Conjugate Heat Transfer Analysis of Gas Turbine Disc and Cavities

[1]Chethan K R , [2] Dr. H R Purushothama.
[1]*M.Tech, Project trainee, HTG, GTRE, DRDO.*
[2]*Professor, Mechanical Engineering Dept., SIT, Tumkur[3], Corresponding author.*
City-Tumakuru, Country-India.

Abstract
The Contemporary gas turbines are operated at very high main stream inlet temperatures to increase the thrust. The turbine inlet temperatures are far above the material temperature limits for turbine blades, these heat from the blade is conducted to blade platform and also to turbine disc. Hence the air is tapped at some stages of compressor and used as coolant. Thermal gradients are developed inside the material as it is exposed to low temperature secondary air. Thermal stress will develop inside the component if the thermal gradient factor is more, this will causes malfunctioning of the components which leads to failure of the disc. In order to avoid this, first we need to know the temperature distribution and thermal gradient over the material. The heat from the blade is transferred to disc and then heat will be taken away by the secondary air by convection. This process can be properly capture by conjugate heat transfer analysis. The primary aim of this project is to estimate the temperature (temperature gradient) on discs and also recognize the flow path inside the cavities. Hence conjugate heat transfer analysis is carried for the static and rotational case by considering it as 2D axi-symmetrical geometry. Mesh has been generated in Ansys pre-processor with acceptable mesh quality of 0.45. The analysis is carried out by using k-ε turbulence model with enhanced wall function, using Ansys Fluent16.0.

Keywords— Turbine disc, Secondary air flow, Cavities, Conjugate heat transfer.

1. INTRODUCTION

The modern gas turbine engine blades are exposed to high temperature (≥1600°C) main stream flow (high temperature and high pressure burnt gas). As per the increase of temperature, heat from the blade is transferred to turbine disc. Hence there is a necessity of cooling the turbine disc and also hot end parts of the engine. The increase in the temperature causes to increase in the efficiency (thrust) of the engine. The mass flow rate also place a major role for increasing the thrust. For this some part in secondary air flow is considered and analysis is carried for the fixed clearances for static and rotational cases.

P. R. Farthing Conducted experiment on two rotating cavities rigs to obtain heat transfer measurement and temperature distribution in disks and the effect of radiation is insignificant. The experiment was studied in cases, for the first case where two discs have similar distribution of temperature and also recognised that the Nusselt number is similar. For the second case in which one disk is hotter compared to other disk, for that Nusselt number were similar to that of cavities in which the temperatures were distributed symmetrically but varying magnitude. The increase in the temperature distribution radially corresponds to increase in Nusselt number radially and decrease in the temperature distribution radially corresponds to decrease in Nusselt number radially. The results obtained from two test rigs are compared and also tested for different gap ratios have no major effect on Nusselt number, from that information basic relation was generated between Nusselt number and Grashof number and Reynolds number [1].

S. H. Bhavani Conducted experiment on coolant flow in cavities for particular length of turbine disc. Pressure and temperature distribution are compared with theoretical data. Good match is found by comparing with earlier published data. They concluded the flow path is strongly dependent on the seal clearance and gap between stator and rotor. [2]

2. COMPUTATIONAL DOMAIN

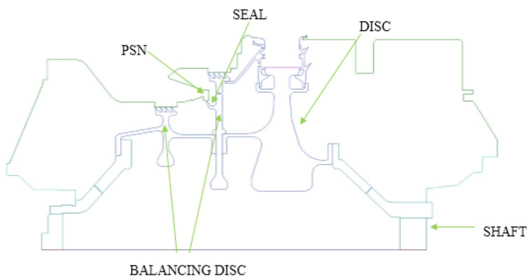


Fig 2.1: 2D axi-symmetric AUTOCAD geometry.

The computational domain consists of turbine disc, balancing discs, shaft, and it also having outer stationary wall as shown in fig 2.1.

3. GEOMETRY MODIFICATION

AutoCAD two dimensional geometry consists of whole gas turbine engine assembly (Turbine disk, blades, bearings, shafts) which includes secondary air path. This 2D axi-symmetric geometry is modified according to the requirement like clearance between stator and rotor (turbine disk and stator, balancing discs), the below figure shows the 2D axi-symmetric AutoCAD geometry after modified.

The given geometry is modified by changing the holes in the circumferential direction to the equivalent slat width, because of the 2D axi-symmetric nature of geometry. This could be done by using mathematical formula is as follows.

