

Prediction of Flow in Stationary Turbine Blade Cooling Passage with OpenFOAM

Avinash Jagdale¹, Dr. Sachin Borse², Dilip P.Borse³

P.G. Student, Centre For Modelling and Simulation, S. P. Pune University, Pune, Maharashtra, India¹

Professor, Department of Mechanical Engineering, Rajarshi Shahu College of Engineering, Pune, Maharashtra, India²

Assistant Professor, Department of Mechanical Engg., Rajarshi Shahu College of Engg., Pune, Maharashtra, India³

ABSTRACT: Numerical prediction of turbulent flow field structure inside the cooling passage of gas turbine blade is carried out using OpenFOAM. High Reynolds number turbulence model $k-\epsilon$ is used. The simulation is run in parallel, on 6 cores. The results are then compared with the experimental values. The complex flow structures like secondary flows induced due to ribs in passage are observed. The geometry is created using open source pre and post processing tool, Salome. Meshing is performed using snappyhexmesh, native mesher provided with OpenFOAM.

KEYWORDS: CFD; OpenFoam; turbine blade cooling, parallel processing;

I. INTRODUCTION

Gas turbine cooling plays a critical role in increasing thermal efficiency of turbine. With increase in turbine inlet temperature, a great demand in more efficient blade cooling technology exists. The turbine inlet temperature may range from 1500K to 1800K. Since this temperature is above the limit that blade material can resist it requires cooling. The cooling air is extracted from compressor. Since there is a practical limit on extraction of cooling air, as it affects efficiency of turbine, there is a high demand on efficient utilization of cooling air. The internal cooling is provided by introducing cooling air through turbine shaft into internal hollow passages of blade through blade's base. This air after passing through different passages exits through tip and trailing edge. Internal passage is further divided into 3 parts depending upon different cooling requirement for leading edge, center portion, and trailing edge of the blade. Leading edge passage- jet impingement cooling. Serpentine passage- two 180 degree bends with ribs to enhance turbulence. Trailing edge passage- pin fins to enhance heat transfer by increasing contact area and turbulence [1]. Film cooling holes are provided to cool external surface of the blade by forming a film of cooling air over the external surface. For the present study we have not incorporated film cooling holes. Experimental studies on flow distribution are reported by Borse and Date [2]. Hwang et. al. [3] has given typical dimensions of gas turbine blade cooling passage parameters.

II. METHODOLOGY

Here CFD study was done using open source CFD software OpenFoam. 3D geometry was created in open source pre and post-processing software, Salome V7.3. Tools like 2D sketch, building faces from closed sketch, and extrusion of the face is done. The geometry is checked for error like independent edges, internal faces, closeness, intersecting faces. Once it is free of any error, geometry is exported into STL (STereoLithography) format, to be used for meshing. Each boundary element to be defined is exported individually. Later all the STL files are combined into a single file. STL format is surface representation of geometrical model. The surface is represented as a collection of triangular faces with their respective surface normal. Below is the picture of geometry in STL format. STL file is shown in Figure 1.

