MATLAB Code Development for 2D Supersonic Nozzle using MOC

Sambhav Jain (SC17B038)
Fifth Semester
Department of Aerospace Engineering,
Indian Institute of Space Science and Technology, Kerala

This report deals with computation of flow in the 2D supersonic nozzle using MOC, assuming steady inviscid flow. Flow in nozzles having sharp corner giving point discontinuity and with smooth wall shapes are solved in this report. Change in mach number, pressure, temperature and density along the flow are studied and the results are compared with CFD results computed through ANSYS software showing good agreement between both the results. The basic working environment used for the flow computation is MATLAB.

Introduction

The physical conditions of a two-dimensional, steady, isentropic, irrotational flow can be expressed mathematically by a non-linear differential equation of the velocity potential. The method of characteristics is a mathematical formulation that can be used to find solutions to the aforementioned velocity potential, satisfying given boundary conditions for which the governing partial differential equations (PDEs) become ordinary differential equations (ODEs).

Although the literature on the design of supersonic nozzles through MOC is extensive, there are very few studies that deal with the study of change in properties across nozzle with definite shape through MOC. This report solves for the change in properties across 2D nozzles having convex cornered wall or convex smooth walls. The flows in the nozzles with convex shapes accompanies expansion wave formation due to which the Mach number increases across the flow and further, in the wall having sharp corner gives point singularity resulting in formation of expansion fan in the flow which has been mathematically modelled in this report.

Nozzle's study has always been an integral part of any systems related to aerospace. Studying flow in nozzles through method of characteristics (MOC) has been a prominent area of research for the researchers dealing with gas dynamics. Some of major researches done recently includes: "Low order supersonic nozzle design using superimposed characteristics"[1], "Inverse method of characteristics with given exit parameters for the computation of supersonic flows"[2], "Numerical study of two-dimensional cylindrical underwater explosion by a modified method of characteristics"[3] and many more. Importance of Method of characteristics in the field of aerospace studies dealing with gas dynamics is the main source of motivation for the work done in this report.

Objective: To compute the flow in the 2D supersonic nozzle using MOC, assuming steady inviscid flow and to plot the axial velocity profile at nozzle exit.

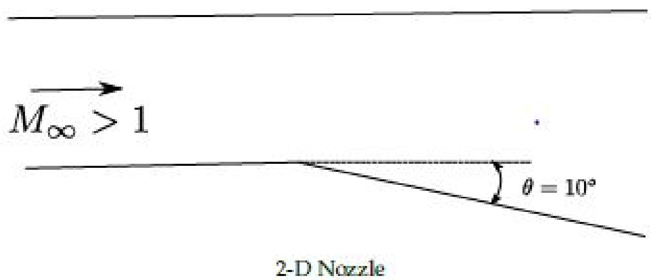


Fig. 1 Problem given for the assignment

Theory

Nozzles

The fundamental purpose of any nozzle is the acceleration of a flow field via the conversion of available pressure and internal energy into kinetic energy. For compressible fluids (gases), it possibly accelerates the flow from subsonic to sonic speeds and ultimately supersonic speeds with sufficiently high inlet pressure ratios. Supersonic nozzles typically consist of three distinct regions: a subsonic converging portion at the inlet, followed by a 'throat' where the cross-sectional area is minimum, and finally a supersonic diverging section ('expander').

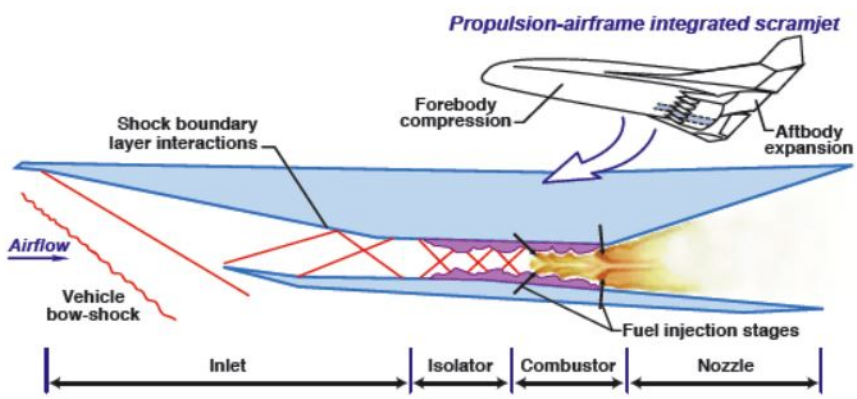


Fig. 2 Scramjet engine. Source: NASA Langley website

In the isentropic flow, following relations are valid and are applicable in nozzle flows where no shock formation is there:

$$\frac{P_o}{P} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\gamma/(\gamma-1)} \tag{1}$$

$$\frac{T_o}{T} = 1 + \frac{\gamma - 1}{2} M^2 \tag{2}$$

$$\frac{\rho_o}{\rho} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{1/(\gamma-1)} \tag{3}$$

Supersonic Expansion by Turning

This section gives the insight into the phenomenon of supersonic flow over a convex turn or convex corner.

WHY IS A TURN THROUGH A SINGLE OBLIQUE WAVE NOT POSSIBLE ?

Normal component of the velocity, u_2, after the mach wave, is greater than the normal component ahead, since the tangential components, v_1 and v_2 must be equal. Although this would satisfy the equations of motion, it would lead to a decrease of entropy as we have [4]:

$$s_2 - s_1 = C_p \ln \frac{T_2}{T_1} - R \ln \frac{p_2}{p_1}$$

From the isentropic relation, we get:

$$\frac{s_2 - s_1}{R} = \ln \left[\left(\frac{p_2}{p_1}\right)^{1/(\gamma-1)} \left(\frac{\rho_2}{\rho_1}\right)^{-\gamma/(\gamma-1)} \right]$$

From the rearrangement and simplifying the momentum equation, we get:

$$\frac{p_2}{p_1} = 1 + \frac{2\gamma}{\gamma + 1}(M_1^2 - 1)$$

$$\frac{\rho_2}{\rho_1} = \frac{(\gamma + 1)M_1^2}{(\gamma - 1)M_1^2 + 2}$$

A more convenient form may be obtained by letting $M_1^2 - 1 = m$. Then,

$$\frac{s_2 - s_1}{R} = \ln \left\{ \left(1 + \frac{2\gamma}{\gamma + 1} m\right)^{1/(\gamma-1)} (1 + m)^{\gamma/(\gamma-1)} \left(\frac{\gamma - 1}{\gamma + 1} m + 1\right)^{\gamma/(\gamma-1)} \right\}$$

Each of three terms obtained by the logarithmic product rule has series expansion and further collecting the term gives:

$$\frac{s_2 - s_1}{R} = \frac{2\gamma}{(\gamma+1)^2}\frac{m^3}{3} + high - order\ terms$$

that is,

$$\frac{s_2 - s_1}{R} = \frac{2\gamma}{(\gamma+1)^2}\frac{(M_1^2 - 1)^3}{3}$$

The increase of entropy is third order in $(M_1^2 - 1)$ and we can again write in terms of pressure as follows:

$$\frac{s_2 - s_1}{R} = \frac{\gamma+1}{12\gamma^2}\left(\frac{\triangle p_1}{p_1}\right)^3$$

In the case of convex turn, it is known that velocity increases and pressure decreases. A reduction in pressure leads to reduction in entropy as seen from above equation, which violates second law of thermodynamics. Hence, flow turn through a single oblique wave is not possible and infinite Mach lines are required to turn the flow. Actually, divergent Mach lines are formed and have a tendency to decrease gradients. Expansion is *isentropic* throughout.

For the case of expansion at corner:

The expansion at a corner occurs through a centered wave, defined by a "fan" of straight Mach lines.

1) The flow up to the corner is uniform, at Mach number M_1, and thus the leading Mach wave must be straight, at Mach angle μ_1. The same argument, together with the limited upstream influence, may be applied to each succeeding portions of flow. The terminating Mach line stands at the angle μ_2 to the downstream wall.
2) The centered wave, more often called a *Prandtl-Meyer expansion fan*, is the counterpart, for convex corner, of the oblique shock at a concave corner.

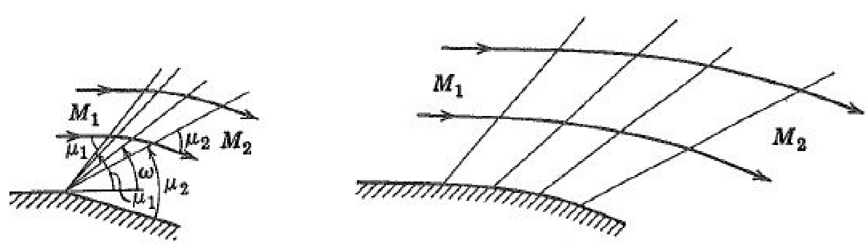


Fig. 3 Expansion fan formation on diverging conditions. Source: Gas Dynamics by: Liepmann and Roshko

The above variation shows a typical expansion over a continuous, convex turn. Since, the flow is isentropic, it is reversible. For example, in the channel formed by any two streamlines, the forward flow is expansive and reverse flow is compressive.

BASIC FORMULATION FOR PRANDTL-MEYER RELATION

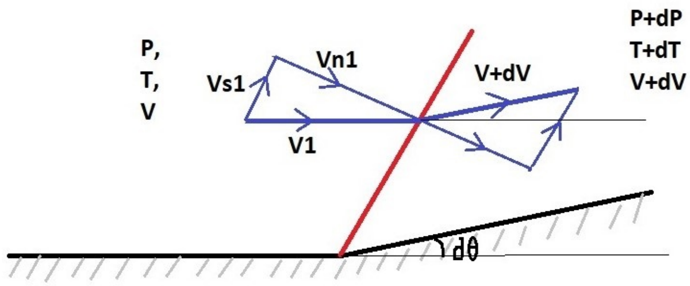


Fig. 4 Properties change across Mach wave

By applying the basic mass conservation and momentum conservation laws, we obtain the following relations:

$$\frac{\triangle p}{p} = \gamma \frac{M^2}{\sqrt{M^2-1}} d\theta \tag{4}$$

$$\frac{\triangle \rho}{\rho} = \frac{M^2}{\sqrt{M^2-1}} d\theta \tag{5}$$

$$\frac{dM}{M} = \left[1 + \frac{\gamma-1}{2} M^2\right] \frac{-d\theta}{\sqrt{M^2-1}} \tag{6}$$

For M=1, $\theta = 0^o$:

$$\theta = \left[\sqrt{\frac{\gamma+1}{\gamma-1}} \arctan \sqrt{\frac{\gamma-1}{\gamma+1}(M^2-1)} - \arctan \sqrt{M^2-1}\right]$$

Right hand side of the above equation is function of Mach number only. Hence, we can define it as a function, ν, which depends of M only and the above relation becomes:

$$\theta_2 - \theta_1 = \nu_2(M) - \nu_1(M)$$

The above relation is known as **Prandtl Meyer Relation**.

Just for brevity, we can define two constants as[5]

$$\alpha = \sqrt{\frac{\gamma+1}{\gamma-1}}$$

$$\beta = \sqrt{M^2-1}$$

Hence, we can rewrite the Prandtl Meyer function as:

$$\nu(M) = \alpha . \arctan\left(\frac{\beta}{\alpha}\right) - \arctan\beta$$

Maximum value of ν occurs when M$\rightarrow\infty$. Hence, we can define maximum value of Prandtl Meyer function or maximum deflection angle as:

$$\nu_{max} = \frac{\pi}{2}\left(\sqrt{\frac{\gamma+1}{\gamma-1}} - 1\right)$$

Method of Characteristics

The general conservation equation for an irrotational flow is given by[6],

$$\left(1 - \frac{u^2}{a^2}\right)\frac{\partial u}{\partial x} + \left(1 - \frac{v^2}{a^2}\right)\frac{\partial v}{\partial y} - \frac{2uv}{a^2}\frac{\partial u}{\partial y} = 0 \tag{7}$$

While solving above equation for $\partial u/\partial x$, above equation have an exception. If expresssion of $\partial u/\partial x$ is evaluated at u=a then denominator becomes zero and then $\partial u/\partial x$ becomes indeterminate and maybe discontinuous. This represent the lines which are called Mach Lines or characteristic lines. The Mach angle of the Mach line is given by,

$$\mu = sin^{-1}\left(\frac{1}{M}\right) = sin^{-1}\left(\frac{u}{V}\right) = sin^{-1}\left(\frac{a}{V}\right) \tag{8}$$

These flow field is solved using this three steps,

1) Finding the characteristic lines in flow field along which flow variables are continuous but its derivatives are indeterminate and along which it may even be sometimes discontinuous.
2) Combining the partial differential conservation equation in such a fashion that ordinary differential equation are obtained which hold only along characteristic lines, this ODE are called compatibility equations.
3) Solving the compatibility equation step by step along the characteristic lines.

