ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Forecast of Heat Transfer Using a Computational Fluid Dynamics Analysis (CFD)

K.Nooka Raju
Department of Mechanical Engineering
Nadimpalli Satyanarayan Raju Institute of Technology, Sontyam, Anandapuram, Vizag, A.P-531173, India.

Mr. P.V.S.Murali Krishna
Department of Mechanical Engineering
Nadimpalli Satyanarayan Raju Institute of Technology, Sontyam, Anandapuram, Vizag, A.P-531173, India.

ABSTRACT

The aim of this project was to the current state of heat transfer forecasting for commonly used CFd software. FINE/Turbo code is used for this software since it has a time accurate advantage. The computational mode utilized a conjugate heat-transfer solution are in the expected range of theoretical value, although the measurement program is still in process. Due to addition of cooling flow to the mainstream flow associated with a high- pressure turbine stage is difficult to model, especially when one is attempting to predict the surface heat-transfer rate. Boundary layer conditions and solid-fluid interactions dominate the region, making accurate computational predictions very difficult. Results of this project have identified areas for computational solutions. Lessons learned from the flat-plate measurement program will be applied to a full-scale rotating turbine stage in the near future, so understanding how to predict the local heat transfer using the CFD codes of major one.

THE RESEARCH PROGRAM AND BACKGROUND INFORMATION

1.1 Introduction to Turbine Cooling

CFD predictions of turbine aerodynamics have recently become quite accurate, allowing for a quicker and more robust design of jet engine turbines. Predicting heat transfer for film- cooled turbines, however, remains a difficult and arduous task, which continues to slow and hamper turbomachine development. Compounding this dilemma is the industry's ceaseless drive to increase engine efficiency, primarily accomplished by raising the inlet temperature of hot gasses to the turbine from the combustor. In many applications, inlet temperatures are at or above the melting point of the metal from which the turbine blades are constructed. Moreover, combustor exit non-uniformities such as turbulence and hot streaks can lead to unbalanced heat loads in the turbine, resulting in high levels of thermal stresses ultimately ending in blade failure

In order to avoid catastrophic thermal failure in the turbines, a variety of innovative techniques have been employed, including coating turbine airfoils with special thermal barriers and introducing a thin film of coolant air over the airfoils for protection from the devastating effects of hot combustion gasses . This addition of coolant air has become common practice in high-performance engines, but since the air is traditionally extracted from the compressor stage, a decrease in thermodynamic cycle efficiency results. Thus, it is advantageous to bleed only the optimal amount of air in order to maintain efficiency while still cooling the airfoils

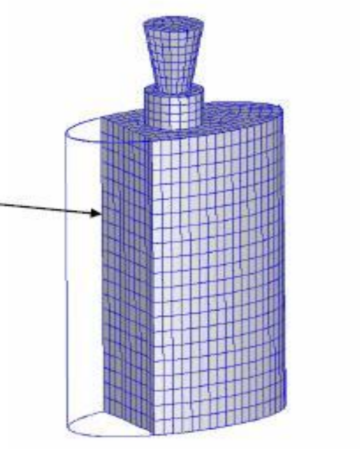
The sophisticated geometry and complex aerodynamic flows of turbine airfoils are frequently modeled in industry as flat plates to simplify calculations and reduce costs. Flat-plate models

- CFD analysis complements testing and experimentation.

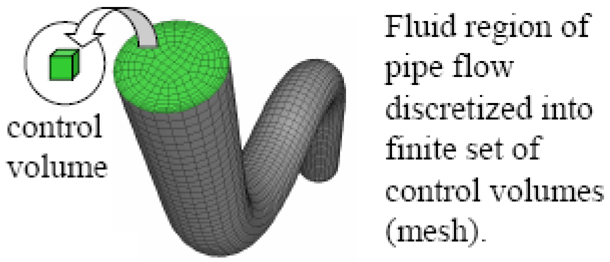
  – Reduces the total effort required in the laboratory.