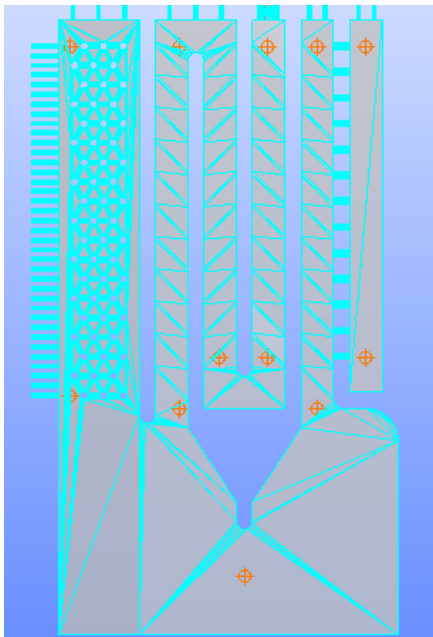


Figure 1: Geometry in STL with pressure probes

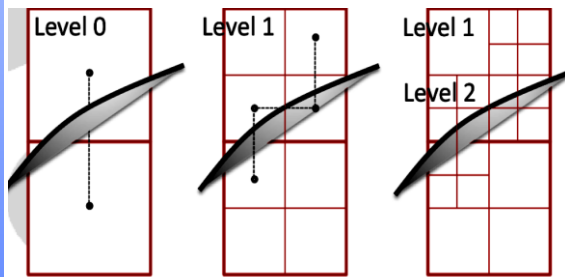


Figure 2: Refinement levels

Mesh is created using snappyHexMesh, OpenFOAM's native meshing tool. Utility snappyHexMesh is used to create high quality hexdominant meshes based on arbitrary geometry. It is controlled by parameters in the file snappyHexMeshDict. It can be executed in parallel. It preserves the feature edges, addition of wall layers. At first a bounding box is created around the geometry, using blockMesh (multiblock structured meshing tool) provided with OpenFOAM. This block is meshed such that each element is close to cube. The edge feature of geometry is extracted using surfaceFeatureExtract tool. All the edges with angle between intersecting faces below 150 degree are extracted and stored in emesh format. Figure 3 and Figure 4 shows mesh of domain obtained.

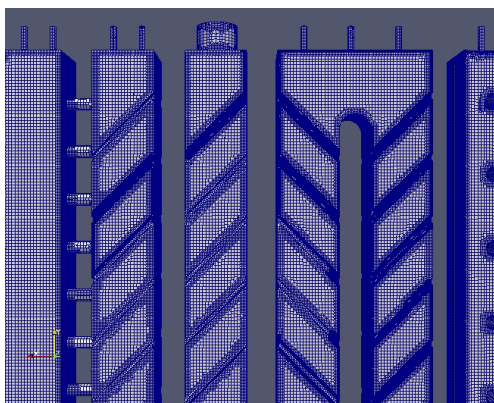


Figure 3: Mesh

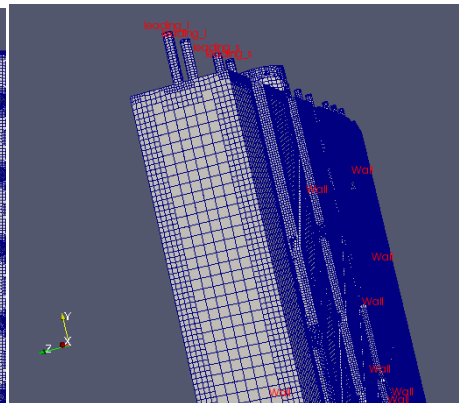


Figure 4: Mesh in passage

Computation was performed for inlet mass flow rate and exit pressure at outlets. Exit pressure was specified from experimental values of Borse [1]. Values were determined with respect to highest exit pressure set to zero, to prevent back-flow in domain due to positive exit pressures. No slip condition was assumed at wall. Velocities at wall boundary are set to zero. Inlet velocity specified as 3.4 m/s. At outlet zero gradient of velocity specified. Turbulent kinetic energy k is calculated from mentioned formula to $0.0528 \text{ m}^2 \cdot \text{s}^{-2}$. At wall, wall function is used and at outlet zero gradient is

specified for k . Dissipation rate is calculated to be $0.7\text{m}^2\text{s}^{-2}$. Turbulent length scale was assumed to be 7% of hydraulic diameter at inlet. Kinematic viscosity of air at ambient temperature was given as $10^{-5}\text{m}^2\text{s}^{-1}$.

All convective terms were discretized using upwind scheme, because of boundedness property. Laplacian term is discretized using Gauss linear scheme. Face interpolation is done using linear scheme.

GAMG - (generalised geometric-algebraic multi-grid solver) was used for pressure. GAMG uses the principle of generating a quick solution on a mesh with a small number of cells, mapping this solution onto a finer mesh; using it as an initial guess to obtain an accurate solution on the fine mesh. Its aim is to be faster than standard methods when the increase in speed by solving first on coarser meshes outweighs the additional costs of mesh refinement and mapping of field data. smoothSolver - solver using a smoother for both symmetric and asymmetric matrices, is used for all other variables. Since the mesh non-orthogonality is sufficiently small, no non-orthogonal corrector is applied. The residual control for all variables is set to 10^{-4} . Relaxation factor for pressure is 0.05, 0.1 for all other variables. Maximum number of iterations are set to 10,000. simpleFoam solver in OpenFoam was used with parallel computations, with 6 cores. The domain was decomposed with scotch decomposition provided with openFoam. It decomposes in a way to optimize the computation time for given cores.