Determination of Characteristic lines

For an irrotational flow, velocity and velocity potential is related by following relation,

$$V = \nabla\Phi \tag{9}$$

where, $V = ui + vj + wk$ and $\nabla\Phi = \frac{\partial\Phi}{\partial x}i + \frac{\partial\Phi}{\partial y}j + \frac{\partial\Phi}{\partial z}k$

By comparing coefficient of i,j,k of both sides of equation 9 we get,

$$u = \frac{\partial \Phi}{\partial x} \qquad v = \frac{\partial \Phi}{\partial y} \qquad w = \frac{\partial \Phi}{\partial z}$$

Continuity equation in terms of velocity potential is given by,

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \tag{10}$$

$$\rho(\Phi_{xx} + \Phi_{yy} + \Phi_{zz}) + \Phi_x \frac{\partial \rho}{\partial x} + \Phi_y \frac{\partial \rho}{\partial y} + \Phi_z \frac{\partial \rho}{\partial z} = 0 \tag{11}$$

By using Euler's equation relation,

$$dp = -\rho V dV = -\rho d\left(\frac{\Phi_x^2 + \Phi_y^2 + \Phi_z^2}{2}\right) \tag{12}$$

where $dp = a^2 d\rho$. Substituting relation used in equation 12 into the continuity equation we get,

$$\left(1 - \frac{\Phi_x^2}{a^2}\right)\Phi_{xx} + \left(1 - \frac{\Phi_y^2}{a^2}\right)\Phi_{yy} + \left(1 - \frac{\Phi_z^2}{a^2}\right)\Phi_{zz} - \frac{2\Phi_x\Phi_y}{a^2}\Phi_{xy} - \frac{2\Phi_x\Phi_z}{a^2}\Phi_{xz} - \frac{2\Phi_y\Phi_z}{a^2}\Phi_{yz} = 0 \tag{13}$$

For the steady, adiabatic, two dimensional irrotational supersonic flow, the governing equation reduces to,

$$\left(1 - \frac{\Phi_x^2}{a^2}\right)\Phi_{xx} + \left(1 - \frac{\Phi_y^2}{a^2}\right)\Phi_{yy} - \frac{2\Phi_x\Phi_y}{a^2}\Phi_{xy} = 0 \tag{14}$$

$$\left(1 - \frac{u^2}{a^2}\right)\Phi_{xx} + \left(1 - \frac{v^2}{a^2}\right)\Phi_{yy} - \frac{2uv}{a^2}\Phi_{xy} = 0 \tag{15}$$

where $d\Phi_x = du = (dx)\Phi_{xx} + (dy)\Phi_{xy}$ and $d\Phi_y = dv = (dx)\Phi_{xy} + (dy)\Phi_{yy}$

$$\Phi_{xy} = \frac{\begin{vmatrix} 1 - \frac{u^2}{a^2} & 0 & 1 - \frac{v^2}{a^2} \\ dx & du & 0 \\ 0 & dv & dy \end{vmatrix}}{\begin{vmatrix} 1 - \frac{u^2}{a^2} & -\frac{2uv}{a^2} & 1 - \frac{v^2}{a^2} \\ dx & dy & 0 \\ 0 & dx & dy \end{vmatrix}} = \frac{N}{D} \tag{16}$$

We get unique value of Φ_{xy} for different dx and dy. However, if dx and dy are selected such that D becomes zero then Φ_{xy} is not defined in that particular direction dictated by dx and dy. Therefore in order to keep Φ_{xy} finite N should also be equal to zero; i.e, $\Phi_{xy} = \frac{N}{D} = \frac{\partial u}{\partial y} = \frac{\partial v}{\partial x}$ is indeterminate. Hence D=0 (and hence N=0) represent characteristic lines. Thus setting D=0 we get,

$$\left(1 - \frac{u^2}{a^2}\right)\left(\frac{dy}{dx}\right)^2_{char} + \frac{2uv}{a^2}\left(\frac{dy}{dx}\right)_{char} + \left(1 - \frac{v^2}{a^2}\right) = 0 \tag{17}$$

$$\left(\frac{dy}{dx}\right)_{char} = \frac{-uv/a^2 \pm \sqrt{[(u^2+v^2)/a^2] - 1}}{[1 - (u^2/a^2)]} \tag{18}$$

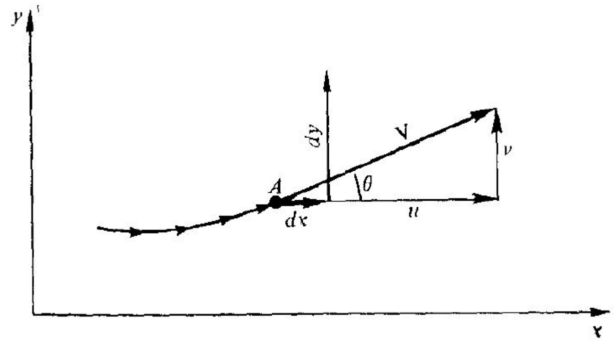


Fig. 5 Streamline Geometry Source: Compressible Flow by Anderson

Inviscid supersonic flow is governed by hyperbolic equations for which M > 1.

At point A, $u = V cos\theta$ and $v = V sin\theta$. Substituting u and v in terms of V and θ and $sin\mu = 1/M = a/V$ in equation 12 we get,

$$\left(\frac{dy}{dx}\right)_{char} = tan(\theta \mp \mu) \tag{19}$$

Compatibility Equation

N=0 determines the compatibility equation. N=0 holds only when D=0, therefore N=0 holds only along the characteristics lines.

$$\left(1 - \frac{u^2}{a^2}\right) du dy + \left(1 - \frac{v^2}{a^2}\right) dx dv = 0 \tag{20}$$

$$\frac{dv}{du} = \frac{-[1 - (u^2/a^2)]}{[1 - (v^2/a^2)]}\left(\frac{dy}{dx}\right)_{char} \tag{21}$$

$$\frac{dv}{du} = \frac{\frac{uv}{a^2} \mp \sqrt{\frac{u^2+v^2}{a^2} - 1}}{1 - \frac{v^2}{a^2}} \tag{22}$$

$$d\theta = \mp\sqrt{M^2 - 1}\frac{dV}{V} \tag{23}$$

Algebraic compatibility equation in terms of Prandtl-Meyer function is given by,

Along the C_- characteristic

$$\theta + \upsilon(M) = const = K_- \tag{24}$$

Along the C_+ characteristics

$$\theta - \upsilon(M) = const = K_+ \tag{25}$$

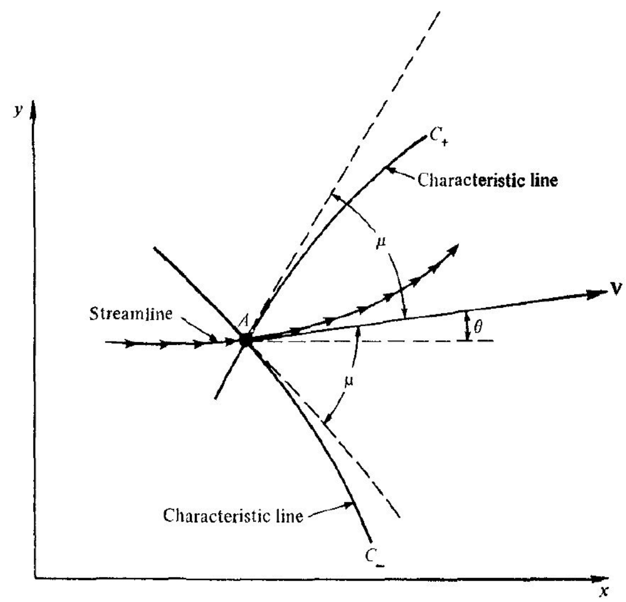


Fig. 6 Illustration of left and right running characteristic lines Source: Compressible Flow by Anderson

MATLAB Script Functions Description

* *prandtl_meyer.m* function

Inputs:

1) M - Mach Number
2) gamma - Ratio of Specific Heats

Output:

1) ν - Prandtl-Meyer Function

MATLAB code:

```
function nu=prandtl_mayer(M,gamma)
% This function returns the value of Prandtl-Meyer function for given Mach
    number and ratio of specific heats
    alpha=sqrt((gamma+1)/(gamma-1));
    beta=sqrt(M^2-1);
    nu=alpha*atan(beta/alpha)-atan(beta);
end
```

* *inv_prandtl_meyer.m* function

Inputs:

1) ν - Prandtl-Meyer Function
2) gamma - Ratio of Specific Heats

Output:

1) M - Mach Number

MATLAB code:

```
function M=inv_prandtl_mayer(nu,gamma)
% This function deals with finding Mach number given Prandtl Meyer function
% Reference: "Inversion of the Prandtl-Meyer relation - I.M. Hall"

    % Finding parameters used in inversion relation specified in reference
    if gamma==1.4
        nu_max=130.4540769*pi/180;
    elseif gamma==5/3
        nu_max=pi/2;
```

```
    end
    y=(nu/nu_max)^(2/3);

    % Finding Mach number value using parameters calculated above and
    % relation specified in reference
    M=(1+1.3604*y+0.0962*(y^2)-0.5127*(y^3))/(1-0.6722*y-0.3278*(y^2));
end
```

* *internal_flow.m* function

Property of each point is defined using: x-coordinate, y-coordinate, θ, Prandtl Meyer function - ν, Mach Number - M and Mach angle - μ

Inputs:

1) Properties of point 1
2) Properties of point 2
3) gamma - ratio of specific heat

Output:

1) Properties at point 3

MATLAB code:

```
function [x_3,y_3,theta_3,nu_3,M_3,mu_3]=internal_flow(x_2,y_2,theta_2,nu_2,mu_2,x_1,y_1,theta_1,nu_1,mu_1,gamma)
    %K minus characteristic line from point 1
    K_minus_1=theta_1+nu_1;
    %K plus characteristic line from point 2
    K_plus_2=theta_2-nu_2;

    %Evaluating Properties at point 3
    theta_3=0.5*(K_minus_1+K_plus_2);
    nu_3=0.5*(K_minus_1-K_plus_2);
    M_3=inv_prandtl_mayer(nu_3,gamma);
    mu_3=asin(1/M_3);

    %Evaluating slope for line joining point 1 and point 3
    slope_1=0.5*(theta_1+theta_3-mu_1-mu_3);
```

```
    %Evaluating slope for line joining point 2 and point 3
     slope_2=0.5*(theta_2+theta_3+mu_2+mu_3);
    %Locating position of point 3 using intersection of above two lines
     x_3=(y_2-y_1+slope_1*x_1-slope_2*x_2)/(slope_1-slope_2);
     y_3=slope_1*x_3+y_1-slope_1*x_1;
end
```

Explanation:

If flowfield conditions are known at two points in the flow, conditions at third point can be evaluated. Point 3 is located by intersection of the C_- characteristic through point 1 and the C_+ characteristic through point 2 (see Fig. 7).

Along the C_- characteristic through point 1 holds:

$$\theta_1 + \nu_1 = (K_-)_1$$

Along the C_+ characteristic through point 2 holds:

$$\theta_2 - \nu_2 = (K_+)_2$$

Hence, at point 3,

$$\theta_3 + \nu_3 = (K_-)_3 = (K_-)_1$$

$$\theta_3 - \nu_3 = (K_+)_3 = (K_+)_2$$

On solving above two equations, we obtain:

$$\theta_3 = \frac{1}{2}[(K_-)_1 + (K_+)_2]$$

$$\nu_3 = \frac{1}{2}[(K_-)_1 - (K_+)_2]$$

Hence, flow properties at point 3 can be evaluated using properties at point 1 and 2.

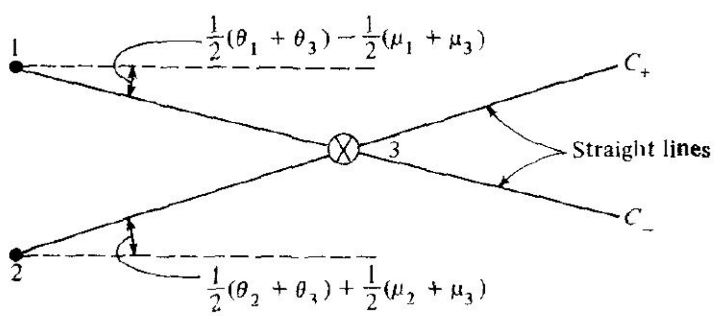


Fig. 7 Approximation of characteristics by straight lines. Source: Compressible Flow by Anderson

The C_+ and C_- characteristics are generally curved lines and we approximate by a sufficiently accurate procedure of assuming characteristic lines to be straight line segments between the grid points, with slopes that are average values.