Cite this article as: K.Nooka Raju & Mr. P.V.S.Murali Krishna, "Forecast of Heat Transfer Using a Computational Fluid Dynamics Analysis (CFD)", International Journal & Magazine of Engineering, Technology, Management and Research, Volume 5 Issue 8, 2018, Page 87-90.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

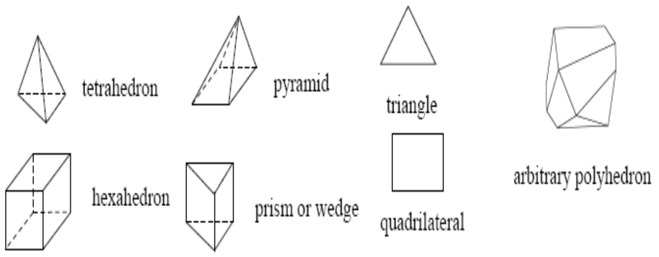


Mesh for bottle filling problem.



1.6 Design and create the grid

- Should you use a quad/hex grid, a tri/tet grid, a hybrid grid, or a non-conformal grid?
- What degree of grid resolution is required in each region of the domain?
- How many cells are required for the problem?
- Will you use adaption to add resolution?
- Do you have sufficient computer memory?



Small Calibration Facility

The CFD simulations of this project were designed to model the real-world experiments conducted in the Small Calibration Facility (SCF) at the OSU GTL. A short description of the SCF and its operation has been included to provide some background information.

The SCF operates as a medium-duration blow-down facility, which is used for both instrument calibration and as a small test facility [5]. In this experimental investigation, the SCF was utilized in its small test facility capacity. A photograph of the SCF and a schematic are included below as Figures 1 and 2.

Figure 1: Photograph of the Small Calibration Facility

3 below shows a picture of the test section, while Figure 4 shows a CAD model of the same section.

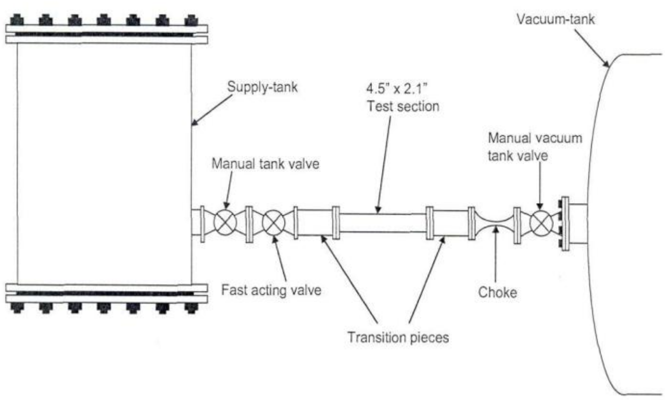


Figure 3: Test Section Containing Flat Plate

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

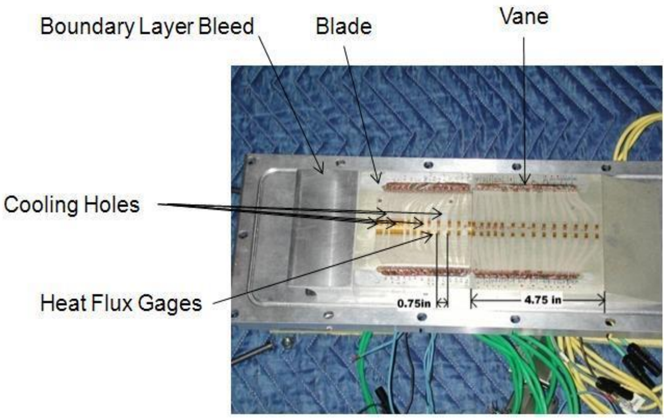


Figure 5: Instrumented Flat Plate

3.2 Boundary Conditions

The boundary conditions used in the CFD simulations were modeled after the flat plate experiments of the SCF. Table 2.1 summarizes the given boundary conditions, while Table 2.2 shows the boundary conditions that required further calculations using the given values.

