Steam Flow Simulation in Low Pressure Turbine Exhaust Hood System

Dr C. Bhasker*
Senior Manager
Corporate R&D Division
BHEL, Vikasnagar
Hyderabad-500093, India
email: bskrtml@gmail.com
www.bskr2k.webs.com


Abstract

Three dimensional compressible flows through a low pressure steam turbine exhaust hood is modeled and simulated. Computational model for exhaust hood comprise diffusers, upper casing, and exit are created using Altair HyperMesh, a pre-processor of Altair HyperWorks Suite. This model has been further subjected to domain discretization for generation of multi-block structured grid for computational domain. The grid has been exported to Fluent flow solver for analysis and defined fluid flow properties and assigned the boundary conditions at inlet/exit locations. It is evident from the flow field, that exhaust casing is characterized by pronounced vortices, which are accompanied by energy losses. In the upper part of diffuser, analysis predicts strong vertex formation and significant pressure loss. In the lower part of the diffuser, the simulation is not indicating presence of eddies or separation region. The efficiency of the exhaust hood is affected by the presence of these vortices and gives rise to losses. Increased velocities of the fluid in exhaust hood chamber are undesirable, which can cause increase in the pressure difference, requires detailed studies with modifications of diffuser profile.

1. Introduction

The transport of steam through control and stop valves via turbine cylinders enter into the exhaust system as shown in the Fig-1 and leaves to condenser. During the operation, transport devices experienced several types of losses which in turn lead to performance degradations. Most of the losses take place in the steam passages of the turbine stage and typical losses are profile loss, secondary loss and leakage loss. In the low pressure section fluctuation/exhaust losses occurs due to non-uniformity in the circumferential direction of the inlet exhaust system. Present demands for increasing thermal efficiency of large steam turbines used for power generation are receiving greater attention [1] to enhance the performance improvements of exhaust end of low pressure element. The exhaust hood acts as an expanding diffuser with the high velocity inside the casing being reduced gradually with the corresponding increase in static pressure. Steam passes over the top of the bearing cone has a greater tendency to impact on the hood end wall rather than flowing towards condenser neck. This creates a pressure wave, which causes losses due to flow separation, local recirculation, obstructions and the large vortex flow and effects pressure recovery parameter. The influence of this parameter will lead to drop in stage output and the combinations of these effects are considered to be the fundamental driving force for assessing the aerodynamic performance.

Retired.
Present mailing address:
402 Residency Apartment, Ashoknagar Bridge, Musheerabad (Narayanguda) Hyderabad-500020
mobile: 9849252948

From the past several years, development of advance procedures for laboratory testing and analysis [2-3] are used to identify, evaluate and select practical upgrade strategies. The research investigation [4] describes an approach to improve losses for existing turbine, which operates over
substantial variations in exhaust pressure conditions during the normal cycle. Some of the improvement can be accomplished [5] with the modification of its internals for performance enhancement. Few of them will be incremental change in diffuser shape, installation flow guides, eliminating flow restrictions/flow obstructions or simply streamlining sharp angles. Computational Fluid Dynamics - CFD software tools [6] are now under extensive use for obtaining the flow characteristics to minimize losses in exhaust hood system. CFD analysis also helps to improve aerodynamic performance by reducing turbulent eddies, since small improvement can result in huge saving in yearly fuel costs. Because, the flow in the hood is highly turbulent, non-symmetrical three dimensional, flow analysis is required to identify uneven velocities. Studies of internal flow through exhaust hood are used to compare the relative improvements. The present work is based on detailed experimental investigation [7] to validate measurement data and provides cost effective solutions for different configurations.