The C_- characteristic through point 1 is drawn with a average slope given by:

$$[\frac{1}{2}(\theta_1 + \theta_3) - \frac{1}{2}(\mu_1 + \mu_3)]$$

The C_+ characteristic through point 2 is drawn with a average slope given by:

$$[\frac{1}{2}(\theta_2 + \theta_3) + \frac{1}{2}(\mu_2 + \mu_3)]$$

* $across\_fan.m$ function

Inputs:

1) M1 - Mach number before expansion wave
2) theta1, theta2 - flow angles before and after expansion wave
3) gamma - ratio of specific heats

Output:

1) M2, nu_M2, mu_M2 - properties after expansion wave

MATLAB code:

```
%This function gives the properties across fan given mach number at inlet and theta on both sides
function [M2 mu_M2 nu_M2]=across_fan(M1,gamma,theta1,theta2)
    % Finding prandtl meyer function values
    nu_M1=prandtl_mayer(M1,gamma);
    nu_M2=nu_M1-theta2+theta1;
    %Finding properties after expansion wave
    M2=inv_prandtl_mayer(nu_M2,gamma);
    mu_M2=asin(1/M2);
end
```

Explanation:

Using Prandtl Meyer Relation:

$$\nu_1 + \theta_1 = \nu_2 + \theta_2$$

where θ is conventionally defined as negative in clockwise direction. From the above relation, since three quantities are known, we can find the mach number after the expansion fan.

Solving on the Walls:

On Upper wall:

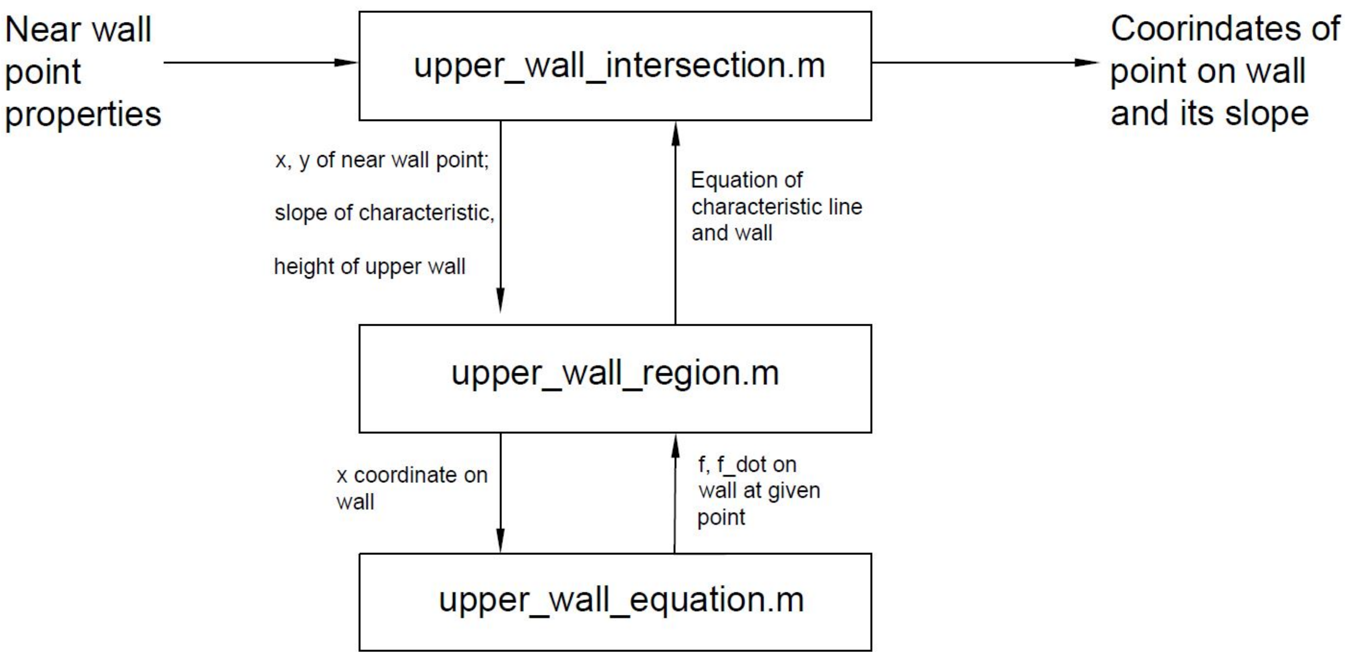


Fig. 8 Flow Chart of MATLAB code used for solving upper wall conditions

* *upper_wall_intersection.m* function

MATLAB code:

```
%This function evaluates slope of characteristics lines and point of
%intersection for characteristics line with upper wall

function [x_5,y_5,theta_5]=upper_wall_intersection(x_4,y_4,theta_4,mu_4,H)
    m=tan(theta_4+mu_4);            %C+ characteristic line
    %Solves two equation to give point of intersection
    x=fsolve(@(x) upper_wall_region(x,x_4,y_4,m,H),[x_4 y_4]);
    x_5=x(1);   y_5=x(2);
    eq=upper_wall_equation(x_5,H);
```

```
    theta_5=atan(eq(2)); %Angle of wall at given x-coordinate
end
```

* *upper_wall_region.m* **function**

MATLAB code:

```
%This function gives output for equation of charaacteristic line and
%equation of wall
function f=upper_wall_region(x,x_1,y_1,m,H)
    %characteristic line intersecting the wall
    f(1)=(x(2)-y_1)-m*(x(1)-x_1);
    %equation of wall
    eq=upper_wall_equation(x(1),H);    y_1=eq(1);
    %Equation of wall
    f(2)=x(2)-y_1;
end
```

* *upper_wall_equation.m* **function**

MATLAB code:

```
%This function is used to evaluate slope and y-coordinate on upper wall for
%given value of x-coordinate
function k=upper_wall_equation(x,H)
    %Initial point for start of upper wall is (0,H)
    f=H+0*x;
    f_dot=0;
    k=[f f_dot];
end
```

On Lower wall:

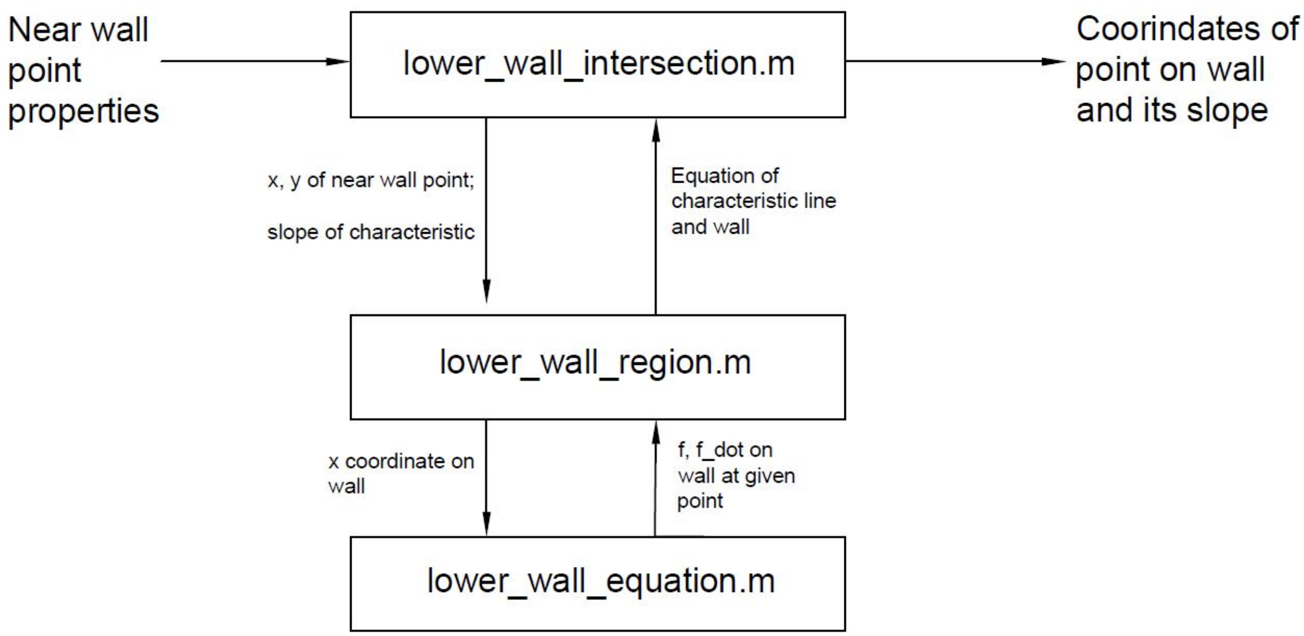


Fig. 9 Flow Chart of MATLAB code used for solving lower wall conditions

* *lower_wall_intersection.m* **function**

MATLAB code:

```
%This function evaluates slope of characteristics lines and point of
%intersection for characteristics line with lower wall
function [x_5,y_5, theta_5]=lower_wall_intersection(x_4,y_4,theta_4 ,mu_4)
    m=tan(theta_4 -mu_4);            %C- characteristic line
    %Solves two equation to give point of intersection
    x=fsolve(@(x) lower_wall_region(x,x_4,y_4,m),[x_4 y_4]);
    x_5=x(1);   y_5=x(2);
    [y,y_dot]=lower_wall_equation(x_5);
    %Angle of wall at given x-coordinate
    theta_5=atan(y_dot);
end
```

* *lower_wall_region.m* function

MATLAB code:

```
%This function gives output for equation of charaacteristic line and
%equation of wall
function f=lower_wall_region(x,x_1,y_1,m)
    %characteristic line intersecting the wall
    f(1)=(x(2)-y_1)-m*(x(1)-x_1);
    %equation of wall
    eq=lower_wall_equation(x(1));      y_1=eq(1);
    f(2)=x(2)-y_1;
end
```

* *lower_wall_equation.m* function

MATLAB code:

```
%This function is used to evaluate slope and y-coordinate on upper wall for
%given value of x-coordinate
function [f,f_dot]=lower_wall_equation(x)
    %Initial point for start of upper wall is (0,0)
    f=-tand(10).*x;
    f_dot=-tand(10);
end
```

* $lower\_wall\_flow.m$ function

Inputs:

1) Properties of near wall point
2) Slope of wall point

Output:

1) Properties at the wall point

MATLAB code:

```
%This function calculates properties at lower wall point using Prandtl Meyer
%equation
function [nu_5,M_5,mu_5]=lower_wall_flow(theta_4 ,nu_4,theta_5 ,gamma)
    nu_5=nu_4+theta_4-theta_5;      %C- Characteristic Line
    M_5=inv_prandtl_mayer(nu_5, gamma);
    mu_5=asin(1/M_5);
end
```

Explanation:

On the lower wall point, the value of C_- characteristic line is same as that of near wall point. Equating, C_- characteristic at the near wall point and lower wall point gives:

$$\nu_5 + \theta_5 = \nu_4 + \theta_4$$

From the above two equations, value of Prandtl-Meyer function at wall point can be calculated.

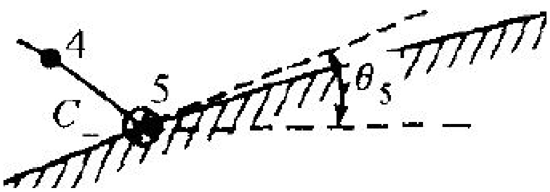


Fig. 10 MOC solving on lower wall. Source: Compressible Flow by Anderson

* $upper\_wall\_flow.m$ function

Inputs:

1) Properties of near wall point
2) Slope of wall point

Output:

1) Properties at the wall point

MATLAB code:

```
%This function calculates properties at upper wall point using Prandtl Meyer
%equation
function [nu_5,M_5,mu_5]=upper_wall_flow(theta_4,nu_4,theta_5,gamma)
    nu_5=nu_4-theta_4+theta_5;        %C+ Characteristic Line
    M_5=inv_prandtl_mayer(nu_5, gamma);
    mu_5=asin(1/M_5);
end
```

Explanation:

On the lower wall point, the value of C_+ characteristic line is same as that of near wall point. Equating, C_+ characteristic at the near wall point and lower wall point gives:

$$\nu_5 - \theta_5 = \nu_4 - \theta_4$$

From the above two equations, value of Prandtl-Meyer function at wall point can be calculated.

