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)

Yerri Naidu Midatana
NSRIT, Sontyam,
Andhra Pradesh 531173, India.

B.Ramanjeneyulu
NSRIT, Sontyam,
Andhra Pradesh 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 model utilized a conjugate heat-transfer model for solid-fluid interactions with the turbine blade hardware. Current heat-transfer solutions are in the expected range of theoretical values, 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 which improvement in the current state-of-the-art are required, and have provided a benchmark 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 code is 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 [2]. 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 [3]. 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 [4]. 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 [5]. 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 [6]. Thus, it is advantageous to bleed only the optimal amount of air in order to maintain efficiency while still cooling the airfoils.

What is computational fluid dynamics?

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process.

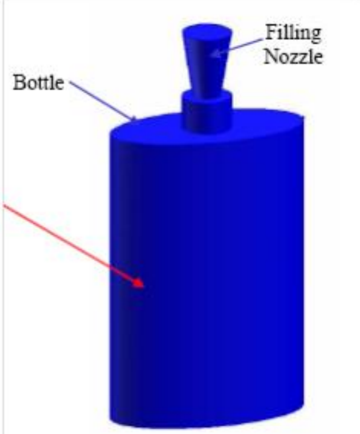
Cite this article as: Yerri Naidu Midatana & B.Ramanjeneyulu, " Forecast of Heat Transfer Using a Computational Fluid Dynamics Analysis (CFD)", International Journal & Magazine of Engineering, Technology, Management and Research, Volume 5, Issue 12, 2018, Page 15-20.

❖ The result of CFD analyses is relevant engineering data used in:
– Conceptual studies of new designs.
– Detailed product development.
– Troubleshooting.
– Redesign.

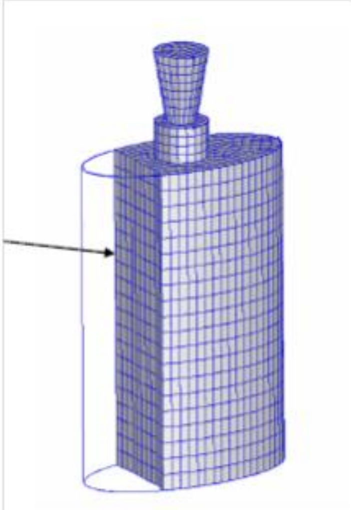
❖ CFD analysis complements testing and experimentation.
– Reduces the total effort required in the laboratory.

1.4 CFD - how it works

- Analysis begins with a mathematical model of a physical problem.
- Conservation of matter, momentum, and energy must be satisfied throughout the region of interest.
- Fluid properties are modeled empirically.
- Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, in viscid, two-dimensional).
- Provide appropriate initial and boundary conditions for the problem.
- CFD applies numerical methods (called discretization) to develop approximations of the governing equations of fluid mechanics in the fluid region of interest.



Domain for bottle filling problem.



Mesh for bottle filling problem.

❖ Governing differential equations: algebraic.
– The collection of cells is called the grid.
– The set of algebraic equations are solved

❖ Numerically (on a computer) for the flow field variables at each node or cell.
– System of equations are solved simultaneously to provide solution.

The solution is post-processed to extract quantities of interest (e.g. lift, drag, torque,
heat transfer, separation, pressure loss, etc.

1.5 Discretization:

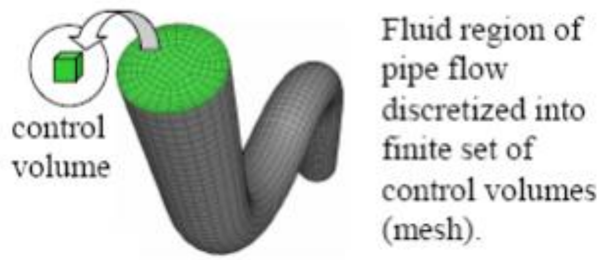
Domain is discretized into a finite set of control volumes or cells. The discretized domain is called the "grid" or the "mesh."