2. PROBLEM DESCRIPTION & GEOMETRY MODELLING

The laboratory model exhaust hood has been considered in the present analysis. The dimensions of component comprise the outer casing, conical section, diffuser profile with steps and curvature has been modeled to study the aerodynamic performance. To reduce the complexity, the internals struts, and ribs/drain pipes which supports for structural stability of system has not been included in the present analysis. The inlet boundary for the model is located at a plane corresponding to the last blade row. The exit plane is located below the spindle center line at a position corresponding to the condenser flange. Using the pre-processor HyperMesh, with the help of geometric entities, three dimensional models for the exhaust hood has been created. The dimensions of exhaust hood components inner cone, upper/lower portion, height/width and diffuser profile dimensions detailed in the geometrical sketch are used. Profile of diffuser which has steps and curvature is the critical internal in the exhaust hood chamber modeled accurately.
3. COMPUTATIONAL MESH GENERATION

Meshing or grid generation is an integral part of the computer-aided engineering (CAE) analysis process [8]. The mesh influences the accuracy, convergence and speed of the solution. Furthermore, the time it takes to create a mesh model often consumes the significant portion of the project duration of CFD simulation for the complex components. Therefore, the more automated the meshing tools, faster the convergence of equation residuals and better the solution. The mesh generation process in this system has started with creation of 2d quad elements in the diffuser and parting planes. The 2d elements shown in red, green and blue are copied to another collector and dragged about the arc to obtain the three dimensional grid in 90° sector exhaust hood upper portion. This grid about a base point located centre of exhaust hood outer casing is reflected to other 90° sector of upper portion. The two grid blocks in 180° of geometry comprises duplicate elements at mating surface are equivalence so that 3d mesh generated for upper portion of exhaust hood is free from the presence of walls. In order to generate the computational mesh towards bottom portion, firstly red portion 2d grid is dragged along 90° s arc in the negative direction.

Similarly, the plane grid shown in green colour is dragged along circular arc to obtain grid towards bottom portion of 90° exhaust hood. With the help of a base point, these grid blocks are reflected to obtain three dimensional bottom half portions. The duplicate elements present in these four parts grids at five mating surfaces are removed through equivalence command. Due to complexity of exhaust-hood outlet portion which has been divided into three blocks, a separate plane grid showing in yellow region matching the elements in directions are created and dragged along the boundary edge of exhaust-hood. The yellow portion 2d grid has been is translated to next block and dragged along the outer boundary edge of geometry. Finally the yellow portion 2d is translated to next block and dragged along the boundary edge. These three grid parts with the base point are reflected to other side of 90° sectors. The duplicate elements at nine mating faces are removed by equivalence. In order to obtain the three dimensional grid at exit section, a separate plane grid shown in purple has been created and translated along the boundary edges. The presence of duplicate elements generated to this part at five mating surfaces in different parts is removed through equivalence commands. The assembled grid which is free from grid distortion and duplicate elements are shown in the Fig-2.
4. BOUNDARY CONDITIONS AND FLOW SIMULATION:

Computational grid comprises 2.20 lakh nodes generated through multi-block grid has imported in solver pre-processing and visualized its quality in terms of mesh distortion and scaled the same into meters. Flow characteristics are obtained using boundary conditions mass flow of 21kg/s at and static pressure at entry and exit locations. The working fluid is steam whose real gas properties are accounted through material libraries.

To account the viscous losses, two $k$ (intensity) - $\varepsilon$ (dissipation) with standard wall functions are used. The temperature for the walls and fluid at inlet are assumed as 373° K. The initialization of flow, turbulent and temperatures from the inlet of computational domain has been made.

With the choice of discretization scheme upwind 2nd order with default time step has been used to iterate the mass, momentum and energy equations till the equation error residuals are reduced to the order of 1e-04 in about 300 steps. After convergence of the simulation, computation of loss coefficients using surface integrals of mass weighted averaged variables computed from the simulation. Flow Solver software Fluent was used to compute governing equations to obtain the three dimensional flow characteristics of turbine exhaust hood system.