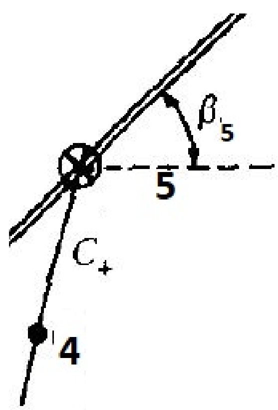
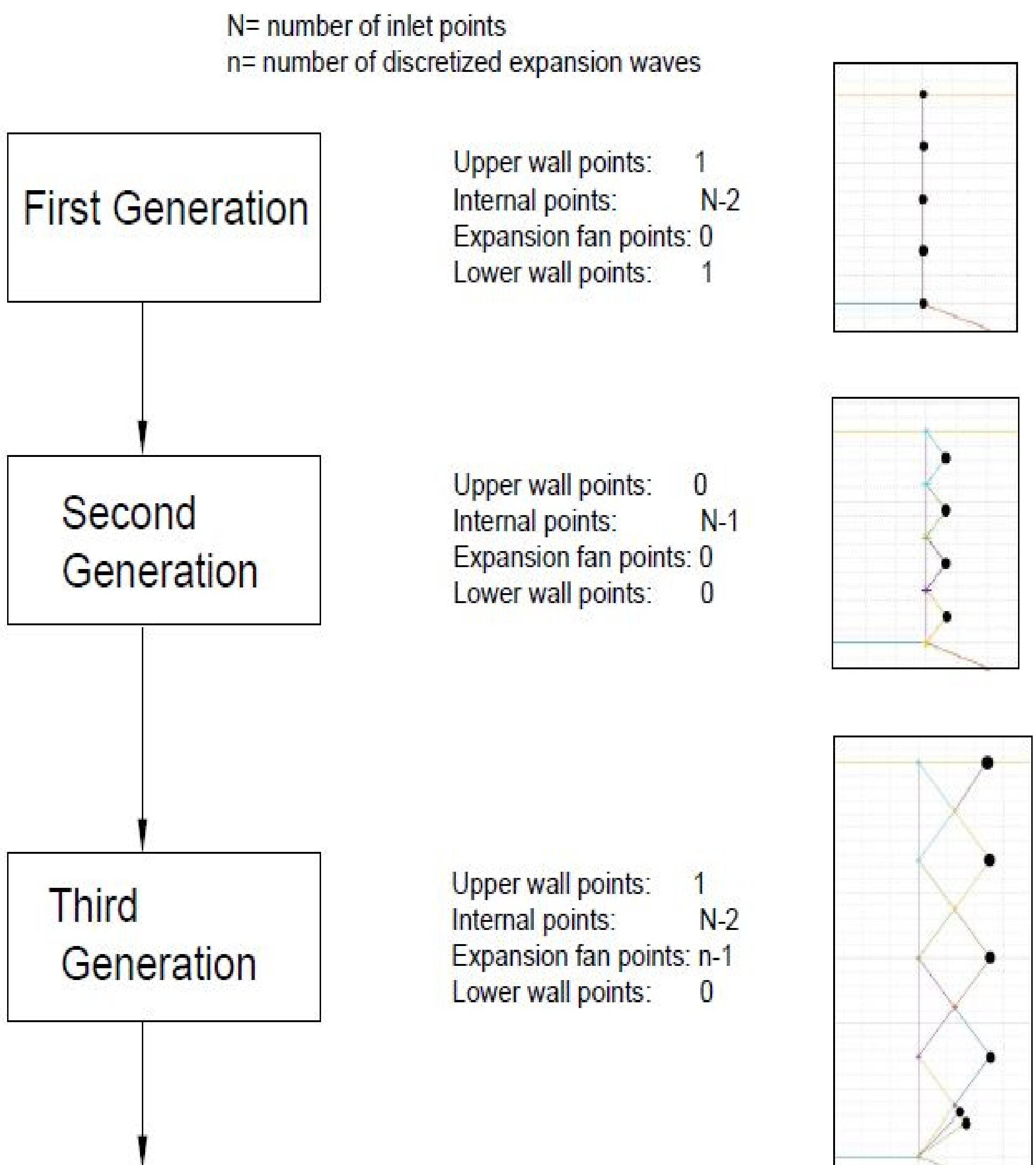


Fig. 11 MOC solving on upper. Source: Compressible Flow by Anderson

Index Notations



jth Generation

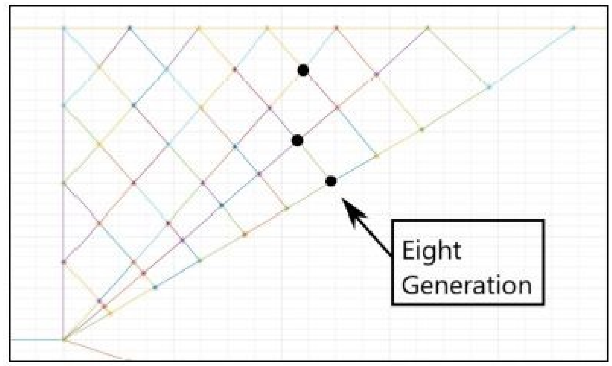
j varies from 4 to 2N+n-1

Internal Points >=0 (Simple Region)

j :	4	5	6	7	8
Upper wall points:	0	1	0	1	0
Internal points:	N-2	N-3	N-3	N-4	N-4
Expansion fan points:	n-1	n-1	n-1	n-1	n-1
Lower wall points:	0	0	0	0	0

Generalized:

Upper Wall Points: mod(j,2)
Internal Flow Points: N-GIF((j+1)/2)
Expansion Fan points: n-1
Lower Wall Points: 0

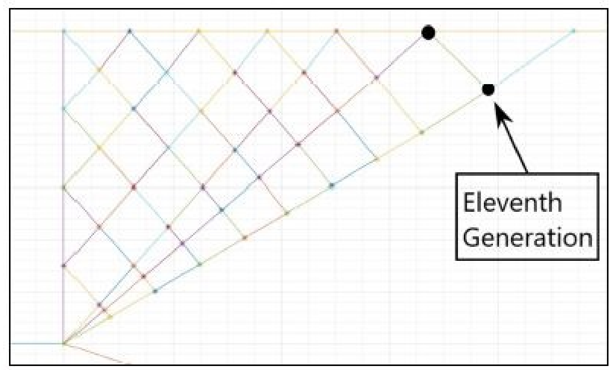


Internal Points <0 (Non-simple region)

j :	11	12
Upper wall points:	1	1
Internal points:	N-6	N-6
Expansion fan points:	n-2	n-3
Lower wall points:	0	0

Generalized:

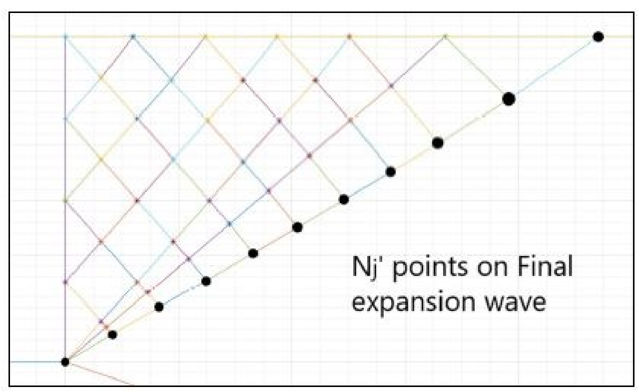
Upper Wall Points: 1
Internal Flow Points: N-GIF((j+1)/2)
Expansion Fan points: (Previous gen. exp. fan points) -1
Lower Wall Points: 0



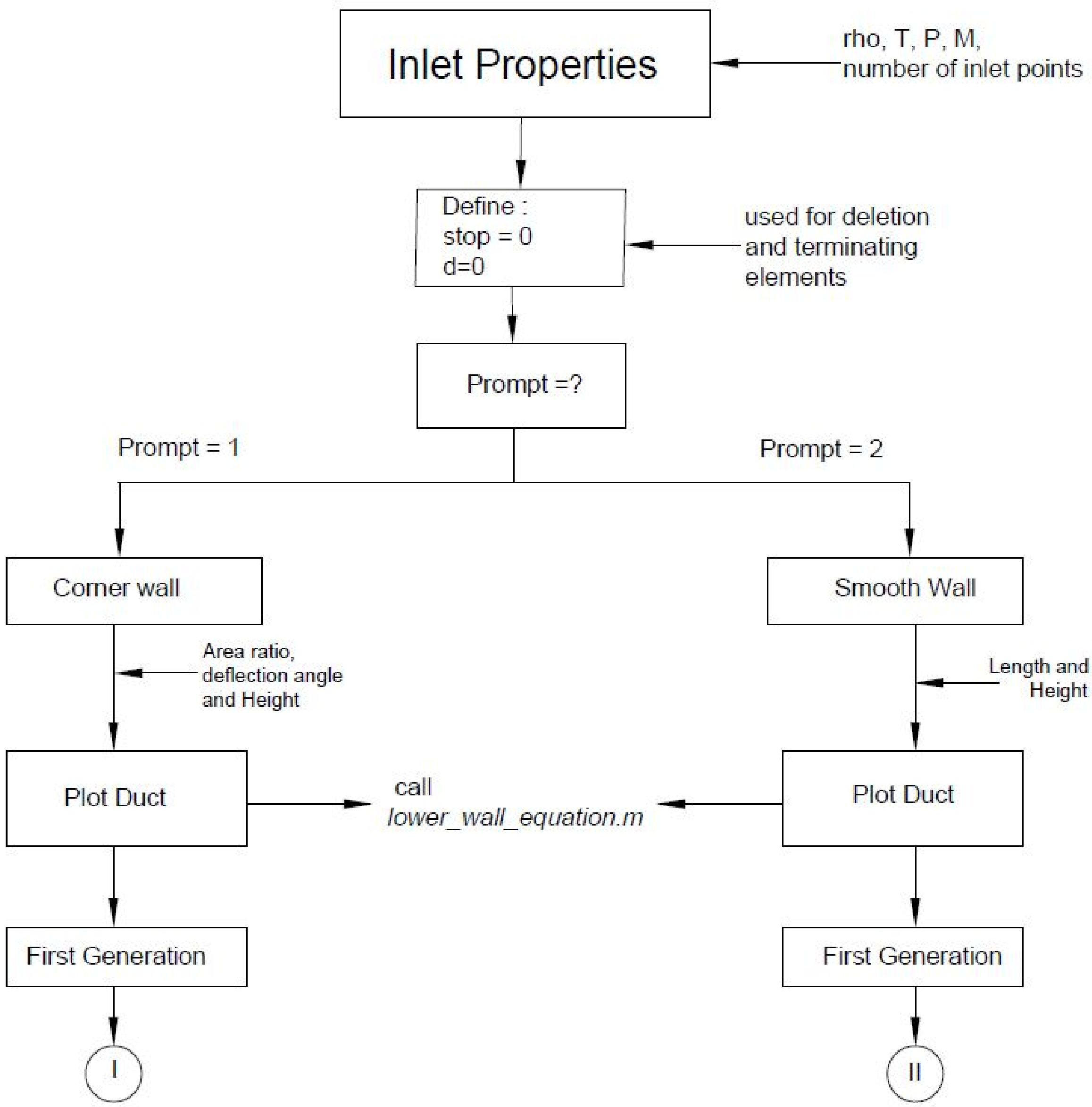
Propagation

j=1 last wave of first expansion fan
Nj' = Number of generations before propagation -1

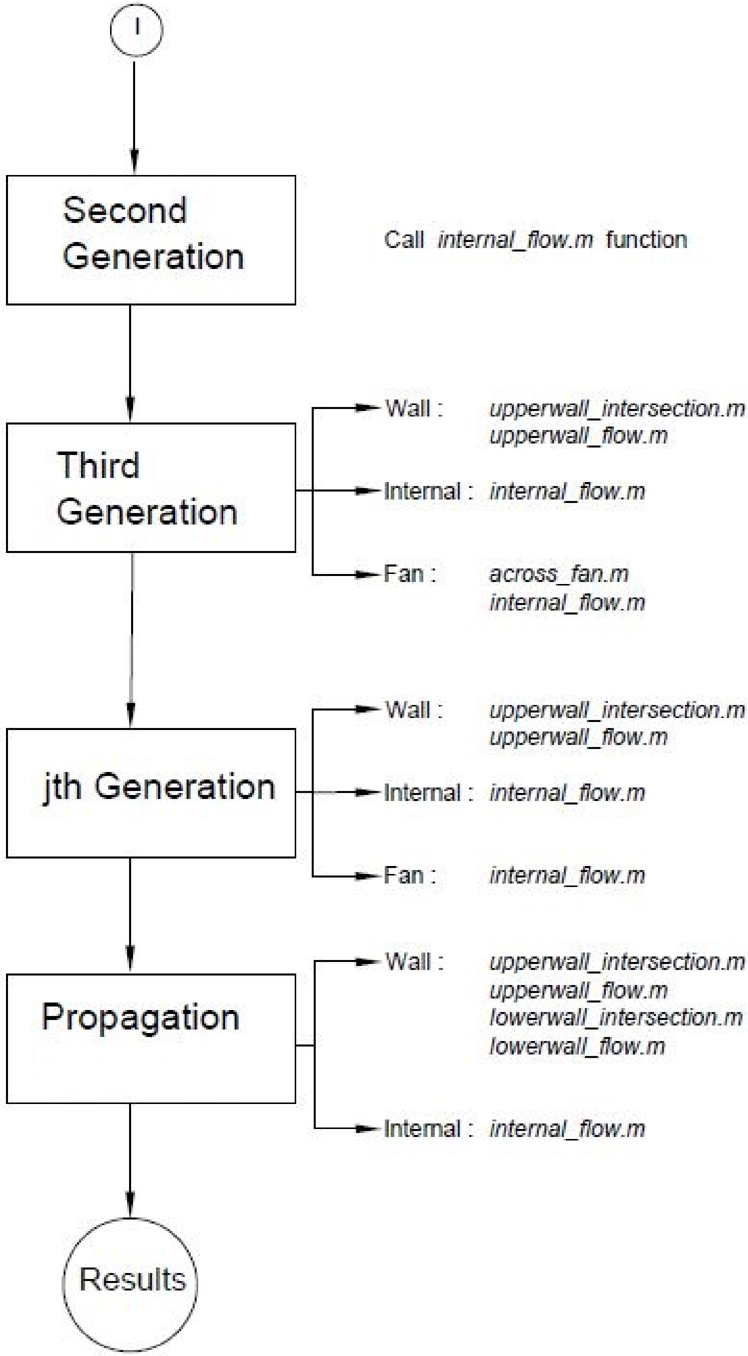
Upper Wall Points: 1
Internal Flow Points: Nj' -2
Lower Wall Points: 1