Product of number holes and area of holes in the Circumferential direction	=	Circumferential equivalent slot width area

$$n(\Pi/4)D^2 = 2\Pi rh$$

Modification has been done according to our analysis requirement by the changing the clearances for the top, bottom balancing discs and seal (a, b & c)) and outlets (O1 to O4). The main intension is to make target mass flow over the outlet as well as to eliminate the problems like leakage of main flow gases in to the secondary air flow channel. Equivalent slot width modification done at the pre swirl nozzle and at the shafts (1, 2, 3 & 4). Modified geometry is shown in below.

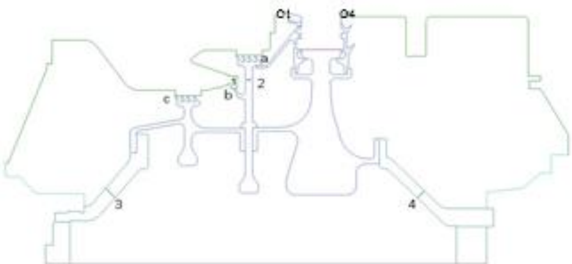


Fig 3.1: 2D axi-symmetric AutoCAD geometry after modified.

Values for modified clearance, gap between stator and rotor for the current analysis is tabulated in below table 1.

Table3.1: Non dimensional values of inlet, outlets and clearences.

Positions	Modified value
Inlet	2.28
Outlet 1	3.42
Outlet 2	2.85
Outlet 3	9.75
Outlet 4	3.4
Position a	1.45
Position b	1.3
Position c	1.07
Position 2	1.3
Position 3 and 4	3.71

4. MESH GENERATION

The grid choice has been done first, weather to go with structured are unstructured mesh type. Unstructured mesh has an advantage like it requires less time to mesh called as automatic mesh, but proper preparation of the geometry should be done. For current work we tried with unstructured mesh type. The correctness of CFD results are mainly influenced by grid system and grid finesse near the boundaries to simulate flow, capture the physics behind the flow. It also dependent on the grid quality. Steps used to generate the mesh is as follows.

- Modified 2D AutoCAD geometric model is converted in to .IGS format.
- The .IGS format model is imported in to Ansys pre-processor 16.0.
- Surface creation has been done for the entire geometry.
- Naming the parts in pre-processor.
- Surface mesh option is used to create the mesh all over the surface.

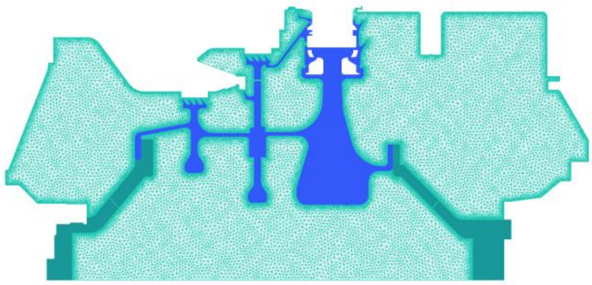


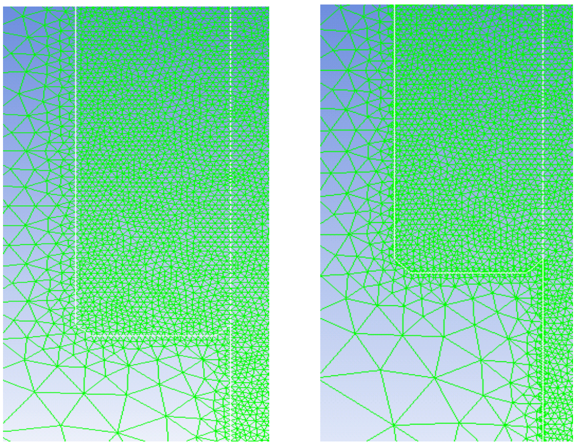
Fig 4.1: Geometry with mesh generation.

5. MESH ADAPT

Mesh adapt has been done for the improvement wall y+ value. First we did mesh as course near the wall boundaries. After simulation y+ value is recorded. Adaptation is done to increase the wall y+ value near the turbine disc and balancing discs in Ansys Fluent.

Table 5.1: Mesh characteristics

Mesh name	Cells	Y+ value
M1	872031	7
M2	1 Million	4
M3(after adaptation)	1.2 Million	2



Before mesh adapt After meash adapt

Fig 5.1: Mesh near the wall before and after mesh adapt.

