CHAPTER 5

COMPUTATIONAL FLUID DYNAMIC ANALYSIS

5.1 INTRODUCTION

This chapter documents the air velocity distribution in a flux reversal generator. The knowledge about the air velocity distribution inside electric machines is useful in undertaking various analyses. For instance, thermal analysis is fine tuned and a more realistic thermal simulation is made possible, when the air velocity distribution assisted thermal analysis is carried out. Quantities such as windage losses and life span of insulators are effectively calculated when attempted via air velocity inside the machines. The design of cooling system also becomes more perfect, if the nature of the circulating air in a machine is pictured. A 3-dimensional computational fluid dynamic analysis is used for this purpose. The profile of the turbulently circulating air inside the machine is captured. This can be used for a thorough thermal analysis.

5.2 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational flow analysis is a scientific process applied to resolve the fluid flow related problems like flow velocity, heat and mass transfer by computer based simulation methods [59-61]. For electrical engineering, although the fluid flow analysis is almost nil, the knowledge of air flow inside the electrical machine will help in a great way for understanding the heat distribution in the machine, insulation coordination, and other thermal-insulation dependent issues. Air is treated as fluid and the flow analysis is performed. When the rotor rotates, the air flow inside the electrical machine is largely tempestuous. This turbulent air can be captured and effectively used for thermal analysis. It is difficult to solve the air velocity by analytical method and hence numerical approach is used for this purpose. This is done to get the air velocity at all air region inside the machine to
predict the thermal characteristics, further more accurately. The axial and radial cross section of FRG is shown in Figure.5.1

Figure.5.1.Cross section of FRG (a) axial cross section (b) radial cross section
At first, the geometry of the air region inside the machine is completely modelled in 3D. The outer diameter of the air region is 75 mm. The diameter of the rotor is 49.40 mm. The stack length is 40 mm. The rotor-stator gap is 0.5 mm. The quadrature model of the FRG showing the machine geometries and the various air portions to be modelled for flow analysis is shown in Figure.5.2 (a). The 3D model of the inter-polar air region alone is shown in Figure.5.2 (b). The material properties required to carry out CFD simulation are set as follows: For 300° Kelvin ambient air temperature, the (fluid) air density \( \rho \), is 1.086 kg/m\(^3\) and kinematic viscosity \( \nu \), is 1.568x10\(^{-5}\) m\(^2\)/sec.

5.2.1 **Boundary conditions and governing equation**

The boundary condition is to define the behaviour of air flow on the surfaces. It is applied in Ansys as inlet and outlet of the domain. In the air model of Figure.5.2 (b), there are two cross-sectional surfaces, one, as shown in Figure.5.2(b), and the other on the back side. Any one side of these cross-sectional surfaces is taken as an inlet. The other cross-sectional surface will then form as the outlet surface. The outlet is set as 0 pascal average static pressure. Total fluid pressure \( P_{\text{total}} \) of 1 atmosphere is applied at an inlet and this will require inputting \( \rho \) and \( V \). This will calculate the static pressure from the given equation,

\[
P_{\text{total}} = P_{\text{static}} + \frac{1}{2} \rho V^2
\]

(5.1)

The machine air region wall is set to zero velocity. This is called a no-slip boundary condition. Then hexahedral mesh is applied in fluid region with maximum mesh size of 3 mm to save the computational time. Steady state model is chosen for CFD simulation. The air volume around the rotor is configured to rotate at 9000 rpm. Solver runs K-Epsilon model taking into account the air velocity and mesh size near walls. The Shear Stress Transport (SST) turbulence model was chosen for modeling turbulent fluid flow to provide closure of the Reynolds-Averaged Navier-Stokes (RANS) equations in the present study.
Figure 5.2. (a). Quadrature model of FRG, (b). 3D model of the inter-polar air region
5.2.2 Governing equation

Determination of air velocity involves the pattern of the fluid flow [62]. A 3-dimensional flow pattern is shown in Figure.5.3. The streamline passing through the point P (x, y, z) is tangential to the velocity V(x, y, z).

\[
\frac{d_x}{V_x} = \frac{d_y}{V_y} = \frac{d_z}{V_z} \quad (5.2)
\]

\[
V_x d_y = V_y d_z = V_z d_x = 0 \quad (5.3)
\]

where \( V_x, V_y \) and \( V_z \) are the velocity component in three dimensions.

Let the streamline function be \( \phi \). So when \( \phi \) is differentiated with respect to \( x, y \) & \( z \), we get

\[
\frac{d\phi}{dx} = V_x, \quad \frac{d\phi}{dy} = -V_y, \quad \frac{d\phi}{dz} = -V_z
\]

For the irrotational flow, there exists a relation

\[
\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} - \frac{\partial w}{\partial z} = 0 \quad (5.4)
\]

where \( v, u, w \) are the velocity vector in \( x, y, \) and \( z \) direction.
\[
\frac{\partial}{\partial x} \left( \frac{\partial \bar{\phi}}{\partial x} \right) + \frac{\partial}{\partial y} \left( \frac{\partial \bar{\phi}}{\partial y} \right) + \frac{\partial}{\partial z} \left( \frac{\partial \bar{\phi}}{\partial z} \right) = 0
\] 
(5.5)

\[
\left( \frac{\partial^2 \bar{\phi}}{\partial x^2} \right) + \left( \frac{\partial^2 \bar{\phi}}{\partial y^2} \right) + \left( \frac{\partial^2 \bar{\phi}}{\partial z^2} \right) = 0
\] 
(5.6)

The equation (5.6) is the three dimensional equation solved by numerically for evaluating the net velocity vector distribution \( \bar{\phi} \). With the fluid density of the whirling fluid being \( \rho \), the governing equation becomes,

\[
\rho \left\{ \frac{\partial^2 \bar{\phi}}{\partial x^2} + \frac{\partial^2 \bar{\phi}}{\partial y^2} + \frac{\partial^2 \bar{\phi}}{\partial z^2} \right\} = 0
\] 
(5.7)

### 5.2.3 Determination of air velocity and pressure

The 3-dimensional CFD analysis is performed in evaluating the air velocity and pressure in the whole air region of FRG.