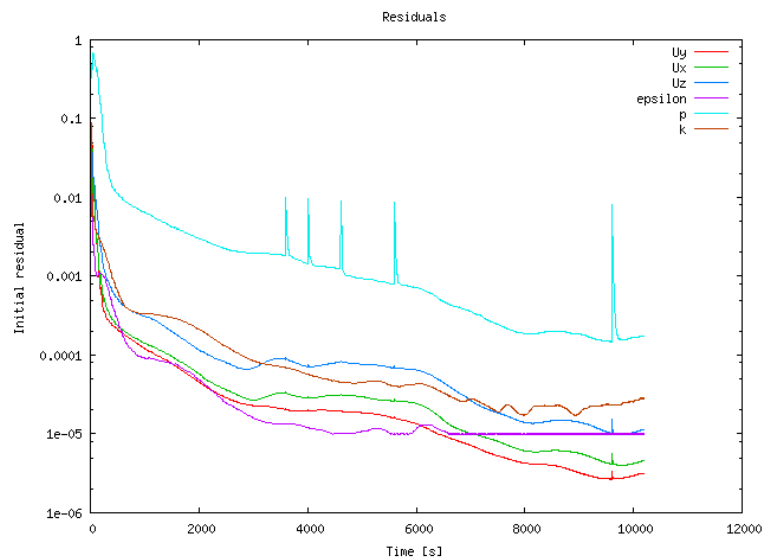


Figure 5: Residual Plot

While residual for all U , k , ϵ were below 10^{-4} , residual for pressure fell upto 10^{-3} only. Hence the simulation ran for 10,100 iterations. The simulation ran for 150 minutes. Mass flow rate and pressure at inlet and outlet boundaries were calculated. There is a large difference in inlet pressure prediction with error of 25%. Residual plot is shown in Figure 5. Results are compared with experimental results of Borse and Date [2].

III. RESULTS AND DISCUSSION

Pressure drop in given passage is computed and compared with experiments. Table 1 show calculated mass flow and pressure. Table 2 gives mass flow and pressure. Table 3 shows calculated pressure drop in passages. Table 4 gives experimental pressure drop in passages. Little pressure drop takes place in leading edge passage as compared to pressure drop in serpentine and trailing edge passage, this supports experimental findings. Pressure drop in serpentine passage visible in the figure, due two bends in serpentine passage. Considerable pressure drop occurs in tip and lateral ejection holes.

Passage	massflow	% m_{exit}	pressure
trailing_tip	0.000732014	3.1%	151.07
trailing_bleed	0.0119606	51.9%	1071.25
serpentine_tip	0.000711776	3.0%	136.35
serpentine_end	0.00856376	37.2%	856.41
leading_s	0.000523513	2.2%	82.40
leading_l	0.000525924	2.28%	82.40
total exit	0.02301759	100%	
inlet	-0.0230176	99.9%	3092.94(calculated)

Table 1: Calculated mass flow and Pressure

Passage	massflow	% m_{exit}	pressure
trailing_tip	0.000813	3.23%	151.07
trailing_bleed	0.012473	49.59%	1071.25
serpentine_tip	0.000688	2.74%	136.35
serpentine_end	0.010127	40.27%	856.41
leading_s	0.000534	2.12%	82.40
leading_l	0.000515	2.05%	82.40
inlet	-0.02556	99.9%	4130.01

Table 2: Experimental mass flow and Pressure [2]

Passage	Passage Pressure Drop	Total Pressure Drop
leading edge passage _L	2.64($P_5 - P_6$)	3009.6 ($P_0 - P_{LS}$)
leading edge passage _S	0.89($P_7 - P_8$)	3009.6 ($P_0 - P_{LS}$)
serpentine passage _T	308.72($P_3 - P_9$)	2235.59 ($P_0 - P_{SE}$)
serpentine passage _E	998.712($P_3 - P_4$)	2235.59($P_0 - P_{SE}$)
trailing edge passage	98.65($P_1 - P_2$)	2940.93 ($P_0 - P_{TT}$)

Table 3: Calculated Pressure drops

Passage	Passage Pressure Drop	Total Pressure Drop
leading edge passage _L	4.26($P_5 - P_6$)	4047.61 ($P_0 - P_{LS}$)
leading edge passage _S	1.45($P_7 - P_8$)	4047.61 ($P_0 - P_{LS}$)
serpentine passage _T	374.71($P_3 - P_9$)	3273.60 ($P_0 - P_{SE}$)
serpentine passage _E	1486.22($P_3 - P_4$)	3273.60($P_0 - P_{SE}$)
trailing edge passage	142.25($P_1 - P_2$)	3978.94 ($P_0 - P_{TT}$)