6. BOUNDARY CONDITIONS

Boundary conditions depends on the type of the physics behind the problem, like steady state, fluid flow, thermal problems. For the current analysis boundary conditions are applied to the flowing fluid, stationary wall and rotating parts.

Flow boundary conditions are applied to inlet and outlet Flow boundary conditions in ANSYS Fluent permits flow to enter and exit through computational domain. Heat transfer boundary conditions are applied to walls and rotating boundary condition are applied to shaft and disc. Location of inlet and outlets are shown below.

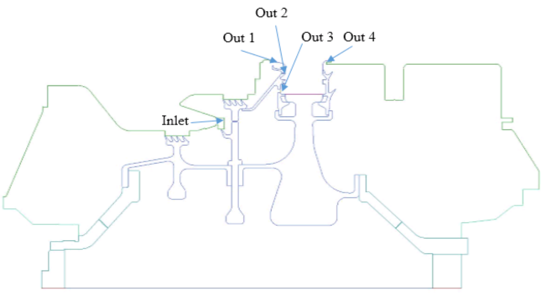


Fig 6.1: Location of inlet and outlets in geometry

Table 6.1: Non Dimensional flow boundary conditions

	Positions	Pressure	Temperature
Inlet	Inlet	2.017	2.78
Outlets	Outlet 1	1.605	1
	Outlet 2	1.605	1
	Outlet 3	2.021	1
	Outlet 4	1	1

Table 6.2: Non dimensional Rotational boundary condition

Heat transfer coefficient	2.2
Temperature	1

7. RESULT AND DISCUSSION

In ANSYS fluent convergence criteria includes residuals plot and surface monitor plots. In our cases total mass flow convergence and temperature at some points at high pressure turbine disc is considered as surface monitor. The convergence history plots are shown below

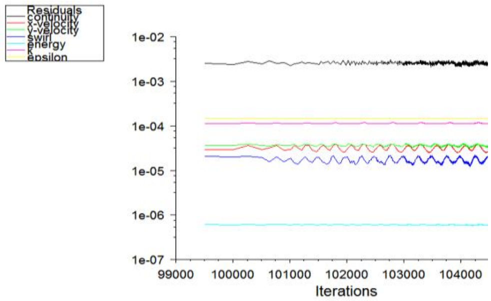


Fig 7.1: Convergence criteria for static case.

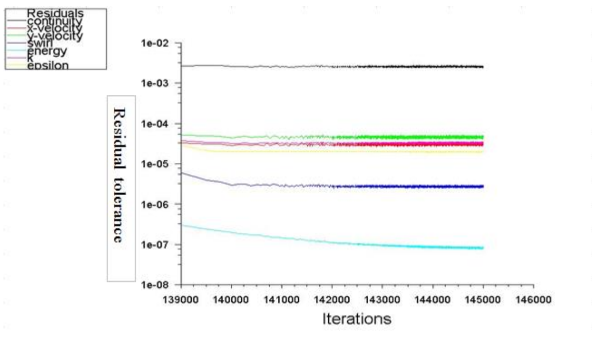


Fig 7.2: Convergence criteria for rotational case.

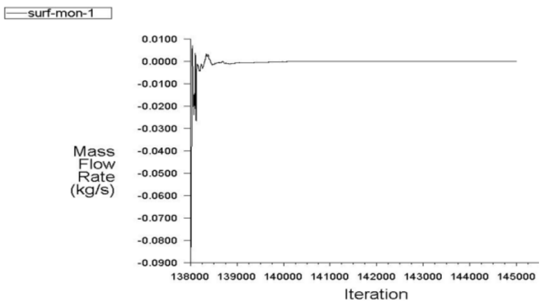


Fig 7.3: Mass flow convergence for static case.

