Proceedings of ASME Turbo Expo 2011
GT2011
June 6-10, 2011, Vancouver, British Columbia, Canada

GT2011-45569

CFD AERODYNAMIC PERFORMANCE VALIDATION OF A TWO-STAGE HIGH PRESSURE TURBINE

Sridhar Murari, Sunnam Sathish, Goswami Shraman
Honeywell Technology Solutions
Bangalore, India

Jong S. Liu
Honeywell Aerospace
Phoenix, AZ, USA

ABSTRACT

In the continual effort to improve analysis and design techniques, Honeywell is investigating on the use of CFD to predict the aerodynamic performance of a high pressure turbine. The present study has a two fold objective. The first objective is to validate the commercially available CFD codes for aerodynamic performance prediction of a two-stage high pressure turbine at design and off-design points. The other objective is to establish guidelines to help the designer to successfully set-up and execute the suitable CFD model analysis.

The validation to model the stage interfaces is performed with three different types of approaches such as Mixing Plane approach, Frozen Rotor approach and Non-Linear Harmonic approach. The film holes on the blade surface, hub and the shroud walls are modeled by using source term cooling and actual film hole modeling techniques for all the analysis.

The validation is accomplished with the test results of a two-stage high pressure turbine, Energy Efficient Engine (E3). The aerodynamic performance data at a design point and typical off-design point are taken as test cases for the validation study. One dimensional performance parameters such as corrected mass flow rate, total pressure ratio, cycle efficiency along with two dimensional spanwise distribution of total pressure, total temperature which are obtained from CFD results are compared with test data. Flow field results are presented to understand the aerodynamic behavior.

INTRODUCTION

With the advent of fast computers and availability of economical memory resources, computational fluid dynamics (CFD) has emerged as a powerful tool for the design and analysis of flow and heat transfer of high pressure turbine stages. CFD gives an insight to flow patterns that are difficult, expensive or impossible to study using traditional (experimental) techniques. However, the application of CFD depends on its accuracy and reliability. This requires the CFD code to be validated with laboratory measurements to ensure its predictive capacity.

Real flow in gas turbines is quite complex, being viscous, unsteady, and three-dimensional. The CFD programs offer different approaches to investigate such a flow field numerically, all of them uses multiple reference frames such as (i) The mixing plane model, (ii) the frozen rotor model and (iii) the sliding mesh model.[1]. The afore mentioned approaches differ in the way the interface between moving and non moving cell zones is treated. In the mixing plane approach, a steady state solution is calculated for each fluid zone and the two adjacent frames are coupled by exchange of flow field variables at the interface. Flow field data are averaged circumferentially for both frames at the interface and then be passed to the adjacent zone as boundary conditions. This spatially averaging technique at the interface removes any unsteadiness that would arise due to variations in the zone-to-zone flow field (e.g., wakes, bow waves, shock waves, flow seperation). The advantage of this approach when compared to the frozen rotor model is the less need for computational resources since due to averaging circumferentially, it is sufficient to model only one passage between two adjacent vanes/blades in each rotor/stator.

For the frozen rotor model, the coupling between the cell zones is done by the maintenance of absolute velocity in the global coordinate, i.e., the velocities are just switched between relative and absolute frames. Thus, one obtains a “snapshot” of the flow field at one fixed rotor position.Wakes between different cells zones are considered in some degree.

Therefore, the frozen rotor approach is chosen in the steady investigation in the presented work. Both the frozen rotor and the mixing plane models assume that the flow field is steady with the rotor/stator effects being accounted for and by approximate means. On the other hand, the sliding mesh model assumes that the flow field is unsteady and hence models the interaction with complete fidelity. Most often the unsteady solution that is sought in a sliding mesh simulation is time periodic. Note that since the sliding mesh model requires an unsteady numerical solution, it is computationally more demanding than the frozen rotor and mixing plane approaches. While the frozen rotor approach is chosen in the steady computations, the sliding mesh approach is used in the unsteady calculations. By comparing the steady results and the unsteady one, the ability and limits of the frozen rotor approach in predicting the optimum clocking positions in multistage turbomachinery are investigated. He [2] introduced a new unsteady approach called as Harmonic Approach. It is a non linear method which decomposes unsteady flow variables as time averaged variables and periodic unsteady perturbations. It is an averaged passage technique which estimates the unsteady flow field as a Fourier decomposition of the periodic fluctuations based on a preselected number of harmonics. Hircsh [3], in his study on Nonlinear Harmonic Method for Rotor-Stator Interactions applied to Thermally Perfect Gas, describes the extension of the nonlinear harmonic method to the solving of periodically disturbed thermally perfect gas flow in a turbomachinery code. Time-mean and fluctuating parts of the flow variables can be derived by linearizing about the time-mean state. The resulting formulas are assessed by the comparison with full unsteady method results on a realistic turbomachinery test case.

Numerical modeling of film holes adds complexity to the CFD simulation due to the requirement of accurate mesh which need to manage the small diameter film holes and comparatively larger blade chords. An unsteady analysis with film hole modeling is a big challenge in CFD. Hunter [4] in 1998 used source term to simulate the film on cooling airfoils. Sources of mass, momentum, energy and turbulence quantities needs to be specified in each cell adjacent to surface with film injection. Such an approach helps to minimize the film hole modeling complexity.

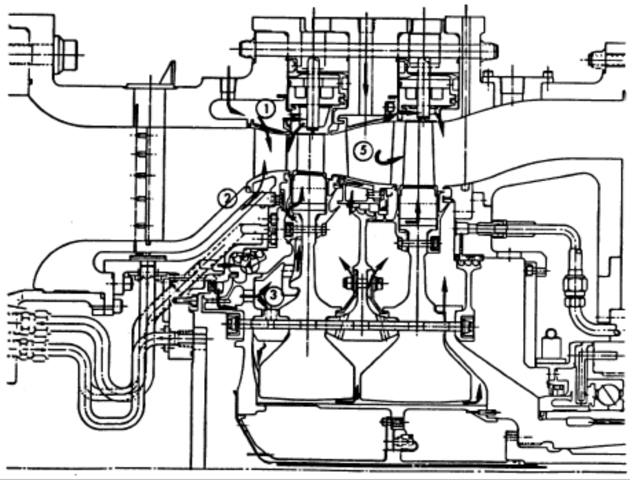
From the above discussion, it is clear that a systematic study which covers the effect of source term model and film hole model on turbine aerodynamic performance and also the effect of stage interface models such as mixing plane model, frozen rotor model and non linear harmonic model on turbine performance needs to be understood. This helps to properly predict the aerodynamic performance of a high pressure turbine. The main objectives of the present study are.