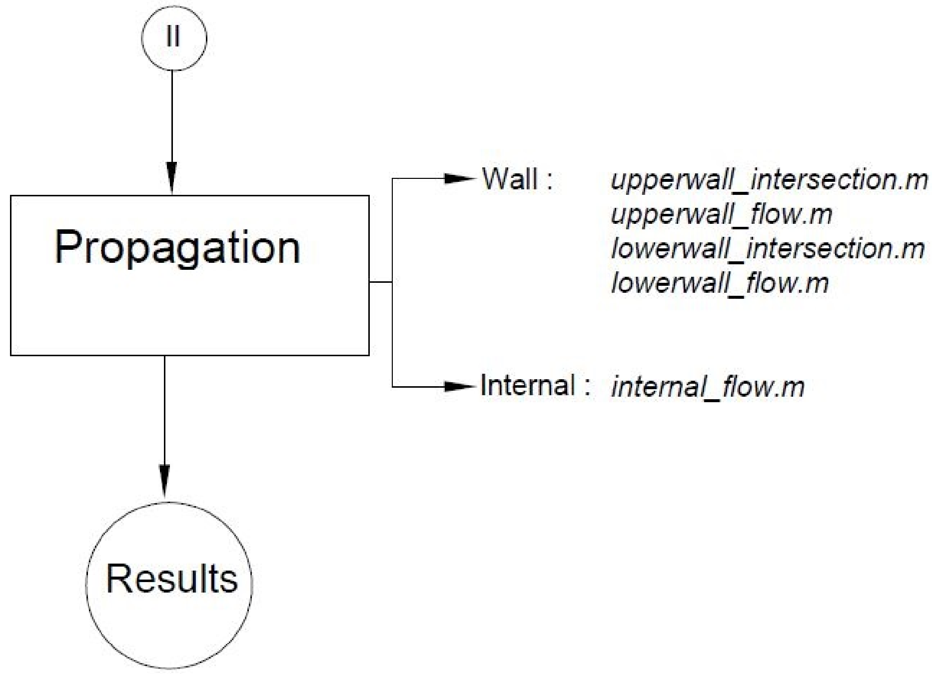
Main Code Flow Chart



Corner wall Flow Chart



Curved smooth wall Flow Chart



Numerical Model

A CFD analysis was performed (using ANSYS Fluent solver) in order to validate the result obtained from the numerical analysis using Method of Characteristics (MOC). The analysis was done on a 2D wedge like nozzle and on a 2D nozzle with one wall having a curved shape.

Geometry

Figure 12 and 13 shows the geometry of the 2D nozzle which was used in numerical analysis. It should be noted that the center used in MOC analysis and ANSYS analysis of geometry used in Figure 12 are different as shown in geometry.

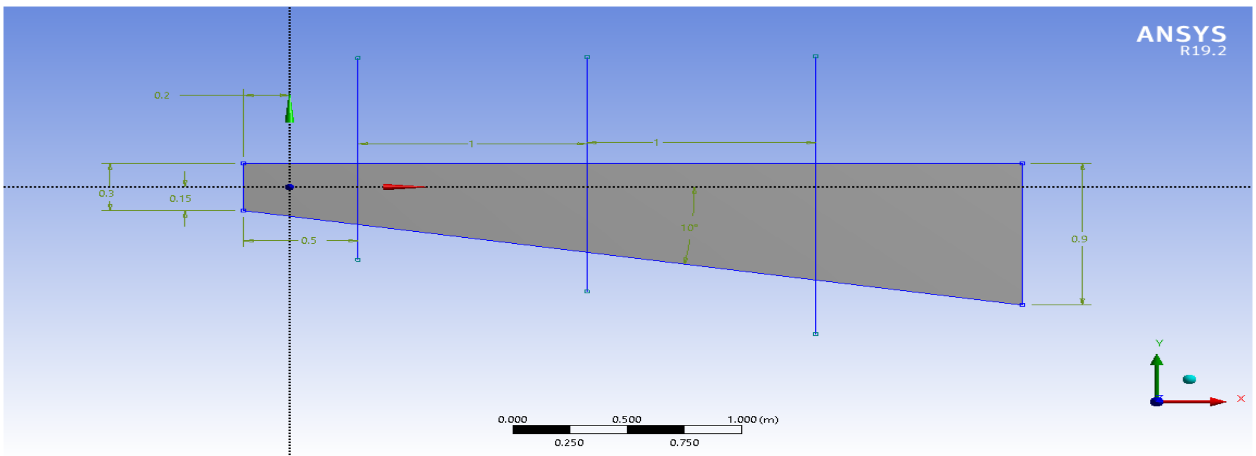


Fig. 12 Geometry of Wedge type Nozzle

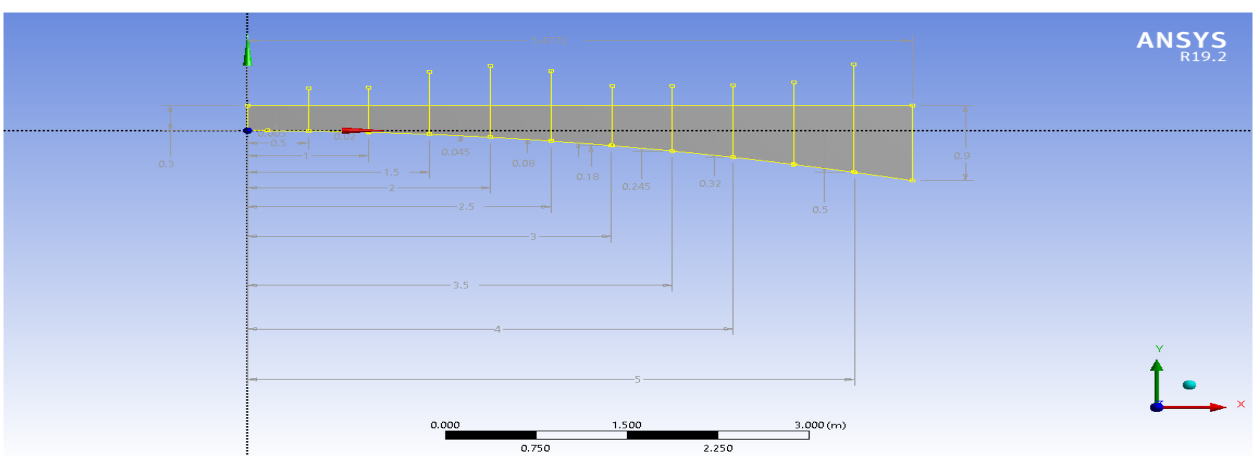


Fig. 13 Geometry of curved wall Nozzle

The basic input parameter given by user are the angle of wall and area ratio. An assumption was made that geometry has a unit thickness and distance between upper wall and lower wall represent the area of the nozzle. Also for the analysis, the inlet area was assumed to be of 0.3 m^2, the outlet area was obtained using area ratio and the lower wall angle was used to fix the length of nozzle. In order to set proper inlet parameters, the geometry was started from the corner point in the Figure 12 and in Figure 13 it was started from the point where the curved wall is starting. For the curved wall, equation $y = -x^2/50$ represent the curved wall. Its length was also obtained with the help of this equation considering area ratio to be 3.

The total dimension of the geometry is also given in the figure 12 and 13.

Mesh

Mesh was designed such that, the mesh grid shape remains almost parallel to top wall, with some slight deviation towards lower wall. An 'all quadrilateral' shaped elements was used. The whole nozzle was divided into several section such that, a uniform number of elements were made with high precision or fine mesh at the inlet. Each section was divided into equal number of sizing elements. A face mapped mesh was used in order to get uniform mesh. Following figure shows the mesh of both wedged and curved 2D nozzles. The total number of elements and nodes in the wedge wall nozzle is 19880 and 20235 respectively, and for curved wall it is equal to 18286 and 18774 respectively.

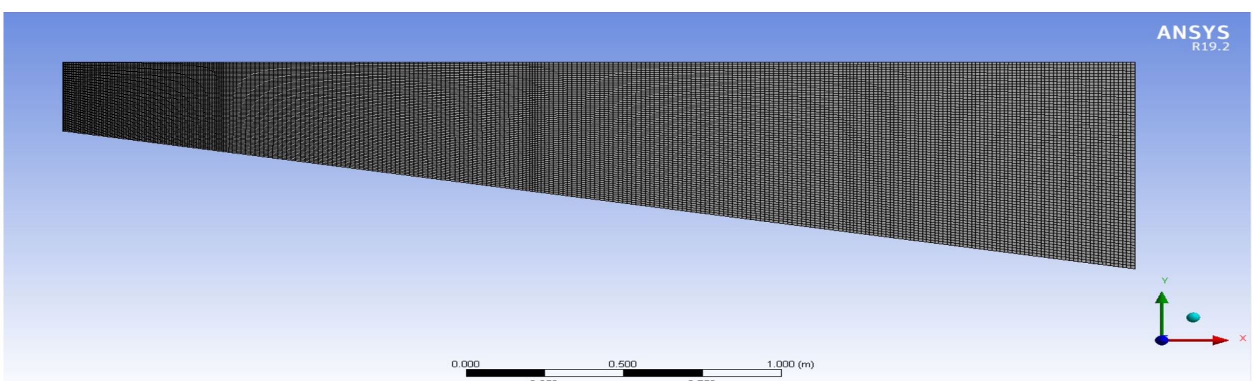


Fig. 14 Mesh of the Corner Nozzle

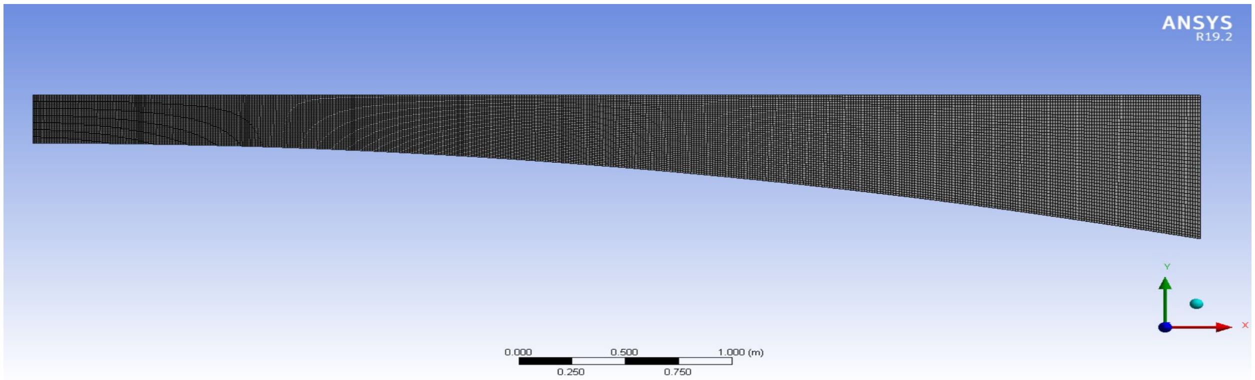


Fig. 15 Mesh of curved wall Nozzle

Input Parameter

The problem is solved using a 'density based' model which is used for compressible flow. Energy equation is also used to account for the temperature change in the supersonic compressible flow. Also the flow is assumed to be inviscid because if friction effect are cosidered then it may lead to boundary layer formation. Also in order to compare the ANSYS result with that of the MOC we need to keep the same assumption as that of the MOC. And inviscid is one of the assumption made in the MOC.

Air was used as fluid material. Air was assumed to be an ideal gas with C_p = 1006.43 J/kg-K and molecular weight of 28.966 kg/kmol.

Boundary Condition

In order to solve a compressible flow, 'Pressure inlet' and 'Pressure outlet' boundary condition at the inlet and outlet are used respectively. 'Velocity inlet' boundary condition at the inlet can't be used for compressible flow. 'Supersonic/Initial Gauge Pressure', 'total pressure', 'Total Temperature' were given as input. An arbitrary Supersonic/Initial Gauge Pressure equal to 10 KPa is used and total gauge pressure used is such that it gives the desired mach number at the inlet. Total pressure is computed using following relation:

$$P_{total} = P_{static}\left(1 + \frac{\gamma - 1}{2}M^2\right)^{\frac{\gamma}{\gamma-1}} = 57457.96Pa \tag{26}$$

Total Temperature was set as 300 K.