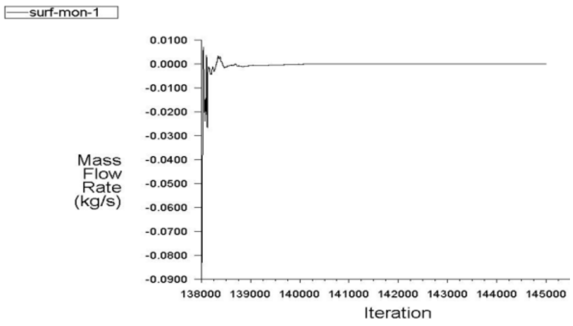
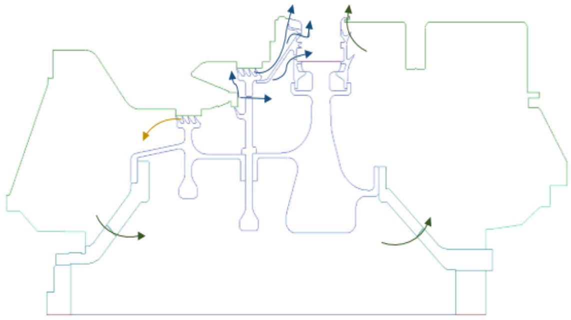


Fig 7.4: Mass flow convergence for rotational case.

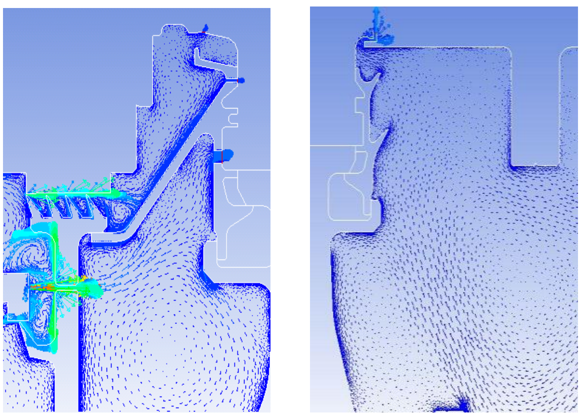
➢ FLOW PATH



Flow path 1 Flow path 2
Flow path 3

Fig 7.5: Actual flow path after simulation.

In our analysis flow path one, two and three are visualized by vector plots. From vector plots we can come to know whether the secondary air following pre-defined path. And also we observed that weather hot gas injection is takes palace to the secondary air flow (reverse flow) region, this information can get by observing vector plot direction at outlets. (If the direction of vectors is in opposite direction to flow path at outlets, which leads to hat gas injection in to the secondary air flow region. Below figures shows the flow path by vector plot.



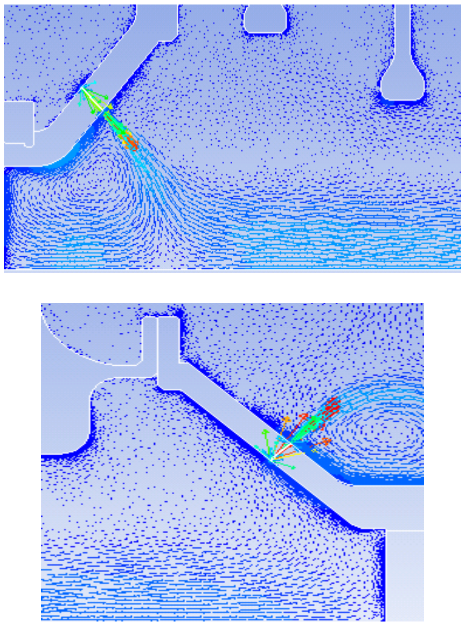


Fig 7.6: Flow path visualization by vector plots at different positions.

➢ TEMPERATURE CONTOURS

Non dimensional static and total temperature for two cases (without rotation and rotation) are shown in below figures, and the range of temperature is from 1 to 1.08 and 1 to 1.061 for static and total temperature respectively for the first case and the range of temperature is from 1 to 1.084 and 1 to 1.055 for second case.

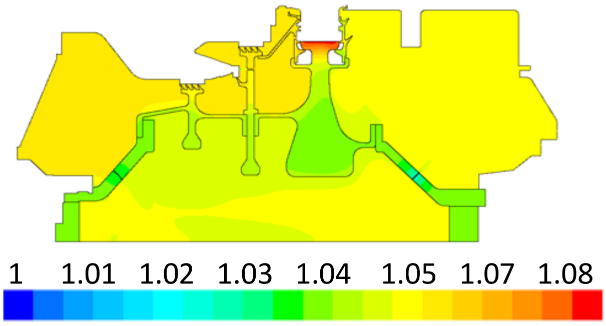


Fig 7.7: Static temperature contour for static case.