1. To establish guidelines to help the designer to successfully set-up and execute the suitable CFD model analysis with proper turbulence model study and mesh independent study.

2. To perform CFD validation with three different types of approaches to model the stage interfaces such as Mixing Plane approach, Frozen Rotor approach and Non-Linear Harmonic approach. The film holes on the blade surface, hub and shroud walls are modeled by using source term cooling and actual film hole modeling for all the analysis.

PROBLEM FORMULATION AND NUMERICAL METHOD

The validation of CFD predictions are carried out with the GE-EEE turbine component test performance report,[5]. This is a two-stage full scale warm air turbine test rig with simulated cooling flows used to verify the design features of General Electric Energy Efficient Engine which is designed for moderate loading. The blade profiles and film holes are modeled based on the details available in the NASA design report, NASA CR 167955 and performance test report, NASA-CR-168289 [5&6]. Figure 1 shows the schematic of the turbine rig with the cooling flow circuits. The tests are done at different operating conditions.



Cooling flow details (kg/sec)							
No.	Location	DP	TP-1	TP-4	TP-6	TP-8	TP-10
1	Aft vane	0.522	0.517	0.595	0.422	0.525	0.514
2	Farword vane	0.383	0.385	0.394	0.317	0.401	0.381
3	Blade 1 & 2	0.758	0.768	0.737	0.393	0.710	0.686
4	CDP	0.157	0.173	0.162	0.162	0.203	0.373
5	Stage2 Nozzle	0.263	0.267	0.281	0.158	0.261	0.248

Fig.1 Turbine rig with cooling circuits [5]

Figure 2 shows the performance map generated after the performed tests. Test point ‘0’ (design point) and test points 1, 4, 6, 8 and 10, (off-design points) are considered for the study. Majority of geometric details regarding turbine hub and shroud flow paths and film hole dimensions are extracted from the details available in[5, 6]. Coordinates of the Blade profile are available at 0%, 50% and 100% span for all rotors and stators. Linear interpolation is performed to obtain the full 3D blade.

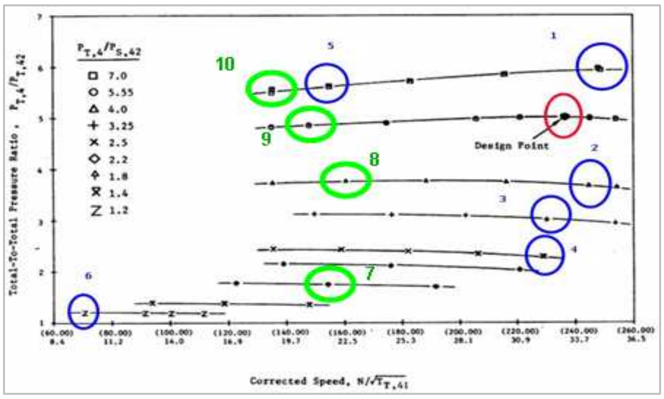


Fig.2 Turbine rig Performance curve [5]

The Film holes location details are not clearly indicated in the report. So, the location on blade surface is approximated based on the picture provided in the report. Fig 3 shows the film hole locations modeled for CFD simulation.

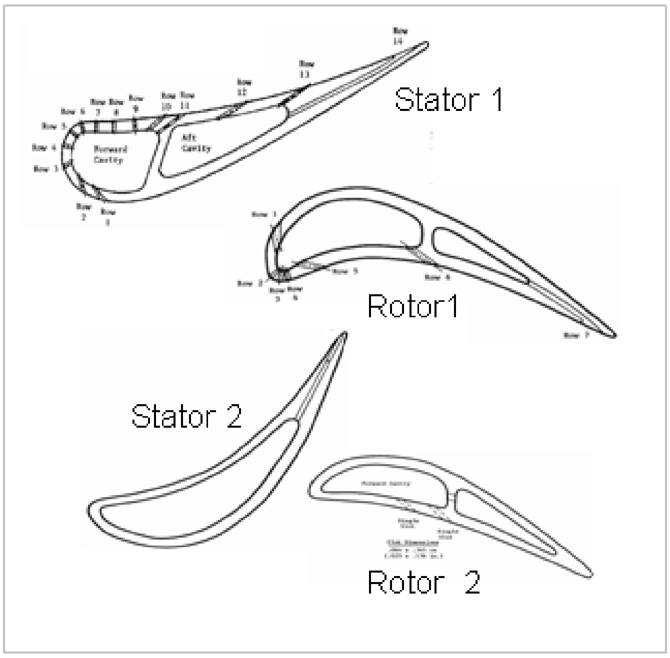


Fig.3 Schematic of film hole locations

Cooling mass flow at the entry of cooling hole is not known explicitly. So, a set of parametric CFD runs are performed to assess the amount of mass flow passing through each hole. This is done with different permutations and combinations of making the total massflow passing through the given cavity as constant. The first mass flow guess was taken based on design intent cooling mass flow information provided in GE-EEE design report [6].

Geometry creation and meshing is accomplished with AutoGrid5 Version 8.8-2. [7]. AutoGrid5 is a Numeca.Inc tool used for modeling and meshing turbo machine geometries. The reason for this tool selection is because of its powerful, automated and fully hexahedral grid generation technique used for turbomachinery. It has the unique capability to generate structured multi-block meshes with high grid quality and to use pre-defined topologies. It also has the ability to model and mesh film holes on the blade, hub and shroud walls and connect the same to the main skin block through a matching connection. The grid can be exported to a number of formats for different solvers. Figure 4 shows the geometry selected for the simulation.