The outlet pressure is arbitrary selected as 3700 Pa. In supersonic flow, this outlet pressure is neglected and is obtained according to the flow. 'Wall' is used as boundary condition for upper and lower wall of the nozzle.

Solver

In order to solve the model, ROE-FDS inbuilt ANSYS solver was used. This was solved with second order upwind flow. The residual monitor was set at $1e^{-12}$.

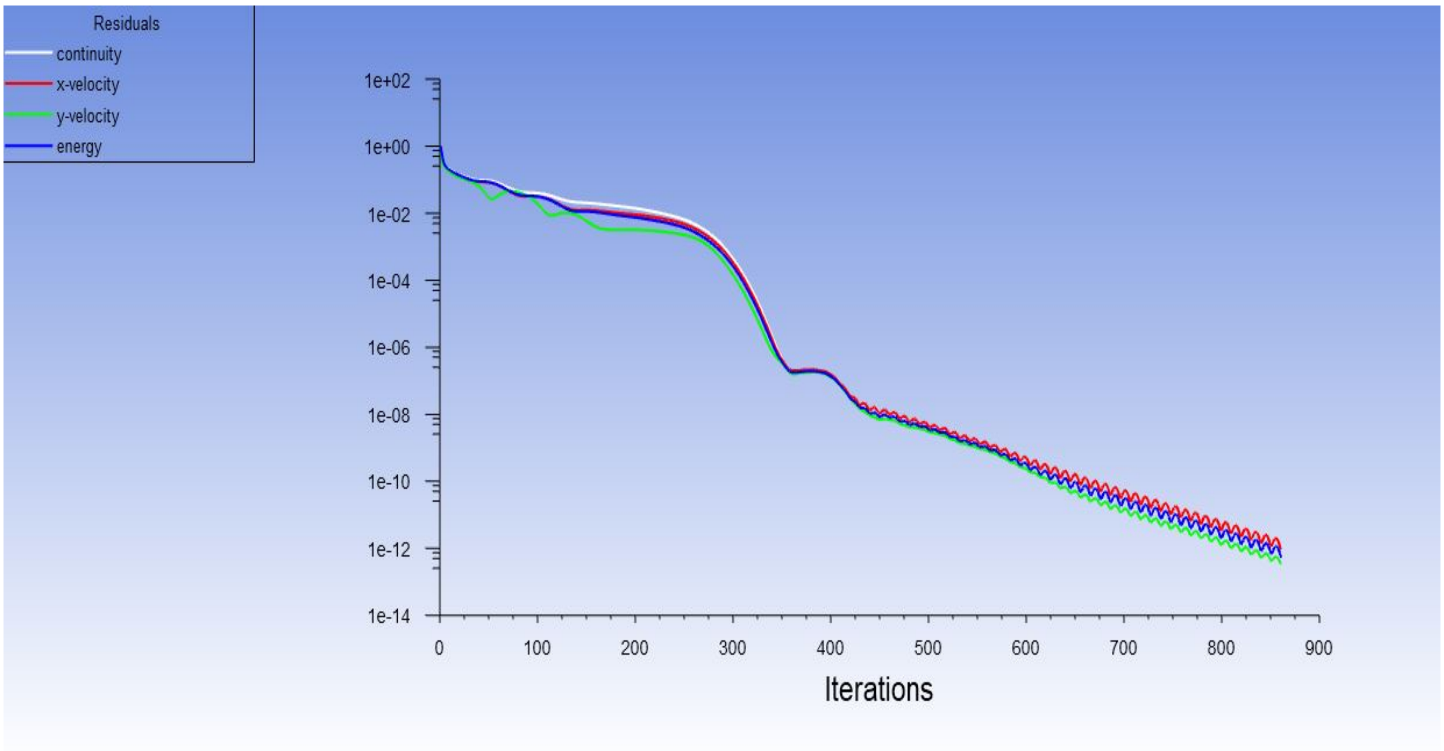


Fig. 16 Residual Plots for CFD Simulation

Results and Discussion

I. Cornered Wall

Characteristic grid network in channel

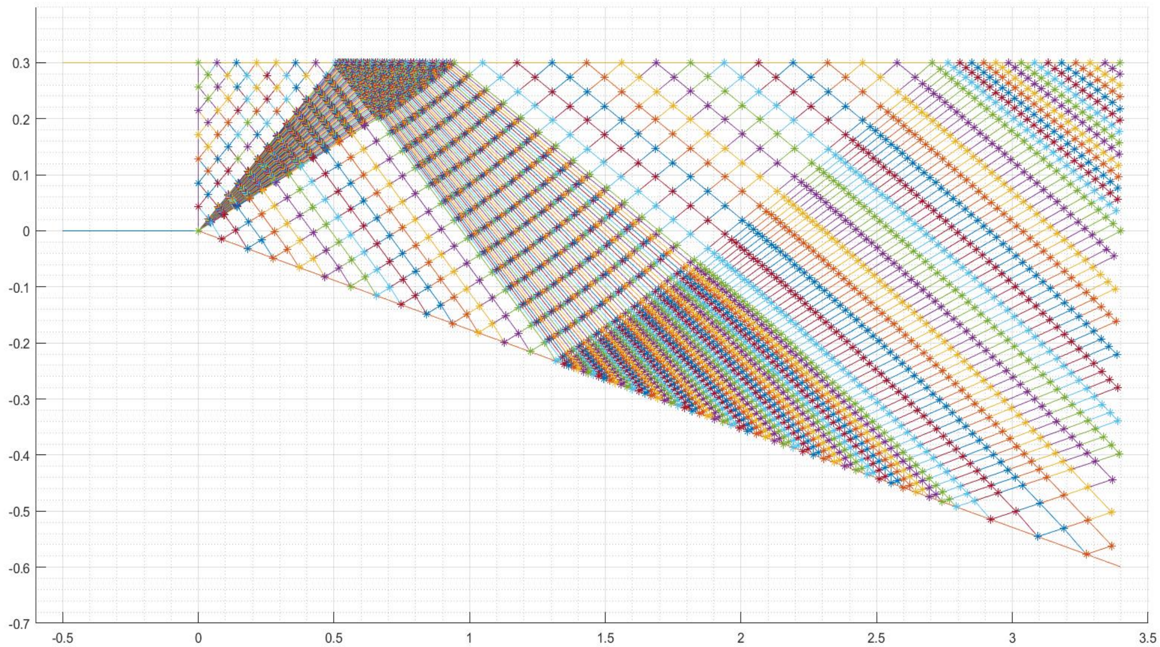


Fig. 17 Characteristic grid network plot for corner wall

Matlab and CFD Results

The results of CFD and MOC almost coincide with each other. The differences arise at the outlet where the density of nodes is less when compared to that at the inlet of the nozzle in MOC. Due to less number of nodes the interpolation (adopted for extracting the results at different sections) leads to approximation errors.

At constant y = 0.15 m along the duct

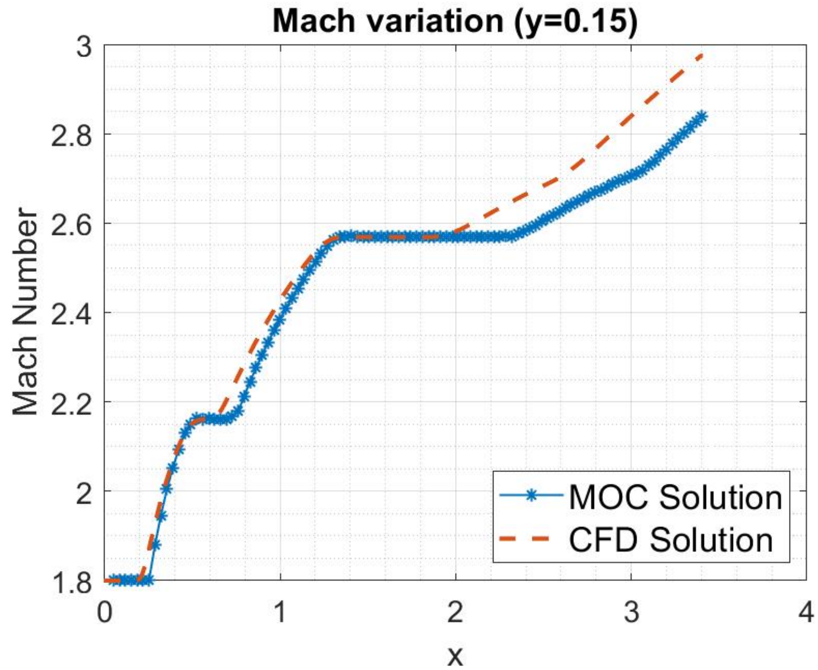


Fig. 18 Mach variation at y = 0.15 m of the nozzle

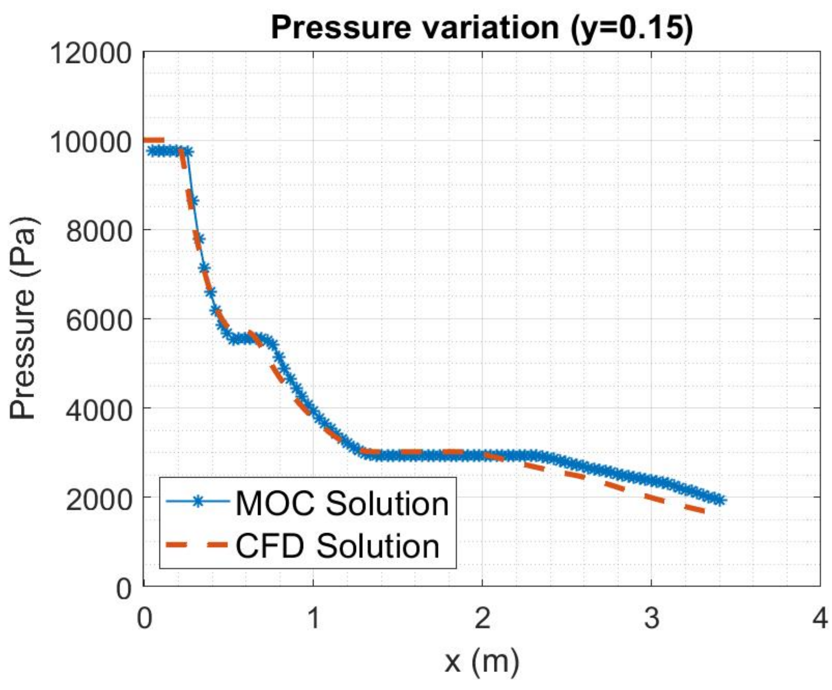


Fig. 19 Pressure variation at y = 0.15 m of the nozzle

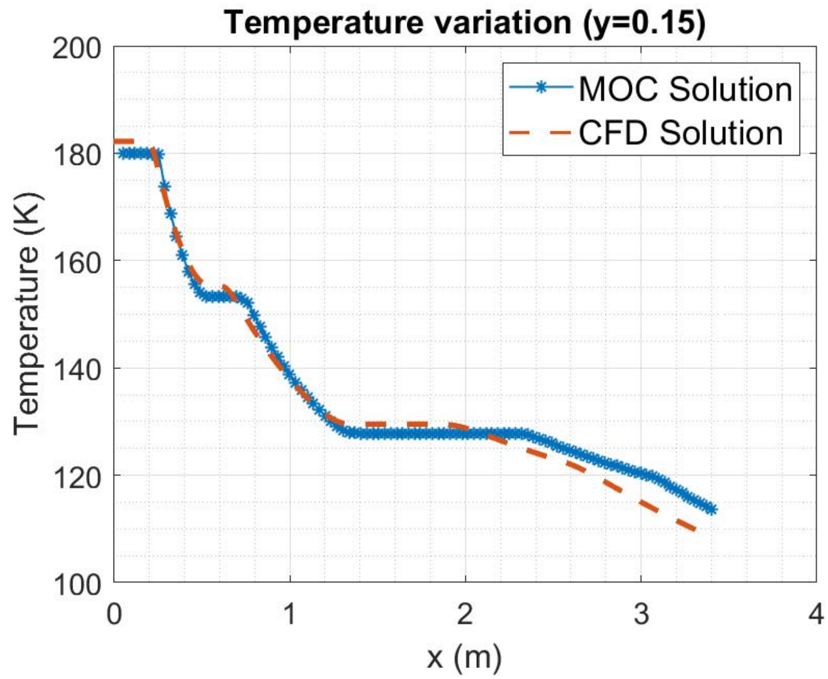


Fig. 20 Temperature variation at y = 0.15 m of the nozzle

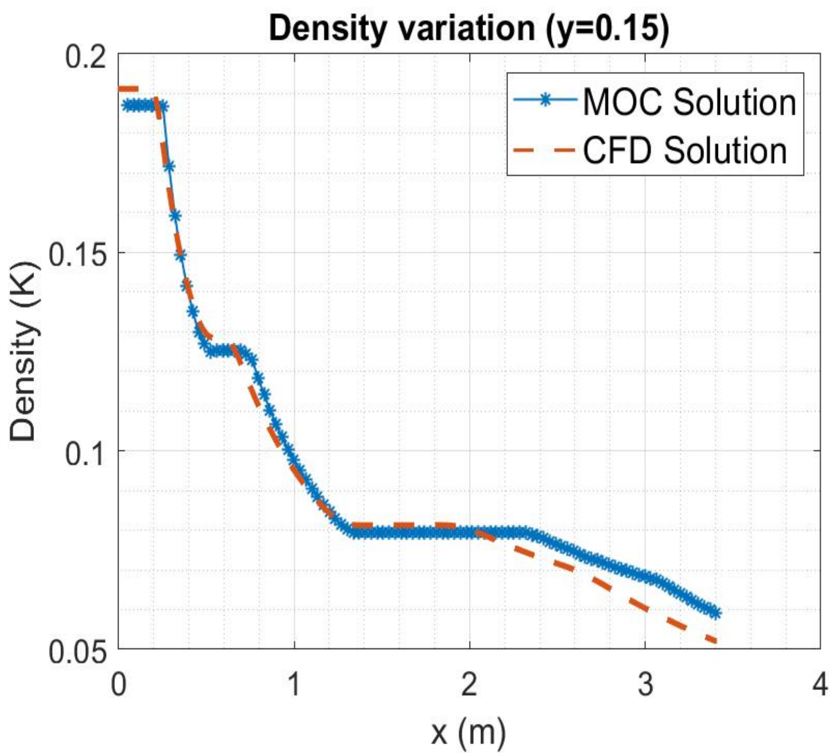


Fig. 21 Density variation at y = 0.15 m of the nozzle

The variation of mach number at y = 0.15 m of the nozzle is shown in figure **??**. It shows that overall mach number increases along the direction of flow. This is because whenever the flow crosses the expansion fan its mach number increases, as according to equation 6, for $d\theta$ being negative, dM should be positive to satisfy the equation. The mach number in the figure is constant in some region as in those regions the flow doesn't encounter any expansion fan.