The air flow inside the air region is shown in the Figure.5.4, with the punched small holes on the enclosure of the generator. This is usually done in electric machines to facilitate better cooling. This increased rate of airflow normalizes the heat distribution inside the generator. The CFD results are shown in Figure.5.5.

From the simulation results it is observed that, at 9000 rpm the surface air velocity is varied from 3.39 m/sec to 6.78 m/sec and the velocity in the mid portion of air region is varied from 1.48 m/sec to 3.39 m/sec, which is highlighted in green and blue colours in velocity streamline plot of Figure.5.5 (b).

Even though the inside surface velocity varies between 3.39 m/sec to 6.78 m/sec, this variation is so rapid that it is almost about 6.78 m/sec only. So for the calculation of heat transfer coefficient the air velocity in the surface is fixed at 6.78 m/sec. The pressure inside the air region is 4.629 Pa which is shown in Figure.5.5 (a).
Generally the required limit of the air velocity for the electrical machine is $2 - 7 \text{ m/s}$. From the above results, it is inferred that in the surface the air velocity lies within this limit. The air velocity depends on the pressure drop. Therefore, it is necessary to estimate the pressure drop in the generator. In electrical machines the primary heat generation starts from the stator, and the highest temperature of the stator occurred at the central position of the machine. The pressure is inversely proportional to the velocity. From the above plot, it is found that pressure at the centre position of the generator is higher, but the velocity at the same position exhibits less value. In the surface area the flow rate is higher but the pressure is less. Hence the obtained air velocity and pressure infer that the heat distribution inside the generator is nominal.

For the simulation of thermal analysis, the accurate prediction of heat transfer coefficient is always good, and this is achieved only through the knowledge of air flow distribution.

Figure 5.4. Air flow direction when punching hole on the enclosure of the generator.
Figure 5.5. CFD results indicating the air velocity and its pressure at 9000 rpm 
(a) Pressure contour (b) Velocity streamlines
Table 5.1. CFD Results

<table>
<thead>
<tr>
<th>S.No</th>
<th>Component</th>
<th>Velocity m/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Stator surface (green color)</td>
<td>6.782</td>
</tr>
<tr>
<td>2</td>
<td>Air gap (blue color)</td>
<td>3.399</td>
</tr>
</tbody>
</table>

5.2.4 Summary of the commands used for flow analysis

5.2.4.1 Pre-processing

Step1: Open the ANSYS CFX software, in the main window, go to the file menu and create a new file, then save with secondary name FRG.

Step2: In the main window toolbar- Analysis systems click ‘A Fluid flow’ and then click geometry. A fluid flow window appears.

Step3: The geometry of 3D air region is created in CAD separately and it is saved. Now go to the file menu of ANSYS, click ‘Import external geometry file’, select the ‘add frozen’ in the detailed view window and then click generate. The 3D model will be imported in the graphics window as shown in the Figure 5.6.

Figure 5.6. Model creation for fluid flow analysis
Step 4: Now to create the bodies for setting the material property, follow this cue: Create menu – Boolean – target bodies – click. The clicking will enable to create the bodies one-by-one.

Step 5: Then go to the main window and apply ‘mesh’. The mesh file name appears in the side outline window. Select mesh file and then select inlet, outlet and wall and rotor of the model using ‘box select tool’. Then, to name them, right click ‘create name selection’ and then enter the respective names.

5.2.4.2 Processing

Step 6: Go to the main window click setup, Ansys Launcher window gets appeared then click ok. The meshed model will appear.

Step 7: For problem setup, follow the below cue: ‘general option’ – ‘steady state solver option’ – ‘models page’ – ‘viscous laminar’ will appear where click ‘the viscous model’ – ‘K-Epsilon realizable model’ – ‘non-equilibrium wall function’ and finally click ‘ok’ to complete the set-up.

Step 8: To set the material property for flow analysis, Go to the material option select air and enter the values of density and viscosity of air. Then go to the ‘cell zone condition’ and there enter the zone name as Fluid.

Step 9: Then go to the boundary page select the inlet region and enter the velocity magnitude values, turbulence intensity and viscosity and then click ok. Same procedure is followed to create the outlet boundary.

Step 10: Select the dynamic mesh to enter the reference values of inlet.

Step 11: Then select the solution method to set coupled scheme, least square gradient, standard pressure, second order turbulence, momentum and turbulence dissipation rate.
Step 12: Then click the solution control option and enter the courant number, momentum and pressure values.

Step 13: Go to the monitor and check the print console and plot options are enabled in the drag, lift and moment off menus.

Step 14: In solution initialisation, select inlet and click initialize. Then go to calculation, enter the no of iterations and click to calculate. The simulator will calculate the solutions.

5.2.4.3 Post-processing

Step15: In results menu, graphics and animation option will show the final results in vector plots.

5.3 SUMMARY

This chapter presented a 3-D finite element analysis based computational fluid dynamics for mechanical characterizations of FRG. Model and simulation procedures to trace (i) air velocity profile and (ii) air pressure profile of a flux reversal machine have been detailed. The results of simulation are presented and discussed. This CFD result can be used for through thermal analysis for the next section.