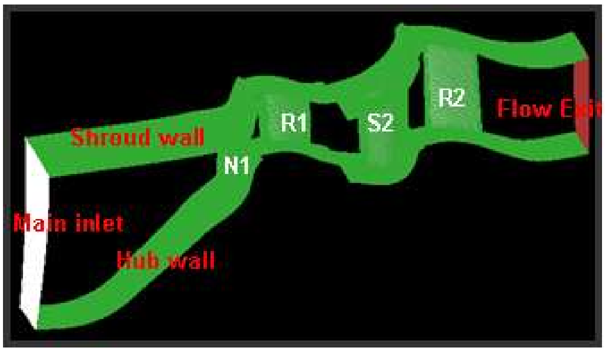


Fig.4 Geometry selected for simulations

The 3D computational domain for mixing plane approach includes a single blade/vane in the middle with periodic boundary conditions employed to simulate the rig test condition. The mesh was populated with the first cell distanced at 0.0005 [in] adjacent to the wall. Subsequent 17 layers with expansion ratio of 1.2 are maintained to form an O-Grid. A y+ close to 1 at first grid point is maintained around the blade. O4H topology available in AutoGrid5 is used to mesh the computation domain. The final mesh is composed with 5 blocks. A Skin block with O-grid topology around the blade surface to resolve the boundary layer and four H grid topology blocks for inlet, outlet, upper core and lower core. Table 1 shows the mesh grid point distribution for the computation domain.

Table 1 Mesh Grid point Distribution

Component	Mesh, I x J x K directions
Stator-1	171 x 97 x 55
Rotor-1	95 x 97 x 63
Stator-2	111 x 97x 47
Rotor-2	95 x 105 x 43
Film holes	17 x 33 x 30
Total	9145279

The film holes on blade surface, hub and shroud walls are modeled and meshed in Autogrid5. Table 1 also shows the mesh grid point distribution for the film holes. The mesh at the film hole and the blade surface are connected through a non matching grid interface. The details of the mesh generation technique are explained in the Autogrid5 user manual [7]. Figure 5 shows the mesh distribution for the two-stage high pressure turbine along with the mesh around the film holes. For the frozen rotor simulation, to match the pitch across the station interface, a domain scaling is done so that a total of two stators and three rotors are modeled.

BOUNDARY CONDITIONS

Two test points (design point and test point 10) as shown in figure 2 are taken for the analysis. The detailed

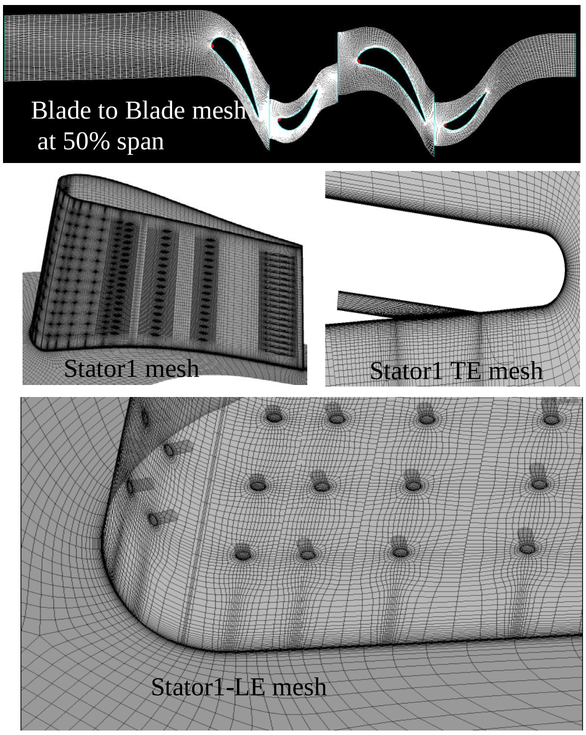


Fig.5 Mesh Details

operating conditions at these two points are given in table 2. The cooling flow is simulated by using source term cooling as well as actual film hole modeling. The source term approach is

Table 2 Operating conditions used for study

Case#	m (lb/sec)	RPM	PT_{in}, psia	TT_{in}, R
DP	24.04	8295	50.18	1276.87
TP-1	24.03	8885	50.19	1275.00
TP-4	23.34	8194	50.13	1276.78
TP-6	16.09	2534	50.18	1285.79
TP-8	24.12	5734	50.14	1276.15
TP-10	24.51	4827	50.84	1278.53

Similar to the approach described by Hunter et al [4] is used to simulate film cooling on cooled airfoils and the end walls. As per this approach, the sources of mass, momentum, energy and turbulence quantities are specified in each cell adjacent to a surface with film injection. Here a row of cooling hole is actually modeled as a slot. Several inputs are required to specify the source terms. These include the cooling mass flow, the geometry angles of hole centre line, the hole size, the coolant supply temperature, an approximate discharge static pressure, the turbulence intensity and the hydraulic diameter of the coolant hole pipe. Details of film hole boundary condition are given in table 3.

In film hole modeling, the details of mass flow rate, total temperature, flow direction, turbulence intensity and hydraulic diameter are provided at the inlet of film hole. The main flow boundary conditions for the CFD solution are:

Inlet:

- Total Pressure: Provided in table 2
- Total Temperature: Provided in table 2
- Turbulence Kinetic energy: 0.15 m^2/sec^2
- Turbulence dissipation rate: 3 m^2/sec^3
- Turbulence intensity: 5%

Walls:

- No-slip and adiabatic
- Rotor RPM: Provided in table 2

Exit:

- Static Pressure = constant value to match the pressure ratio.

Mesh sensitivity study is performed by systematically increasing the first cell distance, expansion ratio (1.1, 1.2 and 1.3), number of layers in the o-grid and the mesh size in the core region. Increase of mesh size in core region is done by increasing the number of layers in span wise direction and then by changing the B2B count. A final mesh with the mesh distribution shown in Table 1 is used for all further analysis. Three different turbulence models, i.e k-ε with enhanced wall treatment, sst-k-ω and S-A models are used to decide the most suitable model for the analyses. We could not notice any major difference in performance data with change in Turbulence model. With the fact that sst- k-ω model resolves the boundary layer in better way, we proceeded further with sst k-ω turbulence model.

