Flow visualization on V-gutter and teethed V-gutter flame holders

Bhaskar S, (BE Aeronautical), Tagore Engg. College, Affl. to Anna Univ., Chennai 600127(T.N)

Abstract**—** Flame holders are studied with various contours and their effectiveness is understood through the flow separation characteristics and recirculation points. The simulations are made on teethed V gutter and conventional one through OpenFoam and studied. The objective of the project is flow visualization of the 3D model and comparison of the 2D contour between the specimen.

Keywords: *Flow visualization, Water channel, Vortices, reattachment, Jet flow*

I. INTRODUCTION

Flame holding and flame stabilization can be achieved by organizing of the recirculation area where the fuel and air can be mixed partially at low velocities. The saw tooth *V-gutter* is shaped like a V with the point in the direction facing the flow of air with saw tooth serrations along the periphery. The distance between flow separation and reattachment is considered as wake region from which the effectiveness of flame holders are determined.

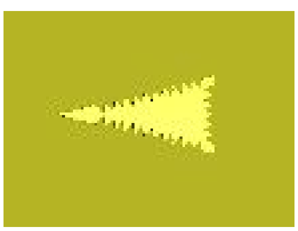

Figure 1: Saw tooth V-gutter, V-gutter

A. Geometry

The geometry of the V-gutter was created using the software *Catia v5.0.* The total length of the geometry is 0.16 m from front to end and width is 0.08 m. The angle of cone is 28^{o}. The body is placed in a domain which is 5L-2L (length-width) where L is the length of the flame holder(0.16m).

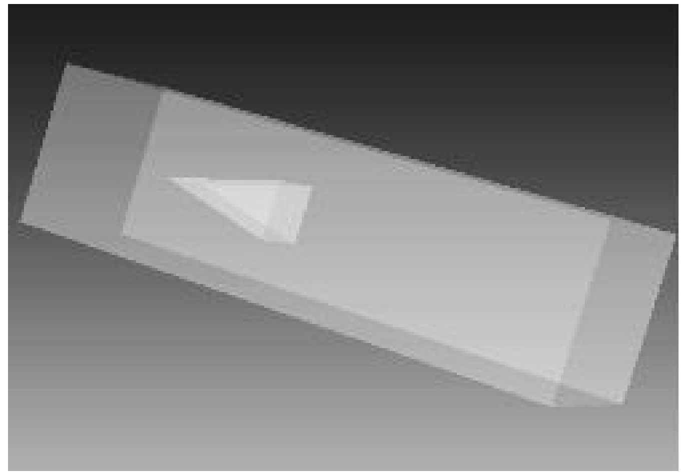

Figure 2: Computational domain

B. Meshing

The meshing for this simulation was done using the software *Ansys Fluent Workbench 15.0* with element size of 0.001m.

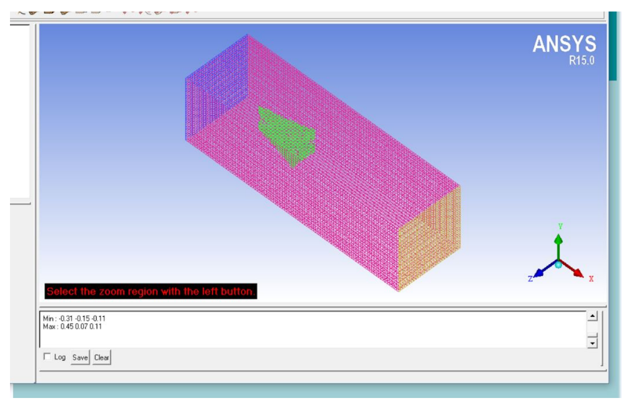

Figure 3: Mesh statistics of V-gutter

II. ANALYSIS

The CFD analysis [redacted] was done using the software OpenFOAM (v-5.0) with HELYX-OS GUI.

A. Boundary Conditions

Air enters the computational domain at a inlet velocity of u_1 = 9m/s normal to the inlet surface. The pressure at outlet was kept as total pressure. The walls of the domain was defined as patch with *inlet outlet velocity* and the front and back surfaces of the domain were modeled as *empty*. The faces enclosing the groove were set as *Symmetry*.

B. Turbulence Model

The Reynold's Number was calculated using the freestream velocity and the length of the body. It came out to be $1.43x10^6$. The Laminar model of OpenFOAM is used for this simulation.The flow is Steady and Incompressible.

Reynolds Number Re= UL/v

where,

U - Maximum velocity of the object relative to the fluid,

L - Characteristic linear dimension,

v - Kinematic viscosity

III. THE PROCEDURE

A.HELYX OS GUI:

The HelyxOS is a GUI for Openfoam and is a part of the ENGYS Software.The interface is used to generate the Openfoam directories.

The Mesh is imported in the Openfoam through *fluent3DMeshToFoam* command and is run through the HELYX-OS after the import.

The computational domain is created and case is generated by describing the desired features of the simulation in the GUI.Once the case is ready, the solution is initialized for desired velocity/pressure values.