• General conservation (transport) equations for mass, momentum, energy, etc., are discretized into algebraic equations.
• All equations are solved to render flow field.

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

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi dV}_{\text{unsteady}} + \underbrace{\oint_A \rho\phi \mathbf{V}\cdot d\mathbf{A}}_{\text{convection}} = \underbrace{\oint_A \Gamma\nabla\phi\cdot d\mathbf{A}}_{\text{diffusion}} + \underbrace{\int_V S_\phi dV}_{\text{generation}}$$

Eqn.	ϕ
continuity	1
x-mom.	u
y-mom.	v
energy	h



In this study the both the single phase models and multiphase models are used for solving the respective category problems. This model will calculate one transport equation for the momentum and one for continuity for each phase, and then energy equations are solved To study the thermal behaviour of the system.

2.1 SINGLE PHASE MODELING EQUATIONS

The single phase model equations include the equation of continuity, momentum equation and energy equation (ANSYS Fluent 12.0). The continuity and momentum equations are used to calculate velocity vector. The energy equation is used to calculate temperature distribution and wall heat transfer coefficient. The equation for conservation of mass, or continuity equation, can be written as follows

2.1.1 Mass Conservation Equation

The equation for conservation of mass, or continuity equation, can be written as follows

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{\upsilon}) = S_m$$

Equation (3.1) is the general form of the mass conservation equation, and is valid for both incompressible compressible flows. The source m S is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources.

2.1.2 Momentum Conservation Equation

Conservation of momentum in an inertial (non-accelerating) reference frame is described By

$$\frac{\partial}{\partial t}(\rho\vec{\upsilon}) + \nabla.(\rho\vec{\upsilon}\vec{\upsilon}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F}$$

Where p is the static pressure, $\bar{\bar{\tau}}$ is the stress tensor (described below), and $\rho\vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively. $\vec{F}$ also contains other model dependent source terms such as porous-media and user-defined sources.

The stress tensor $\bar{\bar{\tau}}$ is given by

$$\bar{\bar{\tau}} = \mu\left[(\nabla\vec{\upsilon} + \nabla\vec{\upsilon}^{T}) - \frac{2}{3}\nabla.\vec{\upsilon}I\right]$$

Where μ is the molecular viscosity, I is the unit tensor, and the second term on the right hand side is the effect of volume dilation

2.1.3 Energy equation:

ANSYS FLUENT solves the energy equation in the following form:

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

$$\frac{\partial}{\partial t}(\rho E)+\nabla.(\vec{v}(\rho E+p))=\nabla\left(\kappa_{eff}\nabla T-\sum_j h_j\vec{J_j}+(\bar{\bar{\tau}}_{eff}.\vec{v})\right)+S_h$$

Where κ_{eff} is the effective conductivity ($\kappa+\kappa_t$, where κ_t is the turbulent thermal conductivity, defined according to the turbulence model being used), and $\vec{J}_j$ is the diffusion flux of species J . The first three terms on the right-hand side of Equation represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. S_h includes the heat of chemical reaction, and any other volumetric heat source.

$$E=h-\frac{p}{\rho}+\frac{\upsilon^2}{2}$$

Where sensible enthalpy h is defined for ideal gases as

$$h=\sum_j Y_j h_j$$

Y is the mass fraction of species j.

$$h_j=\int_{T_{ref}}^{T} c_{p,j}\,dT$$

T_{ref} is used as 298.15 K.

2.2.1.4 Energy Equation

The energy equation, also shared among the phases, is shown below

$$\frac{\partial}{\partial t}(\rho E)+\nabla.(\vec{v}(\rho E+p))=\nabla.(k_{eff}\nabla T)+S_h$$

The VOF model treats energy, E , and temperature, T , as mass-averaged variables,

$$E=\frac{\sum_{q=1}^{n}\alpha_q\rho_q E_q}{\sum_{q=1}^{n}\alpha_q\rho_q}$$

Where q E for each phase is based on the specific heat of that phase and the shared temperature. The properties ρ and eff k (effective thermal conductivity) are shared by the phases. The source term, h S , contains contributions from radiation, as well as any other volumetric heat sources.