Table 3 Boundary conditions at slot/film hole inlet

Blade	No. of Film hole rows	No. of holes on each Row	Hole Type	Flow angle (deg)	Massflow (lb/s)	Temp(R)
Stator1	Row1	22	Gill hole on SS	55	0.464073	587.513
	Row2	23	Gill hole on SS	59	0.464073	587.513
	Row3	12	Shower head hole	90	0.066139	587.513
	Row4	12	Shower head hole	90	0.016535	587.513
	Row5	12	Shower head hole	90	0.016535	587.513
	Row6	12	Shower head hole	90	0.016535	587.513
	Row7	12	Shower head hole	90	0.016535	587.513
	Row8	12	Shower head hole	90	0.016535	587.513
	Row9	12	Shower head hole	90	0.016535	587.513
	Row10	20	Gill hole on PS	35	0.066139	587.513
	Row11	19	Gill hole on PS	35	0.088185	587.513
	Row12	16	Gill hole on PS	35	0.088185	633.953
	Row13	16	Gill hole on PS	35	0.088185	633.953
	Row14	18	Trailing Edge slot	15	0.440924	633.953
Rotor1	Row1	25	Gill hole on SS	35	0.266539	621.514
	Row2	11	Shower head hole	90	0.072752	621.514
	Row3	10	Shower head hole	90	0.072752	621.514
	Row4	11	Shower head hole	90	0.072752	621.514
	Row5	22	Gill hole on PS	40	0.248902	621.514
	Row6	17	Gill hole on PS	40	0.213628	621.514
	Row7	11	Trailing Edge slot	15	0.464954	621.514
Stator2	Row1	10	Trailing Edge slot	10	0.268964	641.5671
Rotor2	Row1	40	Gill slots on PS	35	0.037919	621.514
	Row2	40	Gill slots on PS	35	0.037919	621.514

In the present CFD validation study, the comparison of CFD results obtained by using three different approaches to address the station interface such as (i) Mixing-Plane approach (ii) Frozen-Rotor approach and (iii) Non-Linear Harmonic approach is made. The accuracy of the CFD result is estimated based on the comparison of span wise variation of flow properties such as total pressure, total temperature and Mach number with respect to the test data[5]. 1D performance comparison is also made to understand the need of film hole modeling and the degree of mesh accuracy required for such performance studies. Cooling flows are addressed using two different modeling approaches such as (i) Source term modeling and (ii) Film hole geometry modeling. The comparison study is also made between these two approaches. For all the simulations two different commercial software i.e. CFX Version 12.1 from Ansys.Inc and Fine Turbo version 8.8-2 from Numeca.Inc are used. Details of CFD investigation are provided in table 4.

Table 4 List of CFD investigations performed

CFD Tool	Test point	Mixing plane analysis		Frozen Rotor Analysis		Harmonic Analysis	
		Source term	Film cooling	Source term	Film cooling	Source term	Film cooling
CFX	Design point	Yes	Yes	No	Yes	No	No
	Offdesign point	Yes	Yes	No	No	No	No
Fine-Turbo	Design point	Yes	Yes	No	Yes	No	Yes
	Offdesign point	Yes	Yes	No	No	No	Yes

RESULTS AND DISCUSSIONS

Each simulation is run using both the solvers for convergence. The confirmation of convergence is made based on (i) The root mean square (RMS) residuals for conservation of mass, momentum, energy and turbulence parameters smaller than 10e-4, (ii) Mass inbalance between the inlet and exit mass

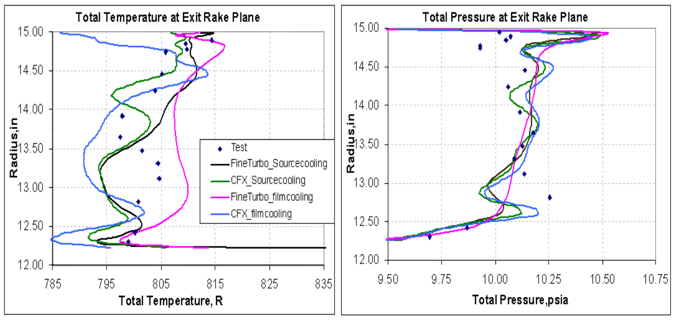


Fig.6 Comparison of flow property at exit plane

flow is smaller than 10e-4, (iii) variation of flow properties such as static pressure and Mach number at stator 1 TE base point is smaller than 10e-4. Figure 6 shows the comparison of total temperature and total pressure at the exit rake plane which is at a distance of 2 inch from the rotor 2 trailing edge. The CFD results are compared with the rig test data. This analysis is done with mixing plane

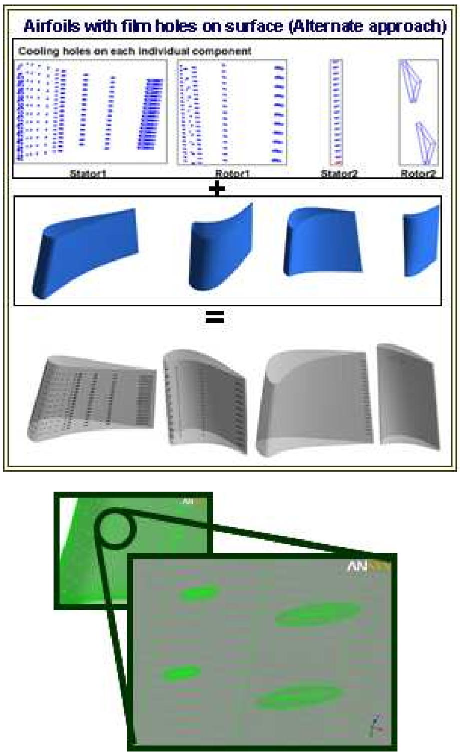


Fig.7 Alternate meshing approach

approach at the station interface. Both the CFD software captured the trend of flow distribution at the exit rake plane. It is also observed that the film hole model helps to properly solve the pressures and temperatures and hence helps to obtain a closer match with test data. Results with fine turbo are closer to the test data due to the reason of the Autogrid mesh compatability with the Fine turbo solver. Usage of CFX solver resulted in a robust mesh import process. The robustness is due to the fact that the imported FNMB patches represent as walls, and need to be manually defined as interfaces. The effort to select and convert all the 800 walls to interfaces is a time consuming process.

