Simulation of transition between Regular reflection and Mach reflection of a poly-tropic gas in a dual-solution domain.

Abstract:

The simulation is based on analyzing the transition of Regular reflection to a Mach reflection in an inviscid dual-solution domain using the software OpenFOAM and the in-built Paraview for post-processing. It also aims to compare the different results obtained using different wedge angles and pressure variations. The theoretical understanding of the gas dynamics behind the transition region and the origin of Mach stem is well understood but this project aims to computationally verify the theoretical and experimental studies using OpenFOAM. The gas with gamma value of 7/5 is taken (Poly-tropic) and the values of wedge angles, pressure and the speed of flow is slowly varied and the results are observed.

When a shock wave propagates over a solid wedge, the flow generated by the shock strikes the wedge thus generating a second reflected shock, which ensures that the velocity of the flow is parallel to the wedge surface. When the angle between the wedge and the primary shock is made sufficiently large, a single reflected shock is not able to turn the flow to a direction parallel to the wall and transition to Mach reflection occurs. The simulation is based on verifying this aspect.

Software Specifications:

Here to create the geometry and initial conditions, I used OpenFOAM for Windows 18.02 (v1) built by CFD Support (based on Symscape). The geometry is stored in the file BlockMeshDict of the polyMesh folder. The initial conditions of p, U and T are coded too, they're saved in the "0" folder of the respective directory. The "controlDict" file has the necessary details regarding the time interval (time step), courant number and end time, stored in the "system" directory. The simulation of the problem is done by the software called Paraview (5.4) that is automatically downloaded (in-built post-processor) along with the OpenFOAM software.

Geometry:

The geometry variation is the key aspect in this computation. The wedge's angles are varied along with physical conditions for the corresponding normal shock or isentropic condition.

Initially the computation was done for Regular reflection of the shock and the geometry which is saved in the BlockMeshDict file is as shown:

```
/*--------------------------------*- C++ -*----------------------------
-------*\
| =========                 |
|
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
|
|  \\    /   O peration     | Version:  2.3.0
|
|   \\  /    A nd           | Web:      www.OpenFOAM.org
|
|    \\/     M anipulation  |
|
\*---------------------------------------------------------------------
-------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
* * * //

convertToMeters 1;

vertices
(
    (0  0  0)
    (0.5  0  0)
    (1  0.94 0)
    (1  1.5  0)
    (0.5  1.5  0)
    (0  1.5  0)
    (0  0  0.1)
    (0.5  0  0.1)
    (1  0.94 0.1)
    (1  1.5  0.1)
    (0.5  1.5  0.1)
```

```
    (0  1.5  0.1)
  );

  blocks
  (
    hex (0 1 4 5 6 7 10 11) (120 60 1) simpleGrading (1 1 1)
    hex (1 2 3 4 7 8 9 10) (120 60 1) simpleGrading (1 1 1)
  );

edges
(
);

boundary
(
    inlet
    {
        type patch;
        faces
        (
            (0 5 6 11)
       );
    }
    outlet
    {
        type patch;
        faces
        (
            (2 8 9 3)
        );
    }
    bottom
    {
        type symmetryPlane;
        faces
        (
            (0 1 6 7)
        );
    }
    top
    {
        type symmetryPlane;
        faces
        (
            (5 4 11 10)
            (4 3 9 10)
        );
    }
    obstacle
    {
        type patch;
        faces
```

```
        (
            (1 2 8 7)
        );
    }
);

mergePatchPairs
(
);

// ************************************************************//
```

The regular reflection is set at a higher value of vertical height for a clearer picture. The blockMesh command is used and the meshing is done.

The geometry for Single Mach Reflection (SMR) was tried for three different cases for the sake of analysis:

1. First was a wedge with the angle of inclination of 30 degrees, supersonic inlet flow and other conditions that are calculated using isentropic shock relations.
2. Secondly, I simulated the same 30 degree wedge with a subsonic inlet condition for the said wedge and normal shock relations.
3. Computed the simulation using similar conditions as case 2 but for 43 degree wedge angle.

The primary difference amongst the three cases that I simulated has to do with wedge angles alone.

Secondly, the variation of inlet velocity condition.

One was for slightly less than speed of sound (subsonic), the other for about 495m/s.

I simulated both for normal shock relation and isentropic shock relations to observe the behavior and difference in the origin of the stem.