Table 4: Experimental Pressure drops

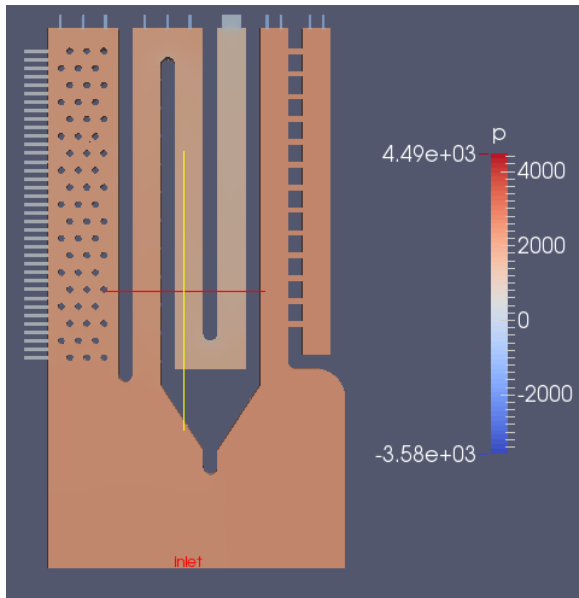


Figure 6: Pressure contours at symmetryplane

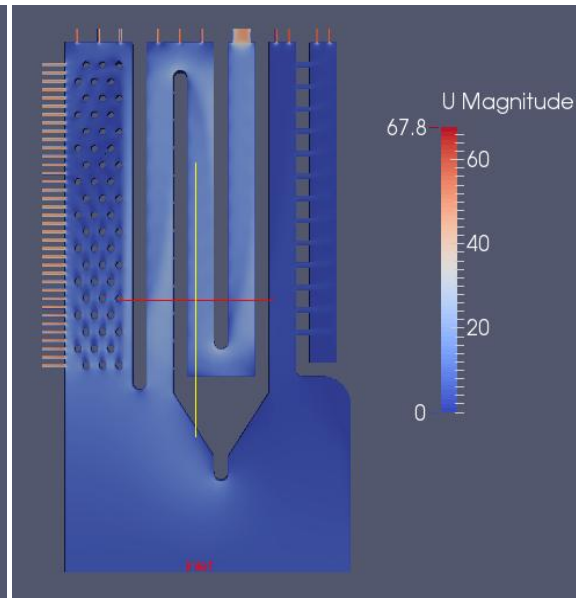


Figure 7: Velocity contours at symmetry plane

The figure 7 shows high velocity in trailing and serpentine passage as compared to leading edge passage. In first and second bends of serpentine passage shows pockets of low velocity at corners. Trailing edge passage clearly shows flow going to lateral ejection holes. The secondary flow developed at serpentine passage due toribs can also be seen in Figure 8.