An alternate approach where the possible FNMB patches could be avoided is attempted. In this approach both the blade geometry and film holes are exported to CFX separately and a non-matching connection is defined at the film hole-skin blade interface. Details of this approach are shown in Figure 7. Both the default and modified (alternate) mesh approaches, in CFX, gives similar trend in span wise distribution of Pt and Tt. A reduction in the CFX problem set up time by 50% is observed in comparison with the default approach. The results obtained with this alternate approach are shown in Figure 8.

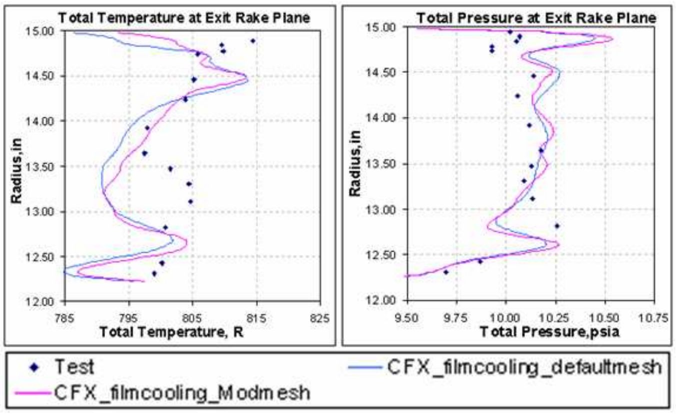


Fig.8 Total temperature and Total Pressure with Alternate approach

Turbine performance maps such as Corrected Mass Flow vs. Total Pressure Ratio and Cycle Efficiency vs. Total Pressure Ratio are made to understand the difference between source term modeling and film hole modeling on turbine performance. Figure 9 shows the performance map. To have a clear understanding on the variation in the performance, mass averaged values of turbine performance numbers is shown in Table 5. Less than 4 % difference in the turbine performance map is observed with source term modeling and film hole modeling for all the cases considered. How ever it is also observed that the difference of 10R is observed between the data and CFD for all the cases except test point 10. The higher difference in the total temperatures for test point 10 (~20R) is due to the fact that the point is operated at high PR and low RPM in comparison with the design point. Since the CFD solution at

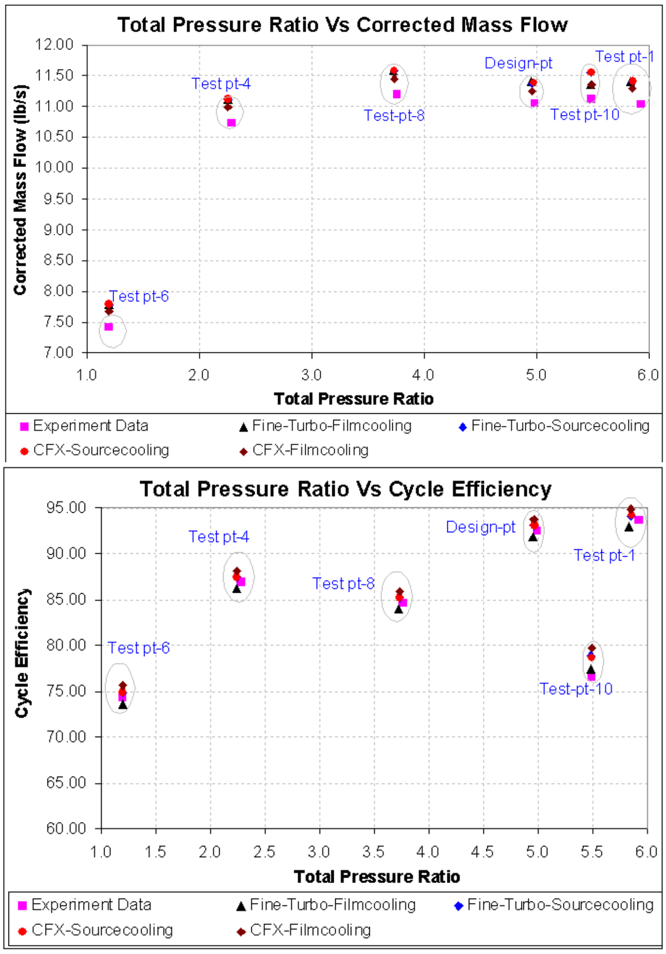


Fig.9 Turbine performance at design/off-design points

Table 5 Performance data

Temperature ratio across Turbine						
		Test data	Source Term		Film hole modeling	
Case#	RPM		Fine-Turbo	CFX	Fine-Turbo	CFX
DP	8295	1.59	1.59	1.60	1.58	1.60
TP-1	8885	1.66	1.66	1.67	1.65	1.66
TP-4	8194	1.31	1.31	1.32	1.30	1.32
TP-6	2534	1.12	1.12	1.13	1.11	1.12
TP-8	5734	1.44	1.44	1.45	1.43	1.45
TP-10	4827	1.46	1.49	1.49	1.48	1.50

Cycle efficiency						
			Source Term		Film hole modeling	
Case#	RPM	Test data	Fine-Turbo	CFX	Fine-Turbo	CFX
DP	8295	92.55	93.09	93.10	91.87	93.80
TP-1	8885	93.61	94.14	94.16	92.93	94.86
TP-4	8194	86.93	87.46	87.48	86.25	88.18
TP-6	2534	74.34	74.88	74.89	73.66	75.59
TP-8	5734	84.64	85.17	85.19	83.95	85.88
TP-10	4827	76.54	78.96	78.68	77.47	79.72

such high pressure ratio point is a challenge and has the maximum probability in the deterioration of the solution accuracy, this point is used along with the design point for further study with frozen rotor analyses and harmonic analyses.

