Leakage flow modelling in gas turbines plan

Engineering

Introduction ANSYS computational fluid dynamics (CFD) simulation software gives you the capability to make predictions withsurety the possible impact of fluid flow in your turbo engine all through the design phase and also during the use by the end user. ANSYS CFD solutions are fully integrated into ANSYS workbench platform which has a number of capabilities such as processing, simulation, post processing and multi physics functionalities.

Objective

To create a plan for the design of an ANSYS CFD in Leakage flow modeling in Gas Turbines.

Plan

For the system to be designed and modeled to completion, the following must be done:

1. Structural analysis

This will involve analysis of factors such as the static, buckling capabilities, harmonics of the system, strain and elasticity. This ensures that the structural integrity of the system is maintained.

Structural analysis is estimated to take approximately 2 weeks to completion.

2. Geometric non linearity and material modeling analysis

In this phase, the tasks involved will include analysis of strain and deflection capabilities of the system. This phase is estimated to take about 3 weeks to be completed.

3. Contact modeling

This will include analysis of factors such as friction and roughness of the moving parts of the system and the parts that are in contact with each other. This section is estimated to take about 3 weeks to be completed.

4. Advanced analysis

In this phase, the tasks involved include testing for functionalities such as rotor dynamics and rezoning of the system sections and functionalities. This phase is estimated to take about 2 weeks to be completed.

5. Thermal analysis

In this phase, thermal processes such as conduction, convection, radiation and phase change are tested and analyzed. This section is estimated to take approximately 3 weeks to completion.

6. Modeling capabilities evaluation

This section will include testing and evaluation of modeling capabilities in 2D and 3D flows, transient flows, steady state flows, turbulence and viscoelasticity. This phase is expected to take about 23 weeks to be completed.

7. Solver options

This includes testing for pressure based solvers, coupled solvers and density based solvers. This section does not involve much analysis and is expected to take approximately one week.

8. Magnetic transient

This phase includes evaluating for rigid motion visualization and translation motion of the leakage system. For this to be achieved, an approximate period of about 3 weeks is required.

9. Advanced materials characteristics evaluation

Core loss mobility, insulation sheet to models cracks, non linear magnetization characteristics. The evaluation of the materials is approximated to take about 3 weeks.

10. Preprocessing

This is the final stage of the design modeling. It will involve meshing of the turbine. This will be accomplished through a number of methods such as Parallel meshing, Prism inflation layers, Swept hex meshing, Cut cell Cartesian meshing and adaptive mesh refinements.

This final stage is expected to take about 3 weeks to be completed.

Conclusion

The for the entire leakage system to be modeled successfully, an estimated period of about 4 months is required taking into account the fact that each task will be performed independently.

References

N. Watson, M. S. Janota (1982) Turbo charging the internal combustion engine–Ed. Wiley

ANSYS 15. 0 Capabilities user guide http://www. ansys. com/staticassets/ANSYS/staticassets/resourcelibrary/brochure