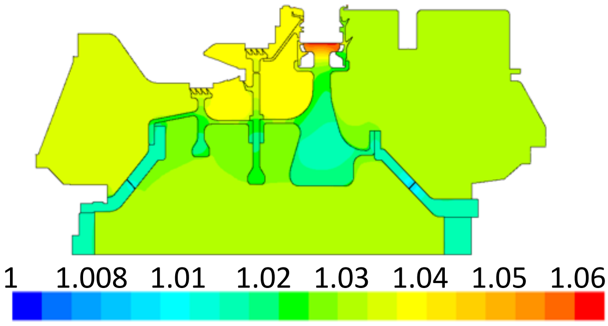


Fig 7.8: Total temperature contour for static case.

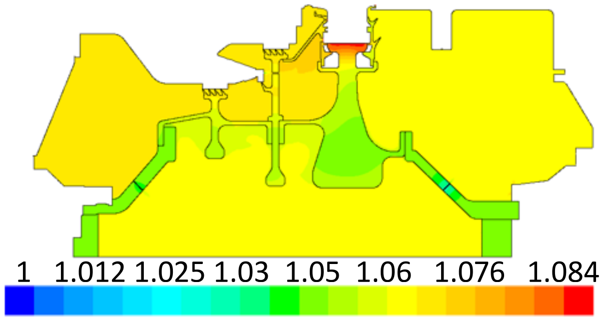


Fig 7.9: Static temperature contour for rotational case.

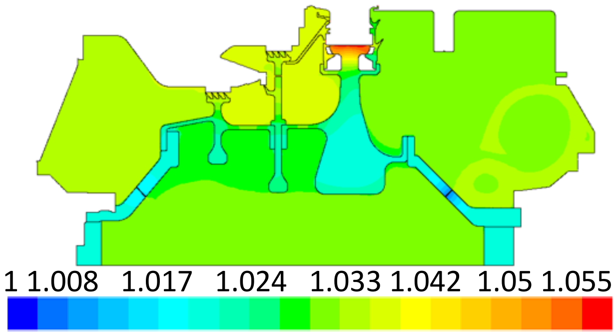


Fig 7.10: Total temperature contour for rotational case.

We absorb the total temperature in two cases in figure 5.7 and 5.9 the minimum total temperature in the first case (without rotation) is less compared to minimum total temperature in second case (rotation). From rotational boundary condition, one is rotating with high speed (HPT disc, balancing pistons) and fluid is not rotating. Because of this high speed and due to friction, viscous heat dissipation will occurs. So that the temperature near wall will increase, hence the minimum temperature in contours of total temperature region for rotational boundary conditions are more.

➢ PRESSURE CONTOURS

Non dimensional static and total pressure for rotational boundary conditions are shown in below figures, the range of Pressure is from 1 to 2.246 and 1 to 1.061 for static and rotation respectively. High pressure region is observed at top balancing piston and low pressure region is observed at right side of turbine disc.

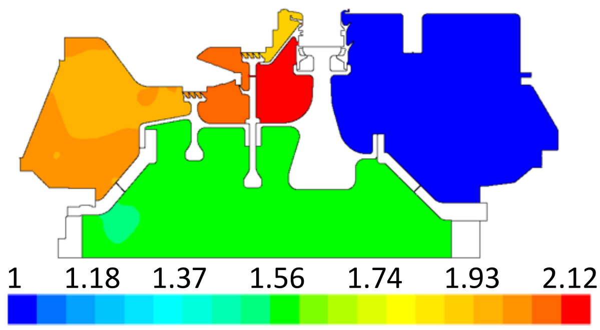


Fig 7.11: Static pressure contour for rotational case.

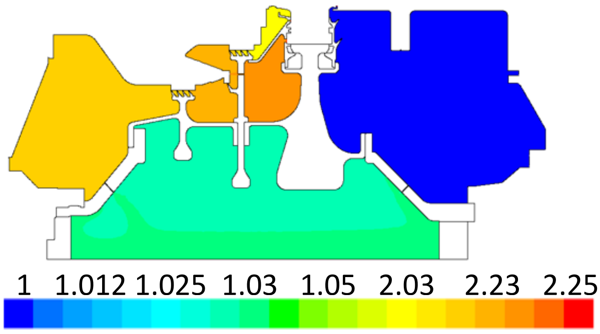


Fig 7.12: Total pressure contours for rotational case.

➢ VELOCITY CONTOURS

Non diemensional velocity values are ploted by velocity contours for the entire geomerty is observed in figure5.12. From below figure we can get the velocity of the fuid at different section in fluid domain. The non dimensional veloicty values is ranging from 0 to 1 Maximum non diemensional velocity is observed at holes near shafts, which is ranging from 0.95 to 1 and the velocity at pre swirl nozzle is ranging 0.1 to 0.7.