The importance of the accurate film hole modeling arises when objective is to study the blade heat transfer behavior and the effect of flow and geometric modifications on the blade and end wall flow distributions. Figure 10 shows the surface temperature distribution for both film hole model and source term model. The difference in the surface temperature distribution between source term cooling and film hole modeling is observed. From the previous results it is seen that the impact of hole modeling on aerodynamic results is not observed. Due to the complexity in terms of effort and time for making a film hole, source term modeling is sufficient to predict aerodynamic performance.

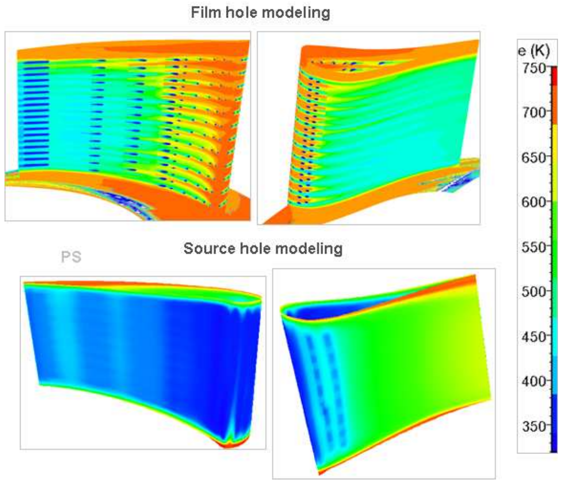


Fig.10 Total temperature contours on blade surface

All the above mentioned studies are made using mixing plane approach where the flow properties are averaged at the station interfaces. Further to understand the impact of frozen rotor analysis (where the flow property at each node of the interface is transported to adjacent cell) on turbine performance, a CFD simulation is made using CFX and Fine turbo.

For the frozen rotor analysis, domain scaling by changing the blade count from 46/76/48/70 to 48/72/48/72 is made. Two different clocking positions are employed which are differed by 7.5 degrees. Figure 11 shows the two different clocking positions considered. Figure 12 explains the importance of frozen rotor analysis to understand the wake penetration effects. Comparision of flow property at the exit rating station is shown in Figure 12 and its 1D performance map is shown in Figure 13.

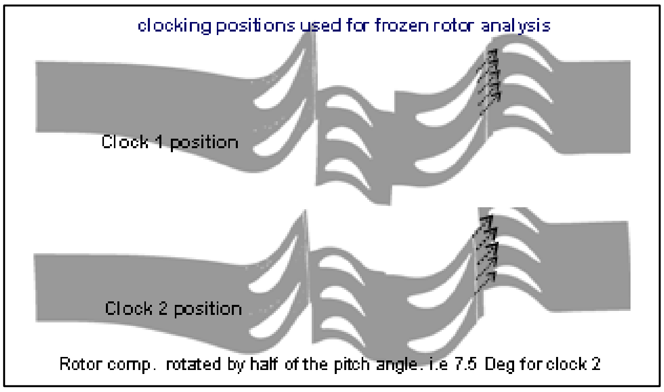


Fig.11 Clocking positions for frozen rotor analysis

(a) Mixing plane approach (b) Frozen rotor approach

Fig. 12 Wake propagation across interface

It is observed that no change in the turbine performance number has occurred with the frozen rotor approach. Comparision of Figure 12 and Figure 13 says that frozen rotor analysis helps to understand the intensity of adjacent blade wake penetration in to the subsequent blade flow.

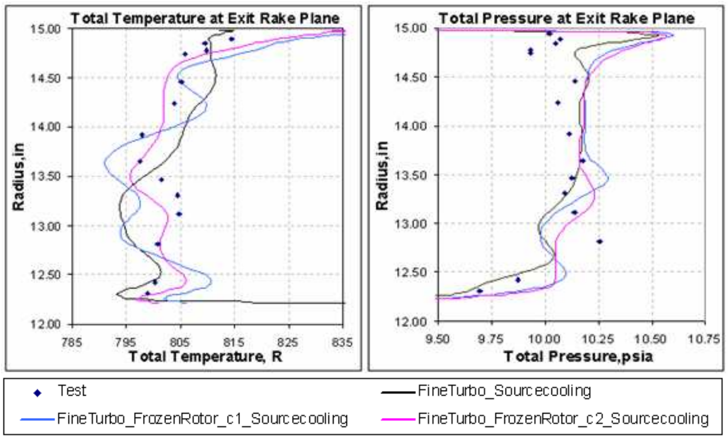


Fig.13 Total temperature and Total pressure with frozen rotor approach

From Figure 13 it is understood that an expected trend is maintained with a change in clocking position. The mixing plane result is inline with the frozen rotor results. Further

Harmonic analyses are solved and the performance data with Harmonic approach is compared with frozen rotor and mixing plane results. Harmonic analysis is explained by He [2] as a non linear method which decomposes unsteady flow variables as time averaged variables and periodic unsteady perturbations. It is an averaged passage technique which estimates unsteady

