International Journal of Applied Engineering Research
ISSN 0973-4562 Volume 10, Number 6 (2015) pp. 15389-15395
© Research India Publications
http://www.ripublication.com

Computational Fluid Flow And Heat Transfer Analysis

N. Ilangovan

Department of Aeronautical Engineering,
Sathyabama University, Chennai – 600 119

ABSTRACT

Impinging jets have received considerable attention due to their inherent characteristics of high rates of heat transfer. Such impinging flow devices allow for short flow paths and relatively high rates of cooling from comparatively small surface area. Various industrial processes involving high heat transfer rates use impinging jets. Few industrial processes which employ impinging jets are drying of food products, textiles, films and papers, processing of some metals and glass, cooling of gas turbine blades and outer wall of the combustion chamber, cooling of electronic equipments etc. Heat transfer rates in case of impinging jets are influenced by various parameters like Reynolds number, jet to plate spacing, radial distance from stagnation point, prandtl number, target plate inclination, confinement of the jet, nozzle geometry, curvature of target plate, roughness of the target plate and turbulence intensity at the nozzle exit

Key words: Heat transfer, Jet impinging, Nozzle, Voxel conversion

INTRODUCTION

The aim is to reduce the heat due to the continuous usage of the plate. This paper deals with the heat transfer in the heated plate using different type of nozzles using the jet impinging method. Jet impinging is used to cool down the heated area. Jet impinging is used in many ways in aircraft they are used in turbine blade cooling because more heat is produced in turbine due to continuous running [1]. Here jet impinging is used to cool down the heated plate using different type of nozzles like cone, bell, multipoint, circular shaped using this nozzle shapes we will find the heat transfer to an impinging air jet as shown in figure 1.

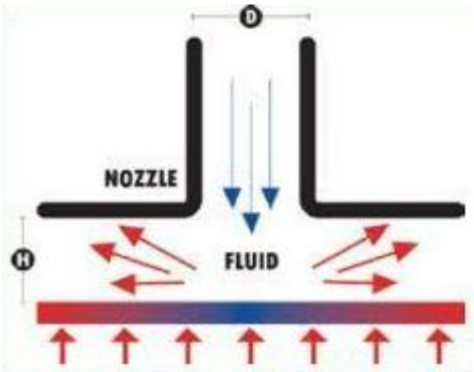


Figure 1 Traditional Jet impinging Design

The time averaged heat transfer distributions are qualitatively compared to velocity flow fields. Simultaneous velocity and heat flux measurements are reported at various locations on the impingement surface as shown in figure 2.

PROBLEM DESCRIPTION

Various diameters and boundary layer conditions were applied. Different nozzle shapes were considered. The jet flow speed is considered as subsonic. Grid generation or meshing techniques were applied [2].

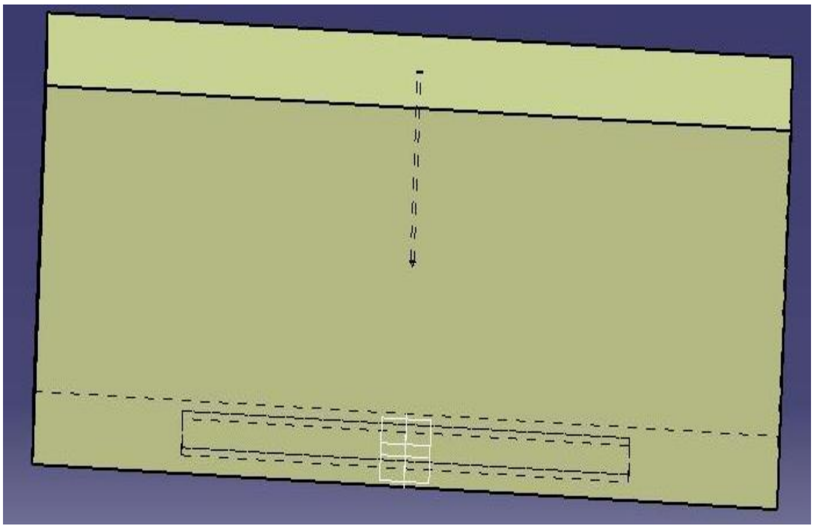


Figure 2 Jet Impinging Model

JET IMPINGEMENT HEAT TRANSFER

Efficient use of the fluid and high transfer rate is better than the other heat or mass transfer arrangements. The magnitude of the flow is smaller than the free wall parallel flow. Multiple jets may be used for larger surfaces [3].