Table 1: Given Test Section Boundary Conditions

Ma	Inlet Mach Number	0.4
T_s	Inlet Static Temperature	470 K
T_{cool}	Cooling Gas Temperature	240 K
P_s	Supply Tank Pressure	517.1 kPa
	Cooling Mass Flow	0.006 kg/s
k_{al}	6061-T6 Thermal Conductivity	177 W/mK

Table 2: Calculated Test Section Boundary Conditions

c	Speed of Sound	433 m/s
V	Flow Velocity	173 m/s
Po	Outlet Pressure	463.13 kPa
a	Air Mass Flow (theoretical)	3.66 kg/s

Table 4: Numerical Model Dimensions

	Length (m)	Width (m)	Height (m)
Flat Plate	0.289	0.114	0.0237
Air Flow	0.289	0.114	0.0536

Table 5: Multigrid Levels for 129 Points

Multigrid level	X Grid Points	Y Grid Points	Z Grid Points
1	2	2	2
2	3	3	3
3	5	5	5
4	9	9	9
5	17	17	17
6	33	33	33
7	65	65	65
8	129	129	129

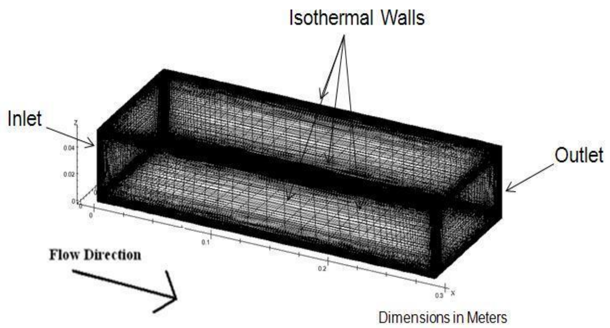


Figure 6: Single Block Wall Boundary Conditions

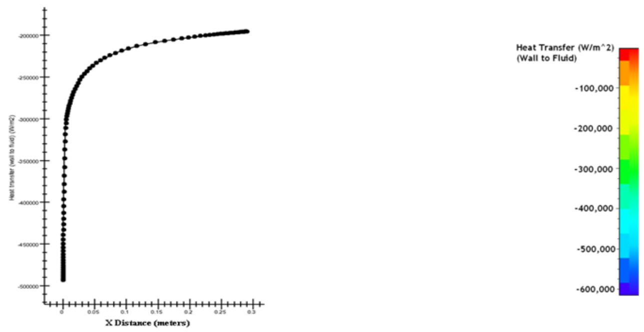


Figure 13: Uncooled Single Block Heat Transfer Graph

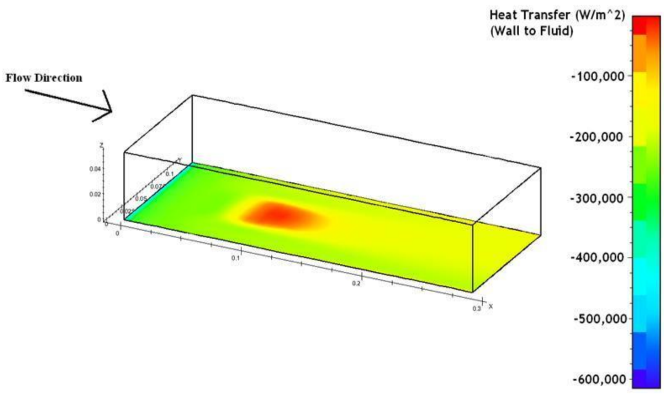


Figure 14: Cooled Single Block Heat Transfer

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

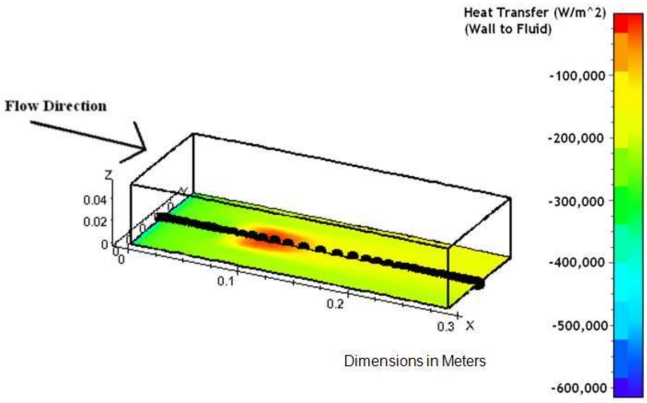


Figure 15: Cooled Single Block Gauge Location

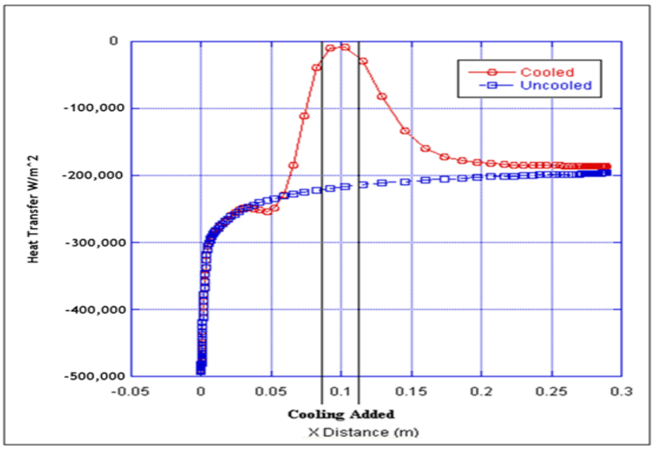


Figure 17: Comparison of Single Block Cooled and Uncooled Heat Transfer