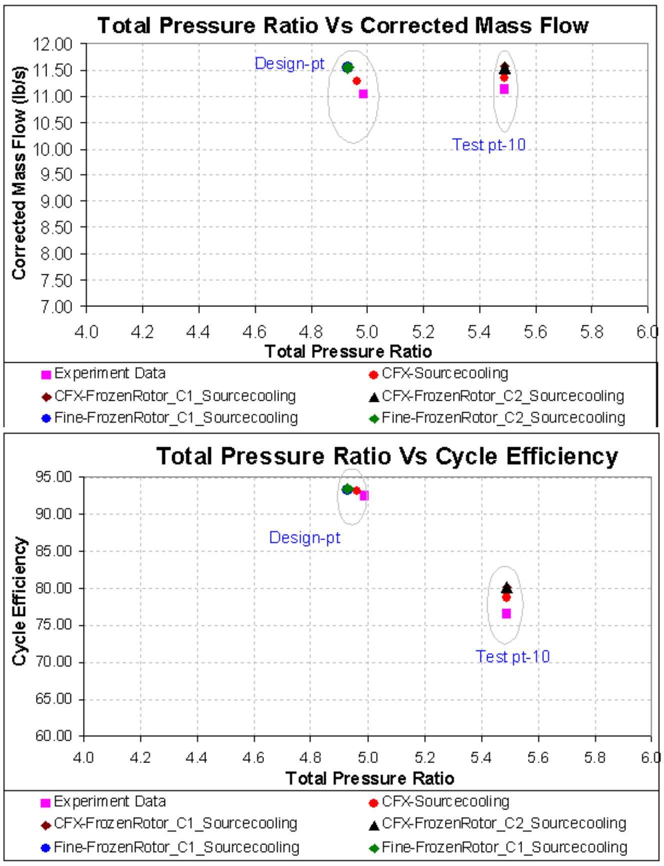


Fig.14 Performance with Frozen rotor and mixing plane approach

flow field as a Fourier decomposition of the periodic fluctuations, based on a pre-selected number of harmonics. It provides full unsteady flow field prediction for the selected Harmonics and also improves the flow connectivity across Rotor/Stator interfaces by reconstruction of Harmonics and time averaged flow on both sides of the interface. The detail harmonic analysis problem set up procedure includes the change in the time configuration to harmonic in flow model and providing the number of frequencies per perturbation to 3 and maximum number of perturbations per blade row to 2. Fine turbo user manual [8] provides the detail problem set up procedure and the theory of harmonic analysis. Figure 15 shows the total temperature and total pressure distribution at exit rake plane with harmonic analysis. In addition to the observed over prediction of temperature in the outer span region, which might be due to the approximation of inlet boundary condition as constant inlet as well as ignoring the blade heat transfer effects by considering the blade as adiabatic, an additional bias is observed with the harmonic analyses. This might be due to the usage of minimum number of pertubations (two) per blade row.

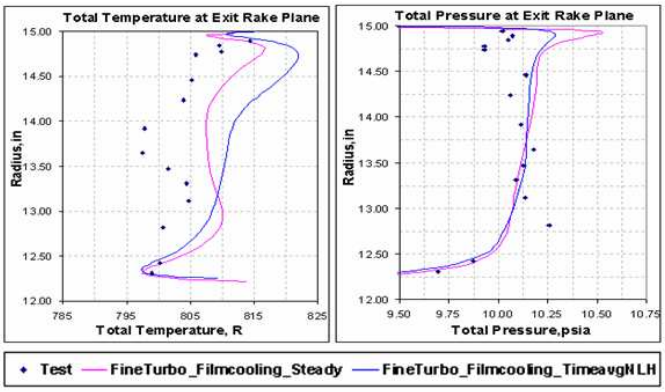


Fig.15 Total temperature and total pressure with Harmonic approach

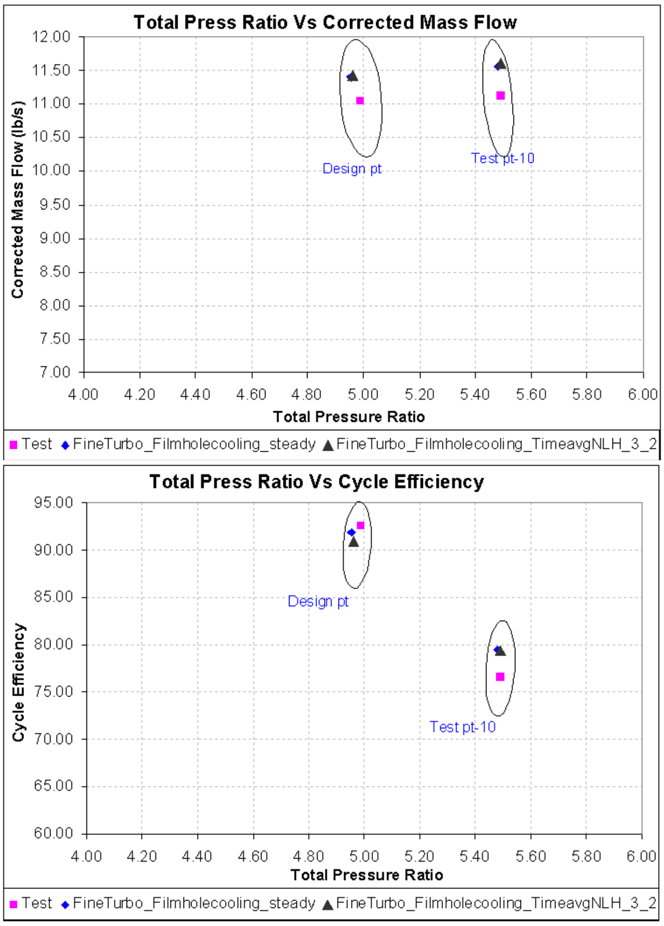


Fig.16 Performance map with harmonic approach

Figure 16 shows the performance map with harmonic analysis. No change in the performance map is observed with harmonic approach. The detail of steady state mixing plane approach with film hole model could reasonably capture the temperature spread in span direction than other methods. Mach number contours at 50% span from all the three modeling approaches studied is plotted in Figure 17.

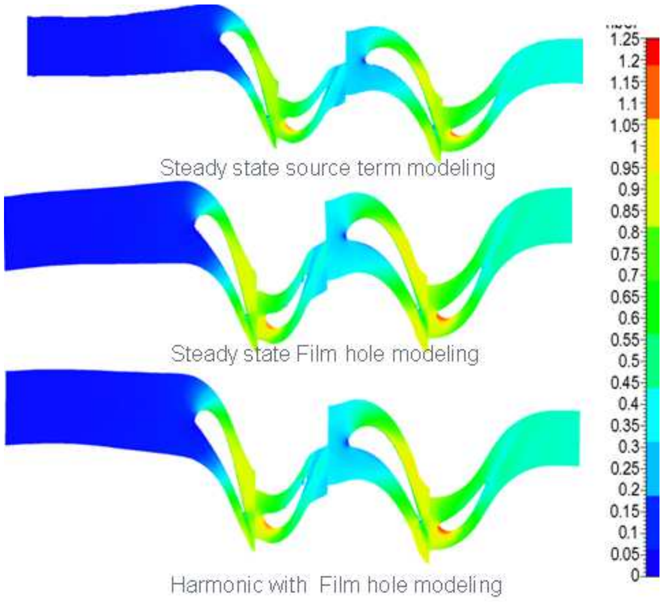


Fig.17 Mach Number Contours at 50% span

The important flow behavior is equally captured by all the methods. In addition to the comparision in terms of performance numbers between different cooling hole modeling approaches and different analysis types, a comparision in terms