IMPINGING JET REGIONS

Depending upon the upstream flow, the jet emerges from the nozzle with a velocity and temperature profile varies. The flow develops in to parabolic velocity profile in the pipe shaped nozzle with moderate amount of turbulence in upstream. Fairly flat velocity profile, less turbulence, and a down stream flow contraction are possible by the use of thin flat orifice.

A round jet with an axisymmetric profile or a long thin jet with two dimensional flow profiles is the typical jet nozzles in use. In the free submerged jet, the jet mass flow is raised by pulling additional fluid along with the jet by which lateral momentum transfer takes place outward. Energy loss of the jet and widening of the velocity profile are happening in this process. Even though there is a drop in velocity and pressure shearing layer remains unaffected and forms a core region with a higher total pressure. The expansion of the shearing layer towards the centre of the jet before reaching the target the formation of core decay region takes place. As the jet spreads radially, the axial velocity component in the central core decreases. The cylindrical nozzle requires simple hand book equations to calculate the pressure losses whereas the orifice plate requires more specialized equations and data [4].

The multiple nozzles give some improvement in efficiency and transfer properties uniformity. The numerical modelling shows that majority of the turbulence in the flow is generated by the shear layer in the wall jet upper portion. This turbulence region spreads in normal direction to the wall. Secondary peak location coincide highest peak kinetic energy location near to the wall [5]. The unsteadiness because of the outer region turbulence in the thermal boundary layer will be outside of the stagnation region as shown in figure 3. The degree of jet interaction is determined by the pitch jet or centre to centre positioning of jets in an array.

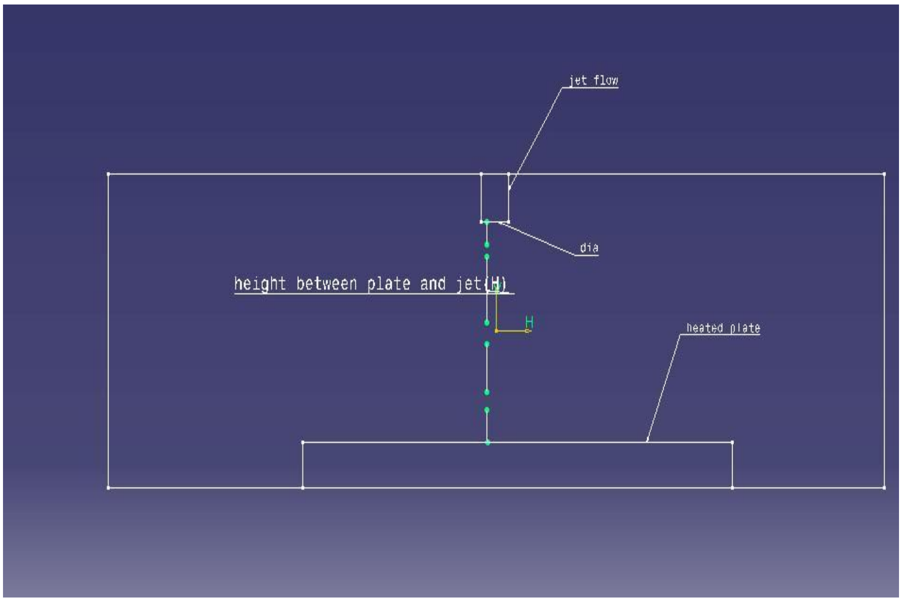


Figure 3 Position display

MESHING

Image-based meshing is the automated process of creating computer models for computational fluid dynamics (CFD) and finite element analysis (FEA) from 3D image data (such as magnetic resonance imaging (MRI), computed tomography (CT) or micro tomography. Although a wide range of mesh generation techniques are currently available, these were usually developed to generate models from computer-aided design (CAD), and therefore have difficulties meshing from 3D imaging data as shown in figure 4.

