$$S_i = -\left[\sum_{j=1}^{3} D_{ij}\, \mu\, v_j + \sum_{j=1}^{3} C_{ij}\, \frac{1}{2}\, \rho\, |v| v_j\right]$$

where S_i is the source term for the ith (x, y or z) momentum equation, $|v|$ is the magnitude of the velocity and D and C are prescribed matrices. The momentum sink contributes to the pressure gradient in the porous cell, creating a pressure drop that is proportional to the fluid velocity or velocity squared in the cell.

For a simple homogenous porous media, the equation above becomes

$$S_i = -\left[\frac{\mu}{\alpha}\, v_i + C_2\, \frac{1}{2}\, \rho |v|\, v_i\right]$$

where α is the permeability and C_2 is the inertial resistance factor. In the matrix form D & C, the diagonal matrices are specified with $1/\alpha$ and C_2 respectively on the diagonals and zero for other elements. The coefficient C_2 can be viewed as a loss coefficient per unit length along the flow direction, thereby allowing the pressure drop to be specified as a function of dynamic head.

For a tube bank type cooler, the permeability viscous term can be dropped and the inertial loss alone can be used yielding a simple formula

$$\Delta P = K_L\, (1/2\, \rho\, v^2)$$

That is, the pressure loss is a multiple of the dynamic pressure, where K_L is the loss factor.

Assume cross-section dimensions of the heat exchanger as 4 m high by 1 m wide.

Assume the manufactures specification for the pressure drop for the heat exchanger as 1.5 kPa at 75 m^3/s flow for 4 heat exchangers in parallel.

The flow area of the heat exchanger if only 75% of the flow area is open is   4 x 1 x 0.75 = 3 m^2.

Velocity in the cooler (for 4 coolers in parallel) is 75/4/3 = 6.25 m/s

Dynamic head in the heat exchanger = 0.5x1.225 kg/m^{3}x 6.25^2 = 23.93 Pa. The density of air is 1.225 kg/m^3.

Inertial loss factor in the heat exchanger (K_L) = ΔP / dynamic pressure = 1500/23.93 = 62.68

ANSYS FLUENT uses synthetic velocity which is the 100% flow area velocity. Therefore the inertial loss coefficient becomes (1/0.75)^2 x 62.68 = 111.43 to maintain the same ΔP.

Assume the depth of the heat exchanger is 0.5 m. Then the inertial loss factor per unit depth is 111.43 / 0.5 = 222.86 m^{-1}. This is the input to FLUENT porous medium definition in the axial direction of the heat exchanger. The inertial loss factors in the mutually perpendicular directions are made 10 times, that is 2228.6 to offer significant resistance in those directions while offering the least resistance in the axial direction for the fluid to pass through.

5 Result

The advantage of modelling the heat exchanger this way as a porous medium makes it possible to match the heat exchanger vendor's pressure drop exactly in the system that is being modelled and estimate how the pressure drop in the frame varies when the heat exchanger pressure drop is varied.

This variation study showed that the pressure drop in the rest of the system, that is the pressure drop in the frame is only 10% of the pressure drop in the heat exchanger. This is a valuable result which was not at all apparent. The results are validated from the benchmark pressure drops observed in similar generators.

6 Conclusion

The method of modelling the heat exchanger as a porous medium in a CFD analysis of the generator frame with the heat exchanger included is shown. The CFD analysis such as boundary layer resolution, etc. are not discussed here as the main focus of this paper is to show how one can model the heat exchanger effectively in a short period of time for optimum results.

7 Reference

a) ANSYS FLUENT Theory Guide, ANSYS, Inc., Release 14.5, October 2013
b) ANSYS FLUENT User's Guide, ANSYS, Inc., Release 14.5, October 2012

8 Acknowledgement

Author's Profile

Dr. Ram K. Ganesh obtained his baccalaureate, B.Tech and masters M.Tech in Mechanical Engineering from Indian Institute of Technology, Madras and Ph.D. from the University of Connecticut, USA in Applied Mechanics, specializing in thermal-fluid sciences. He has been a licensed Professional Engineer, PE in the state of South Carolina, USA for over 20 years. He was an adjunct faculty at the University of Connecticut. He worked as a Lead Engineer, a Senior Engineer and Principal Engineer in companies such as United Technologies, Pratt & Whitney Aircraft, General Electric and Westinghouse. He has authored nine technical papers in various leading international and national journals and is currently a member of ASME. He joined QuEST Global, Charlotte, NC, USA in October, 2014 as a Principal Engineer and currently working in partnership with Siemens AG, Charlotte, NC, USA.

About QuEST Global

QuEST Global is a focused global engineering solutions provider with a proven track record of over 17 years serving the product development & production engieering needs of high technology companies. A pioneer in global engineering services, QuEST is a trusted, strategic and long term partner for many Fortune 500 companies in the Aero Engines, Aerospace & Defence, Transportation, Oil & Gas, Power, Healthcare and other high tech industries. The company offers mechanical, electrical, electronics, embedded, engineering software, engineering analytics, manufacturing engineering and supply chain transformative solutions across the complete engineering lifecycle.

QuEST partners with customers to continuously create value through customer-centric culture, continuous improvement mind-set, as well as domain specific engineering capability. Through its local-global model, QuEST provides maximum value engineering interactions locally, along with high quality deliveries at optimal cost from global locations. The company comprises of more than 7,000 passionate engineers of nine different nationalities intent on making a positive impact to the business of world class customers, transforming the way they do engineering.

QuEST
BORN TO ENGINEER
www.quest-global.com