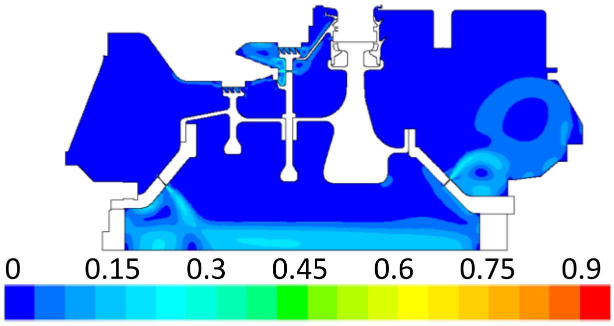


Fig 7.13: Total velocity contour for rotational case.

➢ MASS FLOW CONVERGENCE

The mass flow rates at different positions for without rotation boundary conditions are tabulated in below table.

Table 7.1: Non dimensional mass flow value at different sections for static case.

Positions	Mass flow rate(kg/s)
Inlet	1.22
Out 1	0.2098
Out 2	0.2037
Out 3	0.6293
Out 4	0.1801
Net mass flow rate	2e-4

Table 7.2: Non dimensional mass flow value at different sections for rotational case

Positions	Mass flow rate (Kg/s)
Inlet	1.289
Out 1	0.216
Out 2	0.2261
Out 3	0.659
Out 4	0.188
Net mass flow	1e-04

From the tables 8 and 9 it is noticed that the results of mass flow rate for rotational boundary conditions is more as compared to static boundary condition, and we found out there is a 5% increase in the mass flow for rotational case. The pressure difference generated across inlet to outlet, ie delta P (ΔP) for rotational case is more. Thus rotating components will impart higher kinetic energy to fluid which leads to increase in mass flow rate for rotational case as compared to static case.

➢ EFFECT OF ROTATION ON FLOW PATH

Clearly noticed the rotational boundary conditions will causes some effect on flow path. Because of the rotation the flow path in rotating cavities disturbed, hence flow path gets changes as compared to the static conditions. Rotating part will import energy to flow path in rotating cavities are shown in below figures 5.13 and 5.14 shows flow path in one of the rotating cavity for rotational and static case.

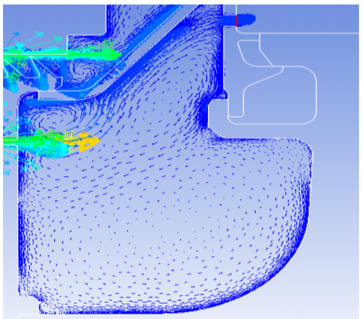


Fig 7.14: Vector plot showing flow path for rotational case

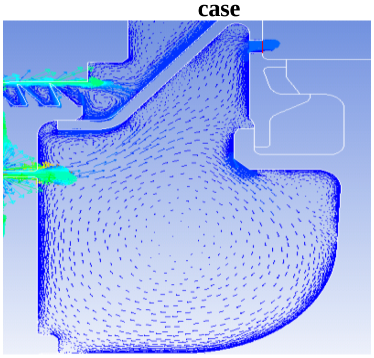


Fig 7.15: Vector plot showing flow path for static case

➢ TEMPERATURE DISTRIBUTION

Below graphs shows the distribution of temperature on turbine dis, balancing disc 1 and disc 2 along its length from the axis of rotation. The graph is plotted by considering total temperature and position from axis of rotation to top on Y and X axis respectively.

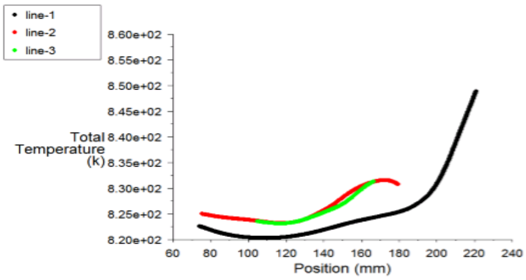


Fig 7.16: Distribution of temperature on turbine disc, balancing disc1 and disc2

Line 1, 2 and 3 represents the temperature distribution in turbine disc, balancing disc 1 and disc 2 respectively. The red line at 180mm comes down, because near pre swirl nozzle the flow is expanding suddenly. Due to the sudden expansion temperature is dropping in the flow. This effects reflects on the top balancing piston. The temperature at rim and bore for turbine disc, top and bottom balancing discs are recorded and tabulated in table 10, and temperature gradients are calculated according to rim and bore temperature data.