The variation of pressure at y = 0.15 m of the nozzle is shown in figure **??**. It shows that overall pressure decreases along the direction of flow. This is because whenever the flow flow crosses the expansion fan its pressure decreases, as according to equation 4, for $d\theta$ being negative, dP should also be negative to satisfy the equation. The pressure in the figure is constant in some region as in those regions the flow doesn't encounter any expansion fan.

The variation of temperature at y = 0.15 m of the nozzle is shown in figure **??**. It shows that overall temperature decreases along the direction of flow. This is in accordance with equation 2, in which the variation in static temperature depends on the variation of mach number as stagnation temperature remains constant (for adiabatic flow). Therefore the temperature changes in the region where the mach number changes and remains constant in the region where the mach number is constant.

The variation of density at y = 0.15 m of the nozzle is shown in figure **??**. It shows that overall density decreases along the direction of flow. This is in accordance with equation 3, in which the variation in density depends on the variation of mach number as stagnation density remains constant (for isentropic flow). Therefore the density changes in the region where the mach number changes and remains constant in the region where the mach number is constant.

At upper wall of the duct

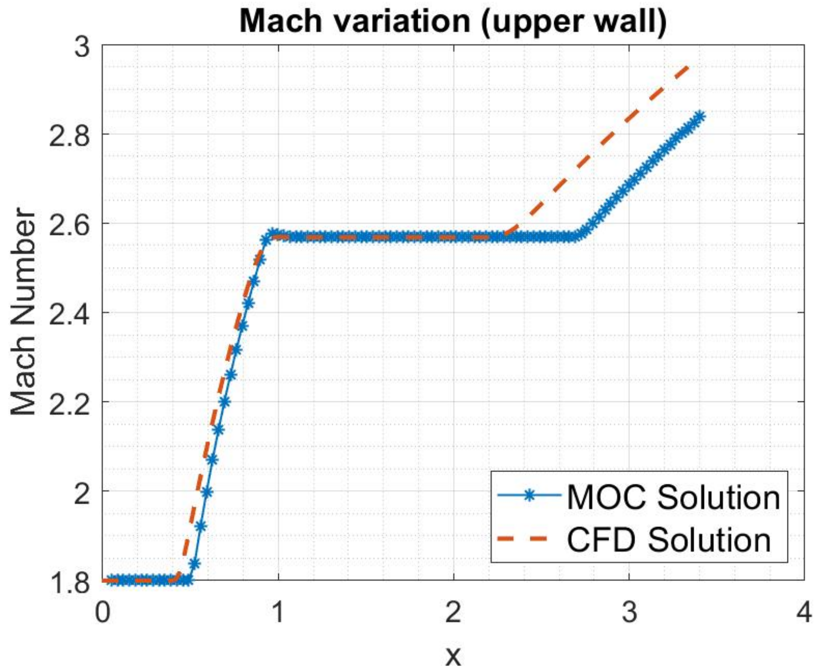


Fig. 22 Mach variation at upper wall of the nozzle

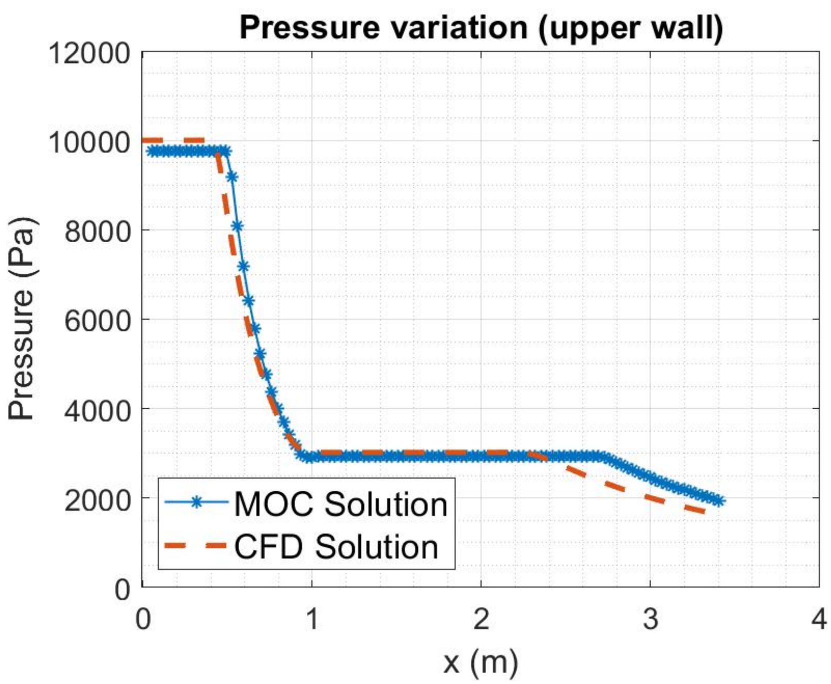


Fig. 23 Pressure variation at upper wall of the nozzle

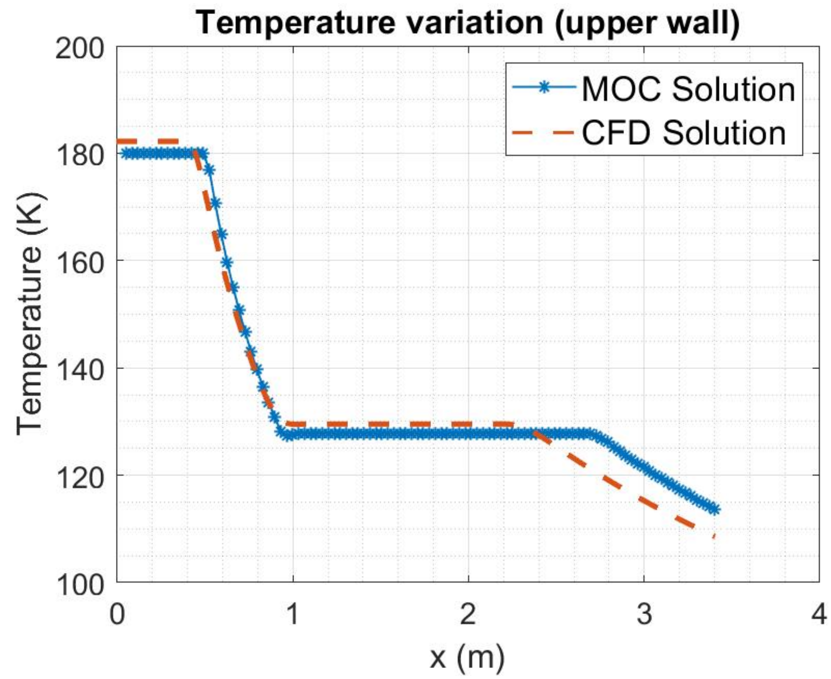


Fig. 24 Temperature variation at upper wall of the nozzle

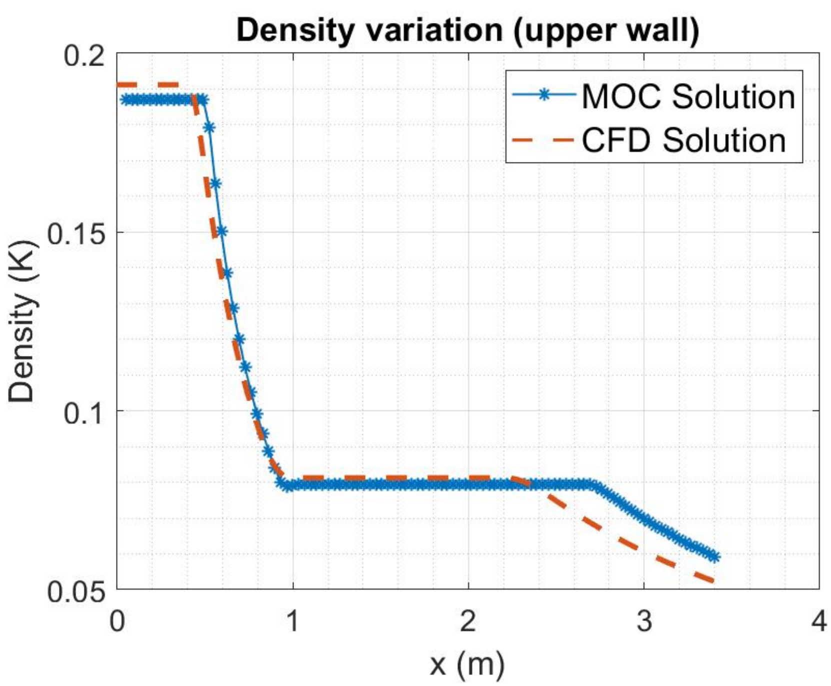


Fig. 25 Density variation at upper wall of the nozzle

The Mach variation on the upper wall could be seen from the figure 22. The variation in mach number should be constant till it encounters non-simple region on the upper wall, after which there should increase in the mach number in the non-simple region. The increase in the mach number is due to relation between θ and dM given in the equation 6. From the figure 22 it could be interpreted that in the uniform region mach was constant and the mach increases in the non-simple region. The simple, non-simple and uniform region could be seen from the figure 17. Thus the obtained results were consistent with the theoretical explanation, and shows good agreement with CFD results.

The Pressure variation on the upper wall could be seen from the figure 23. The variation in pressure should be constant till it encounters non-simple region on the upper wall, after which there should decrease in pressure in the non-simple region. The decrease in the pressure number is due to relation between θ and dP given in the equation 4 or it could also be seen from isentropic relation between P and M given in equation 1. From the figure 23 it could be interpreted that in the uniform region pressure was constant and the pressure value decreases in the non-simple region. The simple, non-simple and uniform region could be seen from the figure 17. Thus the obtained results were consistent with the theoretical explanation, and shows good agreement with CFD results.

The Temperature variation on the upper wall could be seen from the figure 24. The variation in temperature should be constant till it encounters non-simple region on the upper wall, after which there should decrease in pressure in the non-simple region. The decrease in the temperature number is due to isentropic relation between T and M given in equation 2. From the figure 24 it could be interpreted that in the uniform region temperature was constant and the temperature value decreases in the non-simple region. The simple, non-simple and uniform region could be seen from the figure 17. Thus the obtained results were consistent with the theoretical explanation, and shows good agreement with CFD results.

The Density variation on the upper wall could be seen from the figure 25. The variation in Density should be constant till it encounters non-simple region on the upper wall, after which there should decrease in pressure in the non-simple region. The decrease in the Density number is due to isentropic relation between ρ and M given in equation 3. From the figure 25 it could be interpreted that in the uniform region temperature was constant and the temperature value decreases in the non-simple region. The simple, non-simple and uniform region could be seen from the figure 17. Thus the obtained results were consistent with the theoretical explanation, and shows good agreement with CFD results.