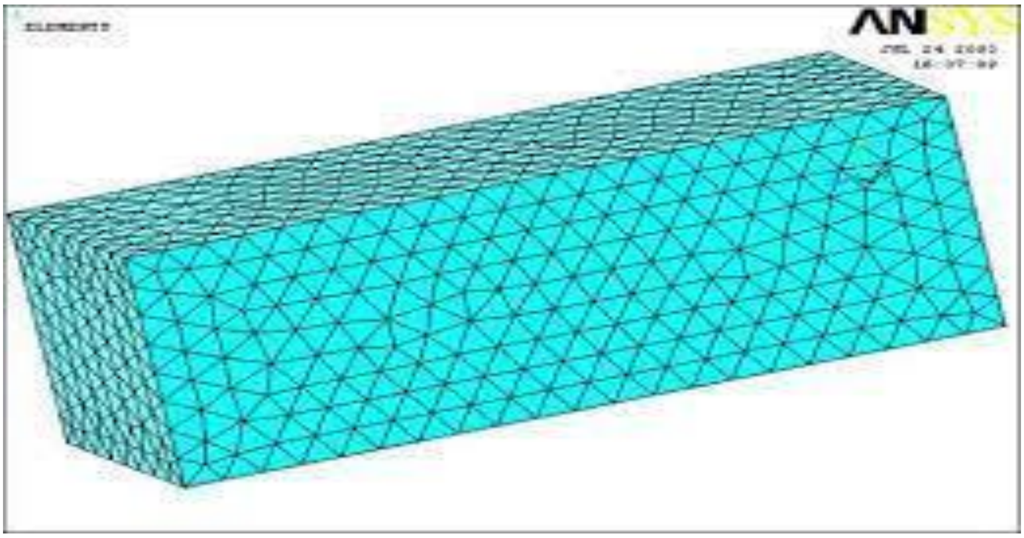
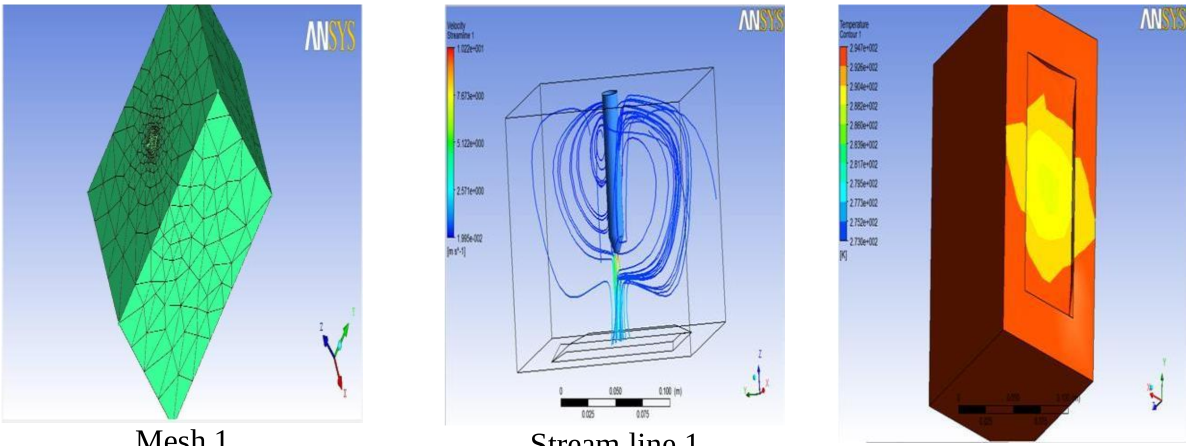