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	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

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

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 6: Cooling Hole Location and Geometry

Row	Number of Holes	Distance from Leading Edge(mm)	Pitch (mm)	Diameter (mm)
1	16	86.36	2.438	0.457
2	15	88.27	2.540	0.457
3	33	91.64	1.219	0.305
4	32	108.03	1.143	0.406
5	17	113.13	0.965	0.356

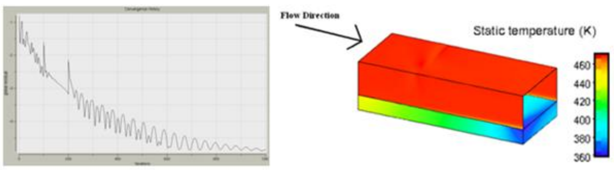


Figure 8: Flow Convergence at 800 Iterations

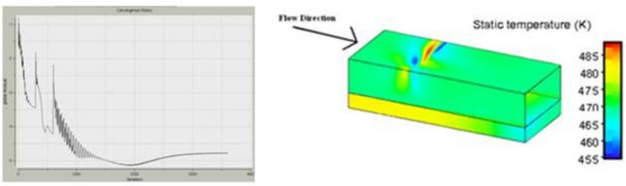


Figure 9: Flow Convergence at 3600 Iterations

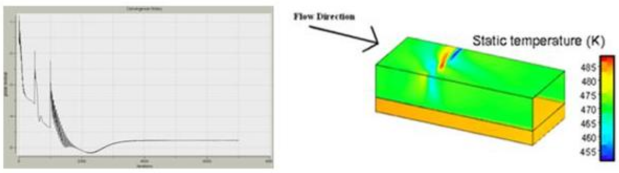


Figure 10: Flow Convergence at 9000 Iterations

The solutions of Figures 8 through 10 above also provided an important lesson on the use of fixed point clustering. In the mesh that was solved to create the CFD solutions above, a fixed point was added about 1/3 of the way downstream of the leading edge near the cooling holes. The fixed point was then gridded with a fine resolution, in order to better capture the cooling flow in the model.

However, it was quickly discovered that the addition of this fixed point was causing strange behavior in the computation model. Even without cooling, unusual temperature spikes were predicted downstream of the leading edge. Figures 8 through 10 above show a model without cooling flow, yet there are temperature spikes at the fixed points. After much experimentation, the use of fixed points was deemed unnecessary and was abandoned in subsequent computation models.

RESULTS AND DISCUSSION

5.1 Single Block Results

Although the single block CFD mesh was relatively simple in nature, it was still able to provide a useful prediction of flat plate heat transfer. Figure 11 shows a color map of the heat transfer for the single block case. Relevant boundary conditions from Tables 1 and 2 are repeated in Table 7 below for ease of reference.

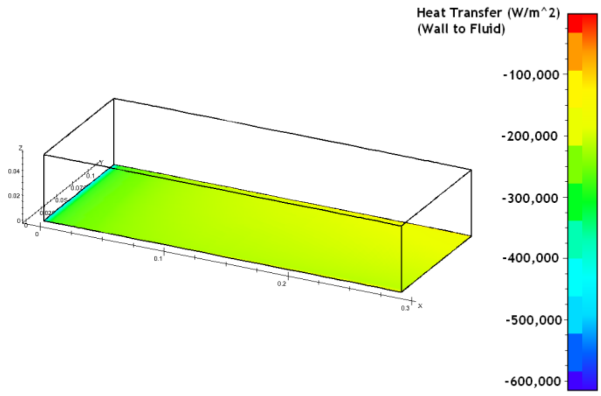


Figure 11: Uncooled Single Block Heat Transfer

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.

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

[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.