The Solver can be chosen from the list of compatible solvers provided by the GUI.The solver chosen here is SIMPLE solver.

The directories can be run directly through terminal.

B. THE SIMPLE SOLVER:

Since we want to analyze steady-state Laminar flow for an incompressible fluid, we have used the simpleFoam solver. We do not need to solve the energy equation due to the incompressibility. The SIMPLE(Semi-Implicit Method for PressureLinked Equations) algorithm[2], which the simpleFoam solver is based upon, is solving the momentum equation and the Poisson pressure equation.

$$(\partial \rho u_i/\partial t + \partial \rho u_i u_j/\partial x_j) = -\partial p/\partial x_i + \partial/\partial x_j(\mu \partial u_i/\partial x_i) + \rho f_i$$

$$\partial/\partial x_i(\partial p/\partial x_j) = -\partial/\partial x_i[\rho u_i u_j]$$

As OpenFOAM utilizes a collocated grid, Rhie-Chow interpolation is used for the pressure-velocity coupling.

Following is the SIMPLE algorithm which is the basis of simpleFoam solver:

1) Set the boundary conditions.
2) Solve the discretized momentum equation to compute the intermediate velocity field.
3) Compute the mass fluxes at the cells faces.
4) Solve the pressure equation and apply under-relaxation.
5) Correct the mass fluxes at the cell faces.
6) Correct the velocities on the basis of the new pressure field.
7) Update the boundary conditions.
8) Repeat till convergence.

C. THE POST PROCESSING

The iterations are done and the project is opened in ParaView to infer the results.

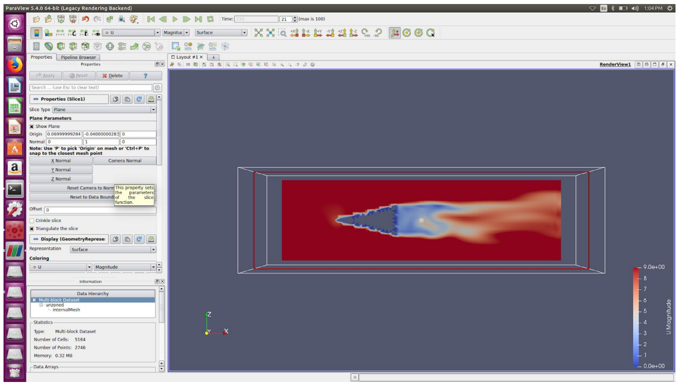

Fig 4 Paraview Interface

IV. RESULTS

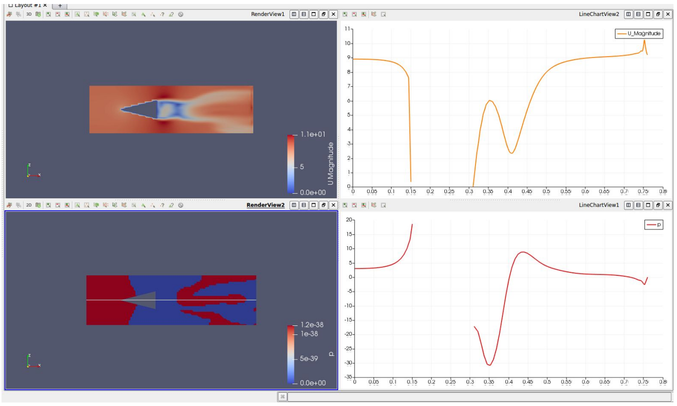

Fig 5 Plots of Conical Flameholders

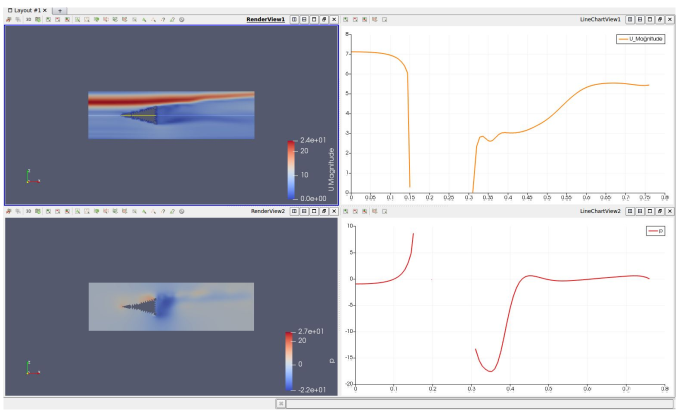

Fig 6 Plots of Saw tooth Flameholders

Inference from plots

- *Saw tooth model has flow attached to the walls and thus gives rise to smaller wake*
- *V gutter produces larger wake but is smaller in length compared to Saw tooth*
- *The reattachment point of Saw tooth model is farther*
- *This result in accordance with the experimental one[1]*

V.REFERENCES

(1)Notched FlameHolders, RISTE 2017, http://data.conferenceworld.in/NITSrinagar/17.pdf
(2)SIMPLE ALGORITHM-Wikipedia, https://en.wikipedia.org/wiki/SIMPLE_algorithm