Figure 4 Meshing Model

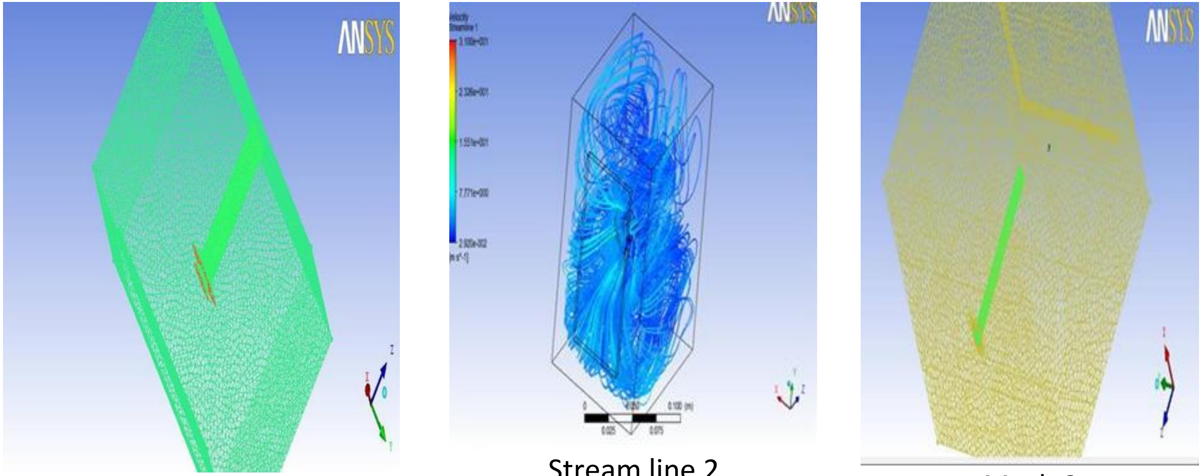
RESULT AND DISCUSSION

Meshing from 3D imaging data presents a number of challenges but also unique opportunities for presenting a more realistic and accurate geometrical description of the computational domain. There are generally two ways of meshing from 3D imaging data.

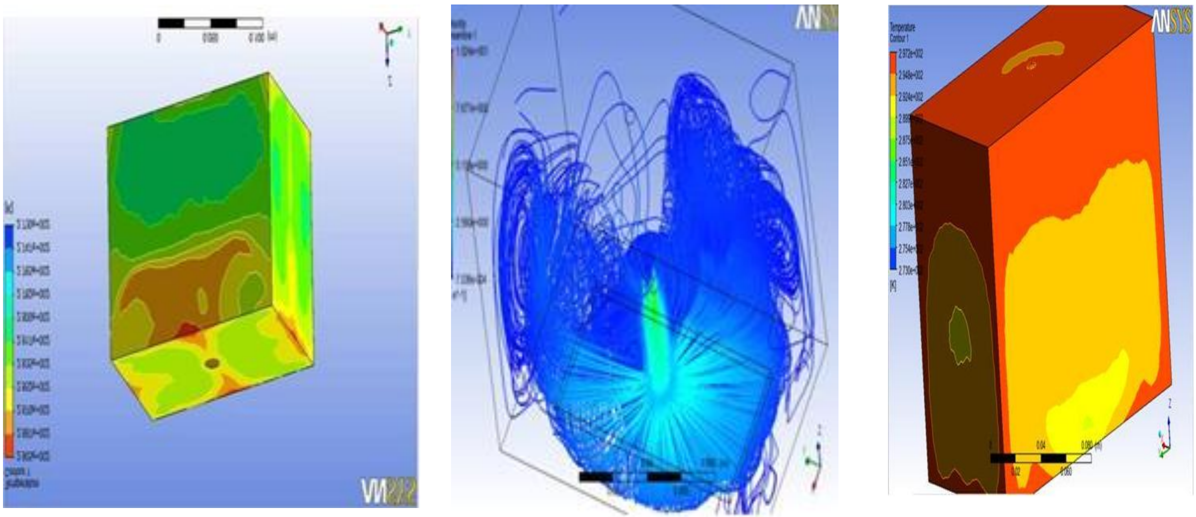
The majority of approaches used to date still follow the traditional CAD route by using an intermediary step of surface reconstruction which is then followed by a traditional CAD-based meshing algorithm. CAD-based approaches use the scan data to define the surface of the domain and then create elements within this defined boundary. Although reasonably robust algorithms are now available, these techniques are often time consuming, and virtually intractable for the complex topologies typical of image data. They also do not easily allow for more than one domain to be meshed, as multiple surfaces are often non-conforming with gaps or overlaps at interfaces where one or more structures meet as shown in figure 5.