Accurately solve a problem of this type. An uncooled solution that was obtained for the two block condition is shown below in Figure 18. A line of measurements is shown in Figure 19, and a plot of the heat transfer is included below as Figure 20.

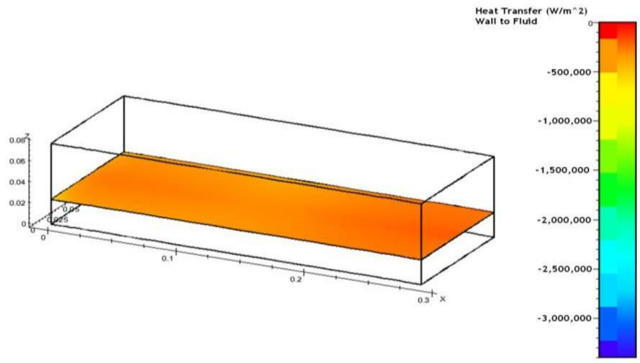


Figure 18: Uncooled Two Block Heat Transfer

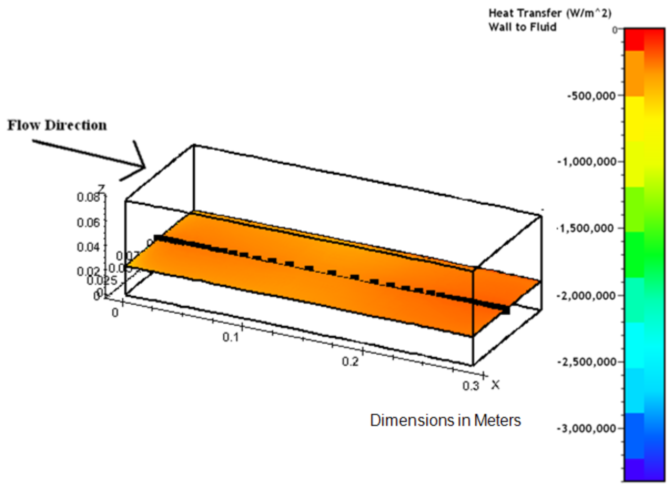


Figure 19: Uncooled Two Block Gauge Location

BIBLIOGRAPHY

[1] Dunn, M. G., 2001, “Convective Heat Transfer and Aerodynamics in Axial Flow Turbines,” ASME Journal of Turbomachinery, Vol. 123, pp. 637-686.

[2] Haldeman, C. W., Mathison, R. M., Dunn, M. G., Southworth, S., Harral, J. W., and Heitland, G., 2006, “Aerodynamic and Heat Flux Measurements in a Single Stage Fully Cooled Turbine- Part I: Experimental Approach,” ASME Paper No. GT2006-90966.

[3] Haldeman, C. W. and Dunn, M. G., 2003, “Heat Transfer Measurements and Predictions for the Vane and Blade of a Rotating High-Pressure Turbine Stage,” ASME Paper No. GT2003-38726.

[4] Varadarajan, K. and Bogard, D. G., 2004, “Effects of Hot Streaks on Adiabatic Effectiveness for a Film Cooled Turbine Vane,” ASME Paper No. GT2004-54016.

[5] Bernasconi, S. L., 2007, “Design, Instrumentation and Study on a New Test Section for Turbulence and Film Effectiveness Research in a Blowdown Facility”. M.S. Thesis, ETH Zurich