Inlet conditions:

Conditions and Type of Reflection.	p	T	U
Regular Reflection 62 Degrees	Internal Field value- 101325Pa Inlet-455962.5Pa	Internal Field- 295K Inlet-495.6K	Internal Field- 0 Inlet-329.43m/s
Single Mach Reflection-1 30 Degrees	Internal Field value- 101325Pa Inlet-371971.366Pa	Internal Field- 295K Inlet-427.53K	Internal Field- 0 Inlet-495m/s
Single Mach Reflection-2 30 Degrees	Internal Field value- 101325Pa Inlet- 249056.85Pa	Internal Field- 295K Inlet-389.4K	Internal Field- 0 Inlet-329.8m/s
Single Mach Reflection-3 43 Degrees	Internal Field value- 101325Pa Inlet- 249056.85Pa	Internal Field- 295K Inlet-389.4K	Internal Field- 0 Inlet-329.8m/s

Internal field in every case was atmospheric pressure, 25 degree Celsius and zero velocity of the internal field.

Normal shock relations were used to calculate the progress of the parameters for three of the simulations. Regular reflection had an initial shock Mach number of 2 and 62 degree wedge angle. While the conditions for SMR were: initial shock Mach number of 1.5 condition.

Solver used- RhoCentralFoam:

The solver used to model the problem is rhoCentralFoam, based on finite volume like all OpenFOAM solvers. This solver considers compressible fluid. It is a density-based solver in the sense that ρ is the first variable whose solution if found. From that, T and u are calculated. It is “central” in the sense that the variables (ρ, T and u) are calculated and

given as outputs at the centroid of the finite volume. In this solver, finite volumes are contiguous polyhedral cells with an arbitrary number of faces and cells.
The finite volume method for rhoCentralFoam consists in solving the differential equations within an integral over a cell volume, assumed fixed in space. The divergence and gradient terms in the governing equations are converted from volume integrals to surface integrals using Gauss's theorem. During the integral calculation, one must know the fluxes at cell faces. These are evaluated by interpolating the cell centre values to the surface points.
The equations regarding the application of rhoCentralFoam are as follows:
$\partial Q/\partial t+\partial F/\partial x+\partial G/\partial y=0$
Where the spatial differential terms represent the fluxes while Q is the conservative variable vector.
Q can be represented by [ρ ρu ρv ρEt]

$dQ/dt= -R_{ij}$

Now we have to calculate Σ $(Fi.\hat{n})\ ds$.

Fi is given by the equation $Fi=1/2[F_L+F_R]-(\Delta Q) \propto i$

The right hand side terms represent the average of the values at the interface and the value of Numerical Dissipation.
Each scheme in OpenFOAM have the differences based on the equation corresponding to the numerical dissipation.
In rhoCentralFoam, the solver obeys the equation:
$Rho=\Sigma|\lambda_i|\alpha R_i$
Where λ represents the eigenvalues, α represents the coefficients obtained by Rho and R is the right eigenvector matrix.

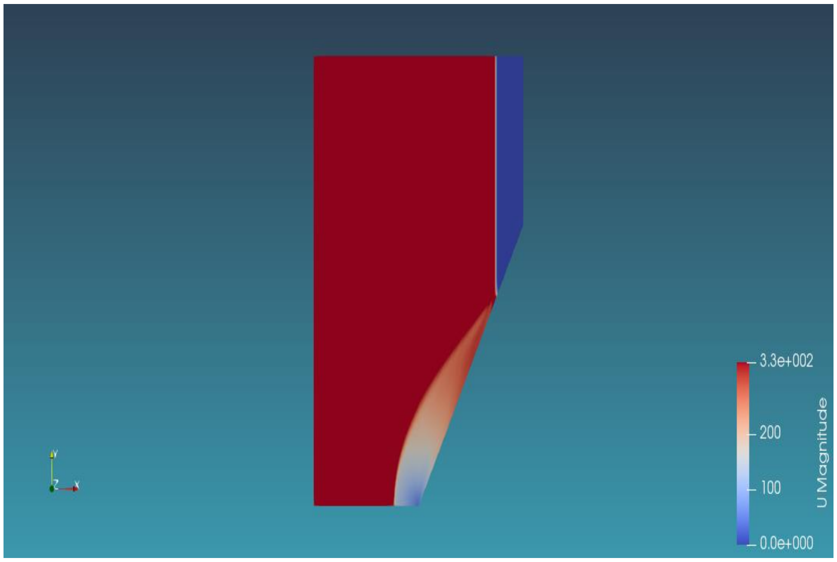
Post Processing:

The post-processing was on Paraview, an inbuilt software in OpenFOAM.
The simulation was compared with the thesis by *Maciej K. Hryniewicki*, from University of Toronto. (Link to the thesis is added to the reference section).