The governing equations based on conservation of mass, momentum energy principles and presence of viscosity give rise to additional transport equations to define turbulence fluctuations and energy dissipation. Complexity of different turbulence models may vary depends on the accuracy level of predicting viscous losses. Turbulent flows are characterized by large, nearly random fluctuations in velocity and pressure in both space and time. These fluctuations arise from instabilities that grow until non-linear interactions cause them to break down into finer and finer whirls that eventually are dissipated by the action of viscosity.

Currently, there are three basic conceptual alternatives for the numerical simulation of turbulence: Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), and simulations based on the Reynolds Averaged Navier-Stokes (RANS) equations. First two approaches for prediction of turbulence is computational intensive and quite involved execution activity in terms of time and resources. Solving the RANS equations with turbulence models that can quickly predict flow distributions.

7. RESULTS & DISCUSSIONS:

The magnitudes of velocities in the diffuser are higher reduces as it travels towards exit location. It is evident from the flow field, that flow in the exhaust casing is characterized by pronounced vortices which are accompanied by energy losses. Much of the upper half flows, however, turns upward and backward around the flow guide and develops into a large horseshoe vertex. These eddies can cause the energy loss, which affects the aerodynamic performance of exhaust hood system. In order to obtain better insights, flow lines on the velocity scale are drawn from the inlet and are shown in Fig-3. Since the total pressure remains substantially constant, the local fluid velocity must increase near the inner radius of the bend. This unequal distribution of velocities causes the average kinetic energy to increase. If the velocity at the inner radius increases, continuity considerations then demand that the local cross-sectional area through which the flow stream decrease to compensate for the increased velocity. From the streamlines plot, the flow path after travelling circumferentially around inner cone becomes straight towards exit location.

The Mach number distribution of flowing fluid in middle plane of the exhaust hood system is described in Fig-4. In the upper part of the axial-radial diffuser the flow is subsonic. At the diffuser outlet section high Mach number is noticed which however becomes lower towards exit locations. The flow turning around the upper edge of the diffuser cowling results in an intensive vortex with the circumferential and a respective pressure drop inside. Both the flow separation and the turbulent eddies contribute to the pressure loss.
The loss parameters are computed using mass weighted averages in the solver post processor. The efficiency of the exhaust hood is affected by the local flow distribution within the hood. Eddies and vortices give rise to losses which cannot be recovered. Increased velocities of the fluid between inlet and outlet of the hood are undesirable, since they result in an increase in the static pressure difference between inlet and outlet of the hood. Since the outlet of the hood is connected to a constant condenser pressure, an increased static pressure difference can thus only result in a higher last-stage pressure and reduced efficiency of the turbine [11]

9. CONCLUSIONS:

The CAD model for axial exhaust hood of 1.8sq m used in low pressure turbine has been generated from the dimensional drawing and generated structured multi-block computational grid. Using the Fluent commercial CFD solver, three dimensional compressible turbulent steam flow has been simulation with the prescription of mass flow/pressure at inlet/exit locations.

From the flow, the analysis exhibits that in the upper half portion, flows turns upward and backward around the flow guide and develops into a large horseshoe vertex. The flow lines drawn from the inlet location traces helical paths and generates strong vortices in parting plane and grows as the travels towards the exit location.

Eddies and vortices give rise to losses which cannot be recovered. Increased velocities between inlet and outlet of the hood are undesirable, since they result in an increase in the static pressure difference between inlet and outlet of the hood.

Simulation of steam flow inside the exhaust hood is subsonic and its magnitude at diffuser outlet is very high. The loss coefficients from the simulation for different cases compares well with measurements obtained for laboratory test models.

The findings of turbulent flow in exhaust system obtained in these paper valuable insights to designers futuristic investigations to address multi-disciplinary simulation involving two phase flows with fluid-solid interaction effects.
REFERENCES:


[3] Lagun VP and simoyu LL, Result from a study of the exhaust hood of a full scale for pressure cylinder In a VK100-5 turbine, Thermal Engineering, Vol 15, 1968


[10] Rhodes, the Prediction of turbine Exhaust Performance, pp. 99-105, private communication