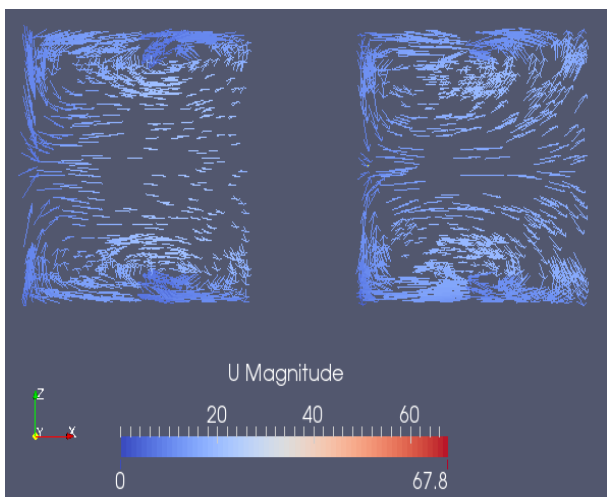


Figure 8: Secondary flow at serpentine cross-section

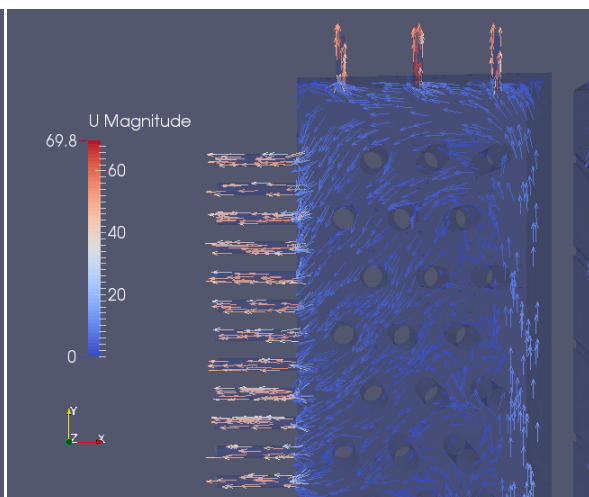


Figure 9: Flow in trailing edge around pin fins

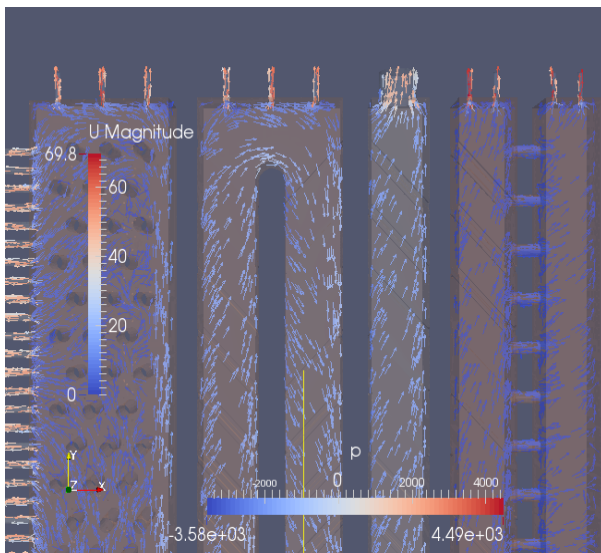


Figure 10: Velocity and Pressure

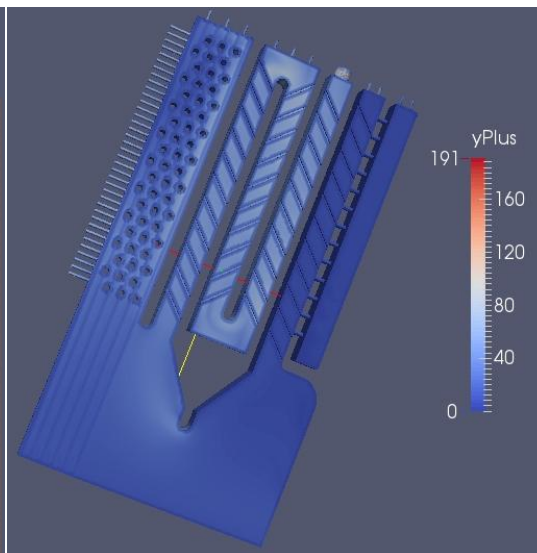


Figure 11: Yplus distribution

Yplus value calculated min: 0.15 max: 226 average: 16.87. Figure 11 shows value of Yplus distribution. Low value in leading edge passage might be reason for error. The reason for low Yplus at leading edge is due to low Reynolds number in the passage. Yplus in serpentine and trailing edge is well above 30.

IV. CONCLUSION

k-ε model was applied on the geometry with 15 lakh mesh elements. Simulation ran for 10,000 iterations, with U, k, ε residuals falling below 10^{-4} and pressure residual failed to fall below 10^{-4} . The simulation was carried out using OpenFOAM in parallel with 6 cores. It took 150 minutes. The predicted mass flow distribution deviated from measured value by 4% to 11% which is very good prediction. Inlet pressure prediction showed high error of 25%. y+ value for leading edge passage is much lower than 30, which is undesirable and might be the reason for error. The qualitative trend in velocity was that expected, e.g. the secondary flow induced due to ribs, flow in serpentine bend. The possible source of error might be, y+ values, Domain of fluid flow - the inlet of the domain might not be the correct representation of the given inlet condition, k-ε model might not be able to resolve the flow accurately.

V. FUTURE WORK

Use of other turbulence model e.g. SSTk - ω, to check its applicability. By applying constant heat flux through wall, possible hotspots in the domain can be found. The actual blade is in rotating motion, hence studying the rotational effect on the flow structure, using Single reference frame (SRF) modeling approach. Search for more efficient geometry for heat transfer, once the model is sufficiently validated.

REFERENCES

- [1] J. C. Han, Sandip Dutta and Srinath Ekkad, Gas Turbine Heat Transfer and Cooling Technology, Taylor & Francis Inc. London, 1st Edition, 2-25, 2000.
- [2] S. L. Borse and A. W. Date, "Flow Distribution inside Internally Cooled Gas Turbine Blade without Film Cooling Holes" 18th National & 7th ISHMT-ASME Heat and Mass Transfer Conference, IIT Guwahati, India, January 4 - 6, paper no. HMTTC-2006-C283, pp. 2036-2041, 2006.
- [3] G. J. Hwang, S. C. Tzeng, C. P. Mao and C. Y. Soong, "Heat Transfer in a Radially Rotating Four -Pass Serpentine Channel with Staggered Half-V Rib Turbulators", J. Heat Transfer. **123**, 39-50, 2001.