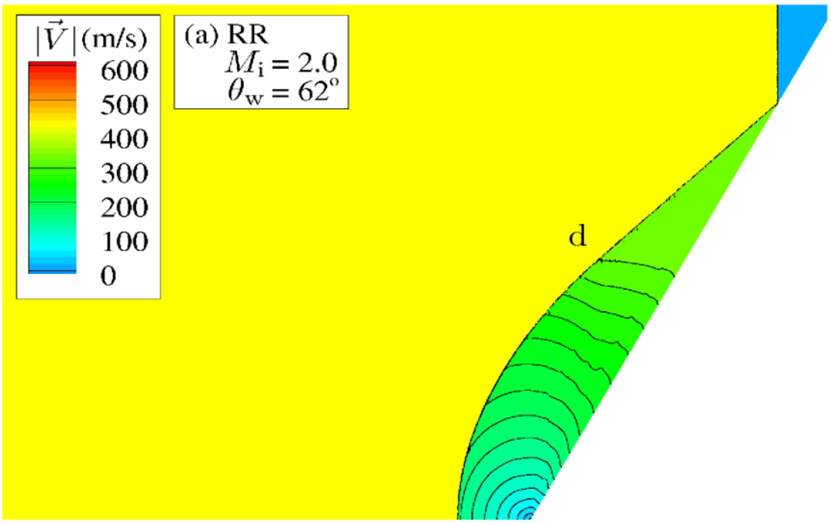
The simulation was primarily focused on analyzing the transition region and conditions.

Regular Reflection:

As mentioned earlier, the height of the wedge was enlarged to show the simulation in a clearer fashion.
We can observe that due to such a high wedge angle, there is an absence of any sort of Mach stem and the reflection is called Regular Reflection. (RR)
This RR pattern occurs at large wedge angles for strong shocks and also at small wedge angles for weak shocks.



Simulation results on OpenFOAM. (Only Mi and angle was similar to the one in the thesis)



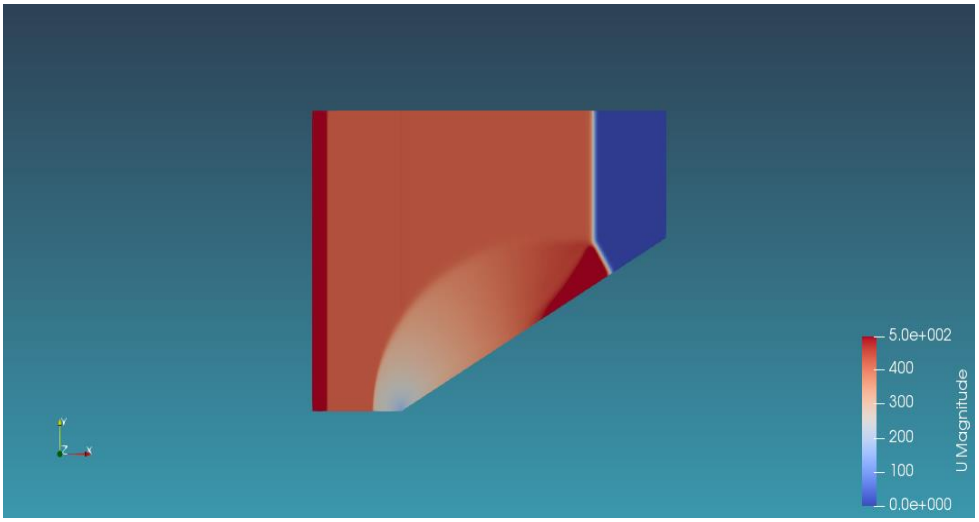
Thesis Results.

Mach Reflection:

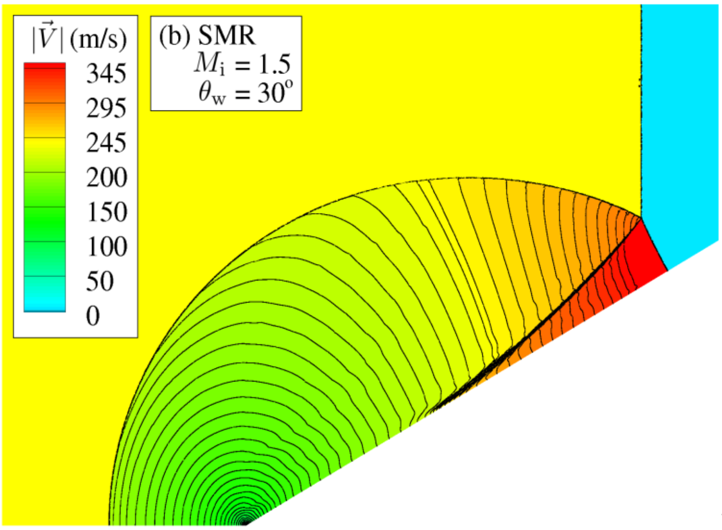
Single Mach Reflection:

1- *30 degree wedge angle: Isentropic relations.*

The simulation was done for 30 degree wedge angle with inlet conditions computed according to isentropic shock relations. The internal field was kept at atmospheric pressure. The inlet velocity was supersonic, and we observe the velocity of the gas near the stem to be about 500m/s. The stem is clearly distinguishable and coincides with the thesis predictions in terms of onset of the stem (planar) but not the flow velocity. (Where the inlet condition was sub-sonic)



Simulation on OpenFOAM.

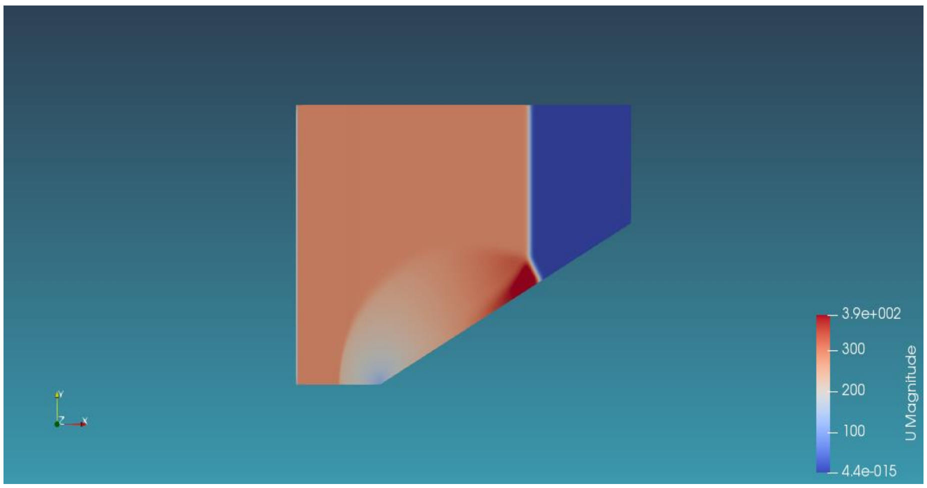


Thesis results.

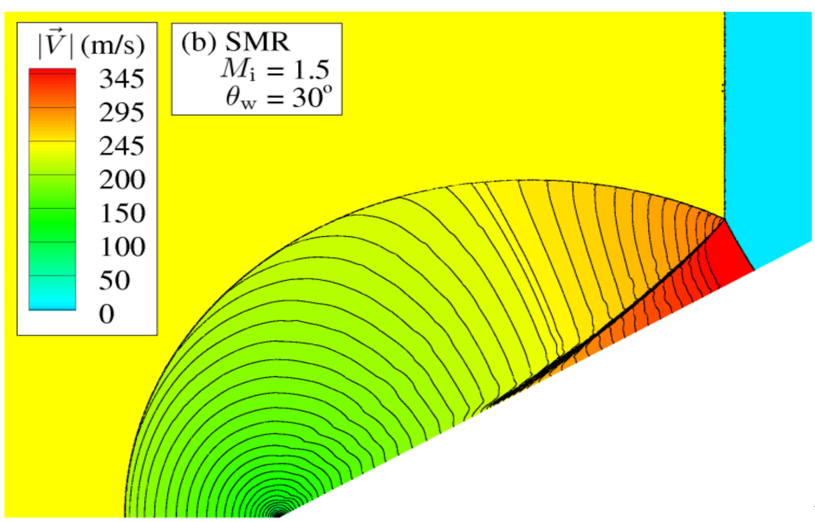
2- Single Mach Reflection:

30 degree wedge angle- Normal Shock Relations.

The simulation was first done for 30 degree wedge angle with inlet conditions computed according to normal shock relations. The internal field was kept at atmospheric pressure. The inlet velocity was subsonic, and we observe the velocity of the gas near the stem to be about 390m/s. The initial shock Mach number was 1.5 and we observe better coincidences in the values of velocity with the thesis when compared to isentropic relation. Onset of the stem position and the flow velocity.