Table 6 Software performance comparison

Process	Parameter	Fine Turbo		CFX		Comments
		Source	hole	Source	hole	
Mesh Generation in AG5	Size	3074936	9145274	3074936	9145274	Modeling of Discrete film holes (i.e shape,Angle,location) inAG5 is time consuming process
	Time	8 hours	24hours	8 hours	24hours	
Mesh importation in Solver	Time	2mins	15mins	45mins	3hours	Imporing CGNS files in CFX isTime Consuming Process
Problem Set up	Time	16 hours	24hours	24hours	160hours	In CFX defining the B.C patch forDiscrete film holes is time consumingprocess (because of large mesh and also interfaces in the fluid region)
Solver	No of Iterations	2000	2000	2000	300	Even the number of iterations for CFX-discrete cooling is low, the time taken per iteration is high, which ends in taking 52 hours to get a converged solution.
	Time	12 hours	14 hours	14hours	52hours	More Time for CFX-discrete cooling set up for a converged solution.

of software performance is also made. Table 6 shows the software performance in terms of flow model generation time and solution time. An alternate approach to minimize the film hole model import into CFX and the solution time delay due to the presence of many FNMB interfaces is suggested which helped to bring down the solver time by 80 percent .

CONCLUSIONS

A numerical study is carried out to validate the commercially available CFD codes for aerodynamic performance prediction of a two-stage high pressure turbine at design and off-design points. The validation is made between the three available approaches to model stage interface such as mixing plane approach, frozen rotor approach and harmonic approach. Cooling flow is modeled with source term and film hole model and the turbine performance is compared with both the approaches. GE-EEE turbine test data as per NASA-CR-168289 is used as the bench mark test data. Based on the present studies, the following conclusions can be derived.

- Maximum deviation of turbine performance numbers by 4% is observed between the CFD models and test data. This deviation is seen only for test point 10 which is considered as critical operating point (High PR and low RPM) and the deterioration in the accuracy of the CFD result is expected.

- Maximum difference of 2% in aerodynamic performance is observed between source term modeling and film hole modeling. Based on the effort in terms of time, space and labour, source term modeling approach can be recommended for performing aerodynamic analysis of HPT. Subsequently, film hole modeling is recommended for turbine heat transfer analysis.

- Maximum difference of 2% in the aerodynamic performance results is observed between mixing plane approach, frozen rotor approach and harmonic approach. Mixing plane approach is recommended for performing aerodynamic analysis based on its simplicity and resource consumption. How-ever this conclusion is made based on the limited amount of study made on GE-EEE HPT turbine which has an aspect ratio of 1.13.

- Definition of source term patches is easy with NUMECA software. The tool has the inbuilt capability.

- Importing AG5 mesh with film holes (approx. 9.14 million cells) in to CFX is a three hour process. The same in to Fine-Turbo is of 15 minutes. This is because of the compatibility between AG5 and Fine Turbo.

- An alternate methodology to handle the interface issue between the blade skin and hole exit in CFX is done. This minimized the time to model the problem in CFX by 80 percent.

- The unsteady effects and cooling flow modeling were not fully investigated in the scope of this paper. Therefore, a continuation of this project is currently undergoing by investigating the usage of large eddy simulation and time unsteady models.

- Mismatch of temperature profiles at the exit rake plane due to the constant inlet boundary condition is observed. To address this issue, certain CFD studies were being performed by extrapolating the turbine inlet temperature data based on the non-dimensional combustor exit temperature profiles (T/Tmax) available in public domain.

- Assumption of adiabatic airfoil walls is also a reason for the mismatch of total temperature data at the exit rake plane. A conjugate analyses considering the internal cooling passages as well as film cooling helps to avoid such mismatches. Such an analyses is expensive in terms of cost and time. Therefore an analysis with the combination of network theory and CFD is under investigation.

ACKNOWLEDGMENTS

The authors are thankful to the management of Honeywell Aerospace and Honeywell Technology Solutions for supporting this research and permitting the presentation of the results.

REFERENCES

[1] Dieter, Bohn., Sabine, Ausmeier., Jing, Ren., 2005, "Investigation of optimum clocking position in a two-stage axial turbine," Intl J of Rotating Machinery, Vol 3, pp. 202 – 210.
[2] L, He., W, Ning., 1998, "Efficient Approach for Analysis of Unsteady Viscous Flows in Turbomachines", AIAA Journal, Vol. 36, No. 11.
[3] Stéphane, Vilmin. Éric, Lorrain., Charles, Hirsch., 2007, "The Nonlinear Harmonic Method for Rotor-Stator Interactions Applied to Thermally Perfect Gas", Proceedings of the 8th International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows" ISAIF8-0066.
[4] S.D, Hunter., 1998, " Source term modeling of end Wall Cavity Flows Effects on Gaspath Aerodynamics in an Axial Flow Turbine", PhD Thesis, Universty of Cincinati. Department of Aerospace Engineering and Engineering Mechanics, November.
[5] L.P, Timko., 1990, "Energy Efficient Engine High Pressure Turbine Component Test Performance Report," NASA-CR-168289.
[6] Kalila, E.E., Lenahan, D.T., Thomas, T.T., 1982, "High Pressure Turbine Test hardware Detail design Report," NASA-CR-167955.
[7] NUMECA, User Manual AutoGrid5 Release 8.4 , NUMECA.inc., Belgium, January 2008.
[8] NUMECA, User Manual Fine Turbo Release 8.5, NUMECA.inc., Belgium, January 2008.