Table 7.3: Non dimensional temperature gradients

	Rim (TOP)	Bore (Bottom)	Gradient
HPT disc	1	0.969	0.031
Top balancing piston	0.975	0.970	0.005
Bottom balancing piston	0.978	0.968	0.01

8. CONCLUSION

A Conjugate heat transfer analysis of turbine disc and cavities was carried out using ANSYS Fluent 16 solver. An acceptable quality grid for the computational domain was create using ANSYS pre-processor 16 with 4.5×e6. The mesh was done using Tet elements. After discretizing the computational domain, coolant flow simulation were run. k-ε SST turbulence mode was selected to employ simulation and also used to analyses conduction and convection effects on high pressure turbine disc and cavities by coolant flow.

Convective heat transfer boundary conditions are applied at rim for turbine disc. Simulations were performed on two different boundary conditions, they are static and rotational boundary conditions.

- The value of total temperature is observed more in rotational case as compared to static stage. Since rotational boundary conditions are given only for disc and shafts not for the fluid, so friction between the fluid and rotating part will leads to cause viscous heat dissipation. Hence the total temperature in case of rotational is more viscous heat dissipation.
- The distribution of temperature on balancing disc are recorded, temperature gradient is also calculated and tabulated. The range of temperature for temperature disc, balancing disc one and disc two are in the allowable range.
- Flow path after simulation is analysed and compared with assumed flow path, which is following the predefined path.
- We noticed the flow path changes in rotational cavities by the effect of rotational boundary conditions.
- Mass flow rates at inlet and outlets are observed for both static and rotational cases, mass flow for the rotational case is more compared to static case. Because in rotational case the rotating part will import kinetic energy on fluid to move.

- **Acknowledgement**

This work was supported by “Gas Turbine Research Establishment,” DRDO Bengaluru and Siddaganga Institute of Technology, Tumakuru.

9. REFERENCE

1. P. R. Fathring[1], C. A. Long, M. J. Owen[2], J. R. Pincombe, “Rotating cavity with axial throughflow of colling air: heat transfer” journal of turbomachinery, Vol 144 January 1992.
2. S. H. Bhavani, J. M. Khodadadi, J. S. Goodling, J. Waggott “An Experimental Study of Flow in Disc Cavities”. Transition of ASME, Vol. 114, April 1992.
3. A. P. Morse, “Application of a low Reynolds number k-ϵ turbulence model to high- speed rotating cavity flows”, Transitions of ASME Vol. 113, Jaunuary 1991.
4. P. R. Farthing[1], C. A. Long, J. M. Owen[2], J. M. Owen[2], J. R. Pincomb, “ Rotating cavity with axial throughflow of cooling air: Flow structure” Vol 144 January 1992,
5. G. P. Virr, J. W. Chew, J. Coupland “Application of Computational Fluid Dynamics to Turbine Disc Cavities”. Journal of Turbomachinery, Vol. 116 October 1994.
6. Michael Ebert, Wei Shyy, Siddharth Thakur and Corin Segal, “Heat Transfer and Fluid Flow In Rotating Cavities”, 37th American Institute of Aeronautics and Astronautics Aerospace Sciences Meeting and Exhibit. AIAA-99-0737 January 11-14, 1999/ Reno
7. Gain Paolo Beretta[a], Enrico Malafa[b1], “Flow and Heat Transfer in Cavities Between Rotor and Stator Disc”. International Journal of heat and Mass Transfer 46 (2003).
8. Z. X. Yuan, N. Saniel, X. T. Yan, “Turbulent heat transfer on the stationary disc in a rotor- stator system”.International Journal of heat and Mass Transfer 46 (2003).
9. Peter E. J. Smith, Jonathan Mugglestone, kok Mu Tham, Christopher A. Long and Daniel D. Coren “Conjugate Heat transfer Analysis of Turbine Disc and Cavities”,ASME Turbo Expo 2012, Turbine technical conference and exposition, Vol.4 Heat transfer.
10. Yoji Okita and Shigemichi Yamawaki “Conjugate Heat Transfer Analysis of Turbine”. ASME Turbo Expo Vol.3 2002.