Mesh 1 Stream line 1 Temperature 1



Mesh 2 Stream line 2 Mesh 3



Temperature 2 Stream line 3 Temperature 3

Figure 3 Different cone shaped nozzle

This approach is the more direct way as it combines the geometric detection and mesh creation stages in one process which offers a more robust and accurate result than meshing from surface data. Voxel conversion technique providing meshes with brick elements and with tetrahedral elements have been proposed. Another approach generates 3D hexahedral or tetrahedral elements throughout the volume of the domain, thus creating the mesh directly with conforming multipart surfaces as shown in figure 3.

The steps involved in the generation of models based on 3D imaging data are:

An extensive range of image processing tools can be used to generate highly accurate models based on data from 3D imaging modalities, e.g. MRI, CT, MicroCT (XMT), and Ultrasound. Features of particular interest include: Segmentation tools (e.g. thresholding, flood fill, level set methods, etc.) Filters and smoothing tools (e.g. volume- and topology-preserving smoothing and noise reduction/artefact removing).

The image-based meshing technique allows the straightforward generation of meshes out of segmented 3D data. Features of particular interest include:multi-part meshing (mesh any number of structures simultaneously). Mapping functions to apply material properties based on signal strength (e.g. Young modulus to Hounsfield scale Smoothing of meshes (e.g. topological preservation of data to ensure preservation of connectivity, and volume neutral smoothing to prevent shrinkage of convex hulls) Export to FEA and CFD codes for analysis (e.g. node sets, shell elements, material properties, contact surfaces)

CONCLUSION

The tasks which are important for the improvement of design and performance of impinging jets are revealed by the review of research publications of impinging jets.

The domination of physical mechanism under various conditions like different Reynolds number have been achieved by resolving the mechanism. The SST and V2-f models give good results with minimum computation time. The properties of the turbulent flow with the wall, jet spreading and the effect in the stagnation region are to be predicted by the improvement in the turbulence model. Anisotropic flow field simulation of a hybrid turbulence model will fulfill this requirement for higher efficiencies, the nozzle and installation profiles are to be modified. This can be achieved by set flow or blower power NU profile improvement. Work on several nozzles like cross shaped nozzle, coaxial nozzle and tap nozzle are representing sample of the practical possibilities.

REFERENCE

[1] Donaldson, C. D. and Snedeker, R. S. (1971). A study of free jet impingement. Part 1. Mean properties of free and impinging jets,. J. Fluid Mech. 45, 281–319

[2] Ferrari, J., Lior, N., and Slycke, J. (2003). An evaluation of gas quenching of steel rings by multiple-jet impingement. J. Mater. Process. Technol. 136, 190–201.

[3] Jambunathan, K., Lai, E., Moss, M. A., and Button, B. L. (1992) review of heat transfer data for single circular jet impingement. Int. Heat Fluid Flow 13, 106–115.

[4] Martin, H. (1977). Heat and mass transfer between impinging gas jets and solid surfaces. Adv. Heat Transfer 13, 1–60.

[5] Viskanta, R. (1993). Heat transfer to impinging isothermal gas and flame jets. Exp. Thermal Fluid Sci. 6, 111–134.