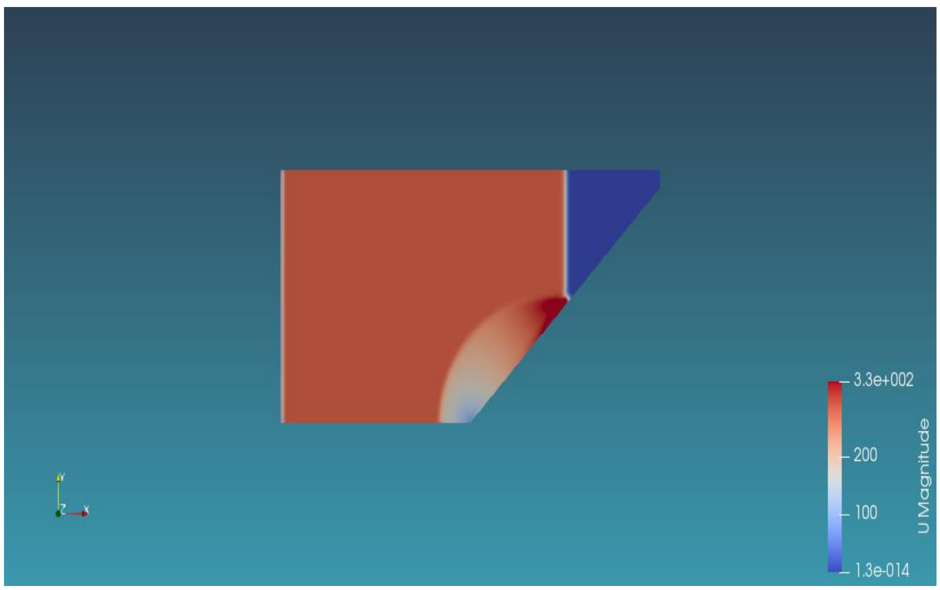
Simulation on OpenFOAM.



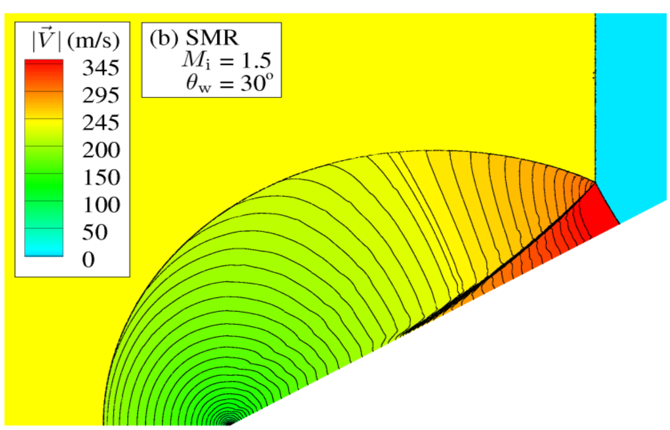
Thesis results.

3- Single Mach Reflection
43 degree wedge, Normal Shock Relations.

The simulation was done for 43 degree wedge angle with inlet conditions computed according to normal shock relations. The internal field was kept at atmospheric pressure. The inlet velocity was subsonic, and we observe the velocity of the gas near the stem to be about 330m/s. The initial shock Mach number was 1.5. We observe the flow velocity at the shock region is closer to 330m/s, lesser than the one observed in a 30 degree wedge angle. The stem is not as clear as it was in the 30 degree wedge and the origin of stem seems curved when compared to the incident planar shock.



Simulation on OpenFOAM.



Thesis results.

Conclusion and Future Work:

The goal of the case-study was to simulate and verify the transition of regular reflection to Mach reflection in an inviscid flow of a poly-tropic gas over wedges of different angles. The initial simulation of Regular reflection was a reference over which SMR was understood. The single Mach reflection was simulated for various cases, and observed that normal shock reflection of 30 degree wedge angle showed the appropriate onset of Mach stem and accurate flow conditions, along with a planar reflected shock as opposed to a curved one in the 43 degree wedge angle case. The transition takes place not according to isentropic behavior which is analytically understood but now computationally verified.

The next step is to simulate Triple Mach reflection and Double Mach reflection for various wedge angles, where there would originate a kink which is observable clearly in DMR.

References:

1- https://thesis.library.caltech.edu/36/
2- http://arrow.utias.utoronto.ca/~groth/theses/Thesis-2016-Hryniewicki.pdf