At lower wall of the duct

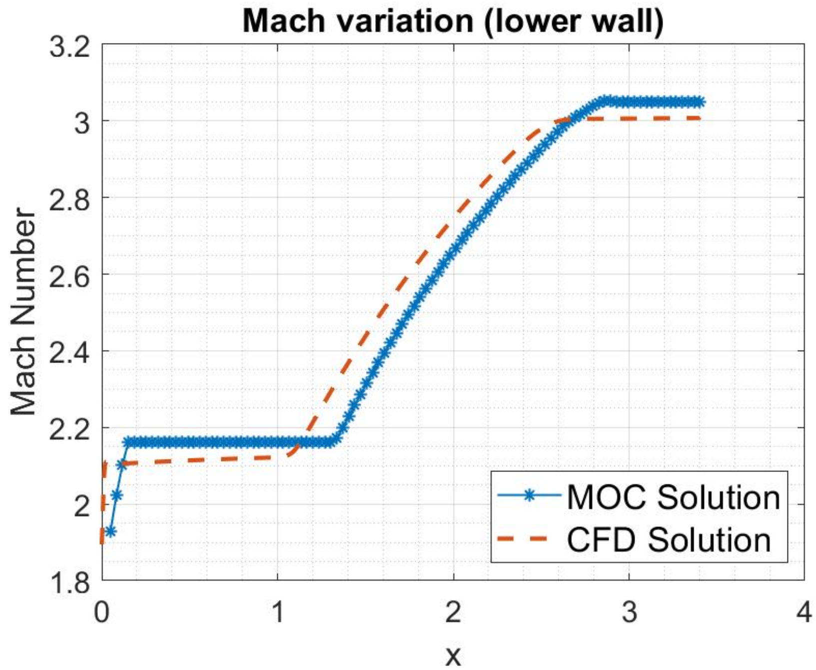


Fig. 26 Mach variation at lower wall of the nozzle

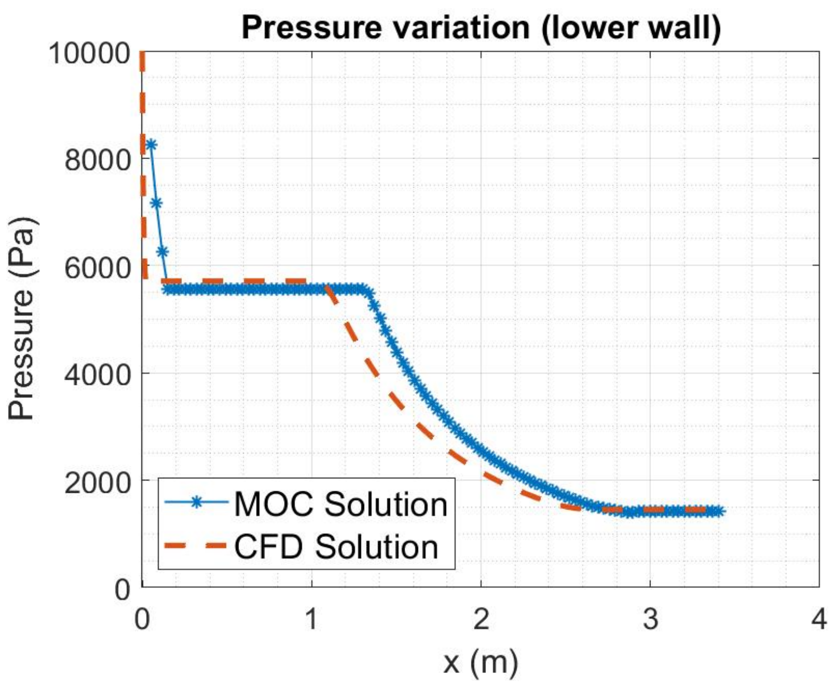


Fig. 27 Pressure variation at lower wall of the nozzle

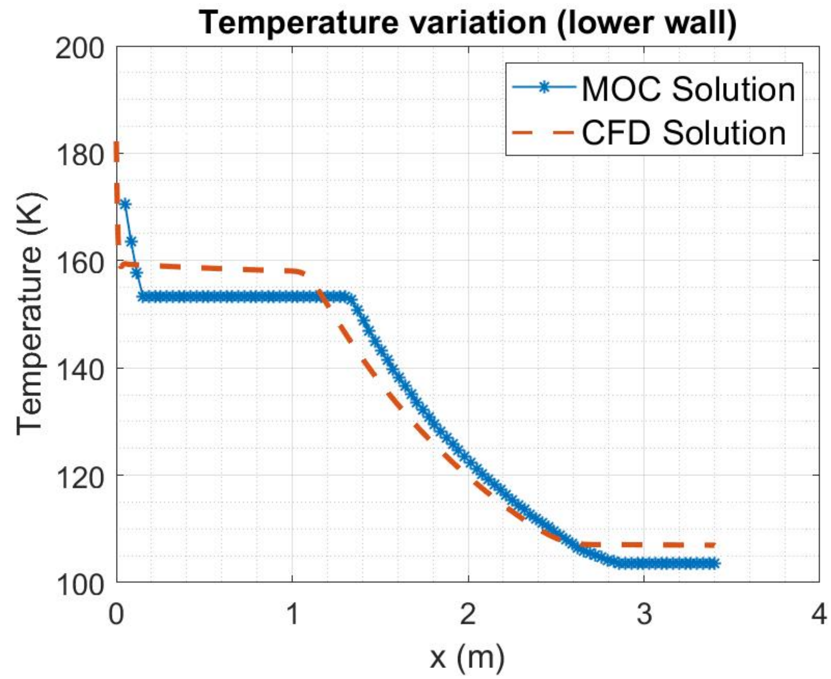


Fig. 28 Temperature variation at lower wall of the nozzle

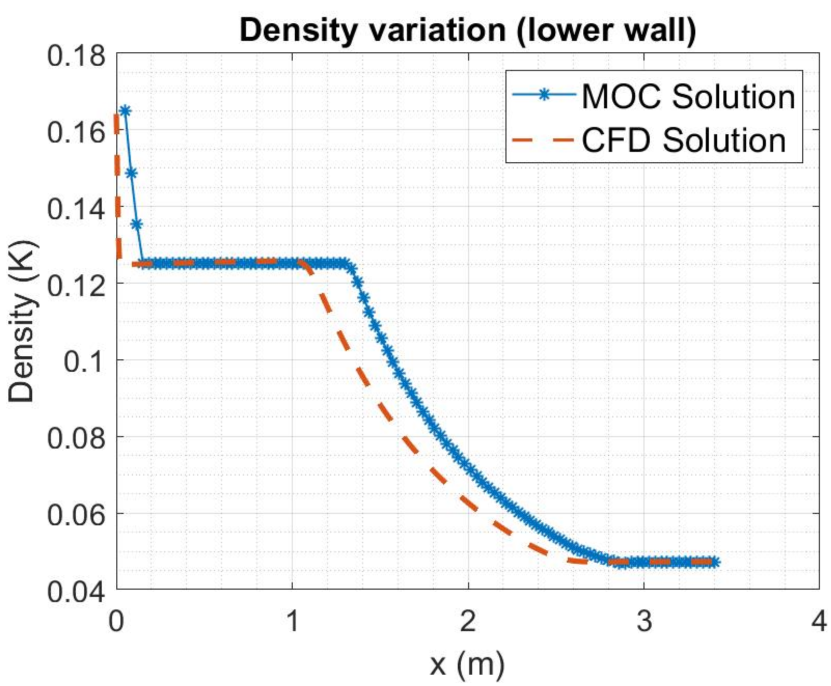


Fig. 29 Density variation at lower wall of the nozzle

From the figure 26 it can be observed that Mach number first increases upto certain distance. This is due to the presence of expansion waves at the corner point. After this point the Mach number becomes constant which is due to the presence of uniform flow at the lower wall. This can be observed from the figure 17. After this point, we can observe from figure 26 that there is an increase in Mach number which is due to presence of non-simple region formed at the lower wall. In this region the Mach number is increasing. This increase in the Mach number is in accordance with the equation 6. After this point again a uniform region is formed and again a constant change in Mach number is observed in the graph. From the figure 26 it can be observed that the result obtained from MOC and ANSYS are nearly same with very slight deviation.

Figure 27 shows the variation of pressure at the lower wall. Similar to the Mach number here also different regions are formed at the lower wall because of which there is a variation in the pressure. At the uniform region the pressure is becoming constant. At expansion and non-simple region the pressure decreases. This variation is in accordance with the equation 4. The various region can be observed from the figure 17. From figure 27 it can be observed that the result obtained from MOC is near similar to the result obtained from ANSYS. From the plot only it can be observed that MOC and ANSYS results are very close.

Figure 28 shows the variation of temperature at the lower wall. Similar to the Mach number and pressure here also different regions are formed at the lower wall because of which there is a variation in the temperature. At the uniform region the temperature is becoming constant. At expansion and non-simple region the temperature decreases. This variation is in accordance with the equation 2. The various region can be observed from the figure 17. From figure 28 it can be observed that the result obtained from MOC is near similar to the result obtained from ANSYS. From the plot only it can be observed that MOC and ANSYS results are very close.

Figure 28 shows the variation of density at the lower wall. Similar to the Mach number and pressure here also different regions are formed at the lower wall because of which there is a variation in the density. At the uniform region the density is becoming constant. At expansion and non-simple region the density decreases. This variation is in accordance with the equation 5. The various region can be observed from the figure 17. From figure 29 it can be observed that the result obtained from MOC is near similar to the result obtained from ANSYS. From the plot only it can be observed that MOC and ANSYS results are very close.

At outlet of the duct

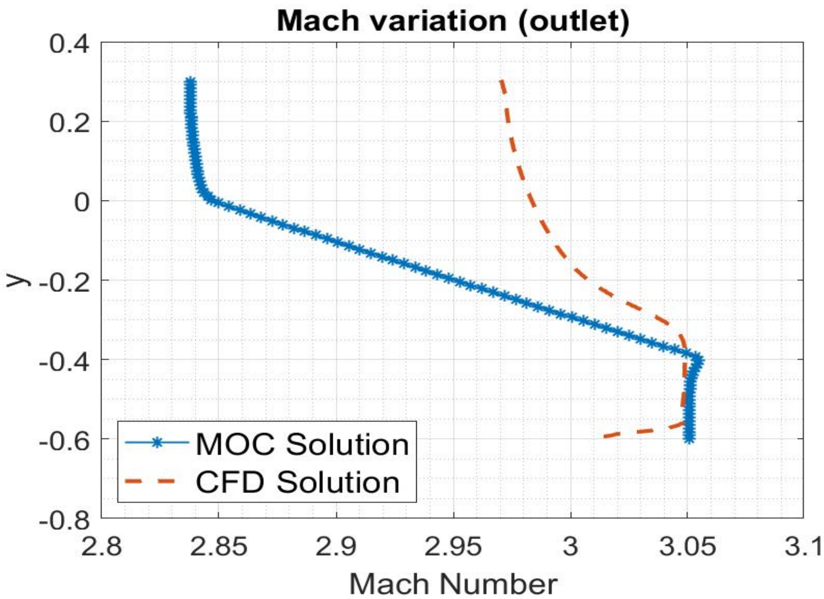


Fig. 30 Mach variation at outlet of the nozzle

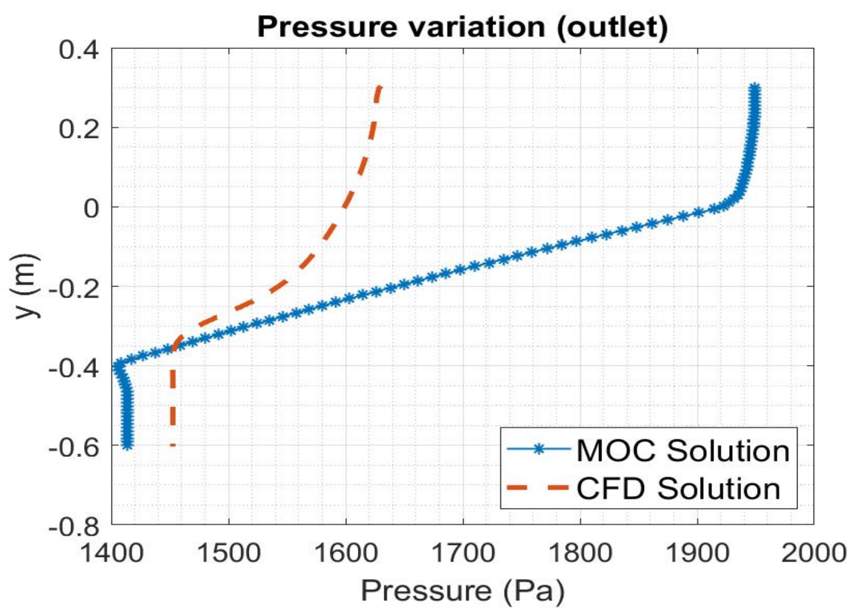


Fig. 31 Pressure variation at outlet of the nozzle

The difference in results of MoC and CFD is arising at the outlet. As the density of nodes is less when compared to that at the inlet of the nozzle in MoC. Due to less number of nodes the interpolation (adopted for extracting the results at different sections) leads to approximation errors. This error doesn't arise in region away from the confirming that the error is due is due to less density of nodes only. Although there is error but the shape of the curve is similar for both MoC and CFD results.

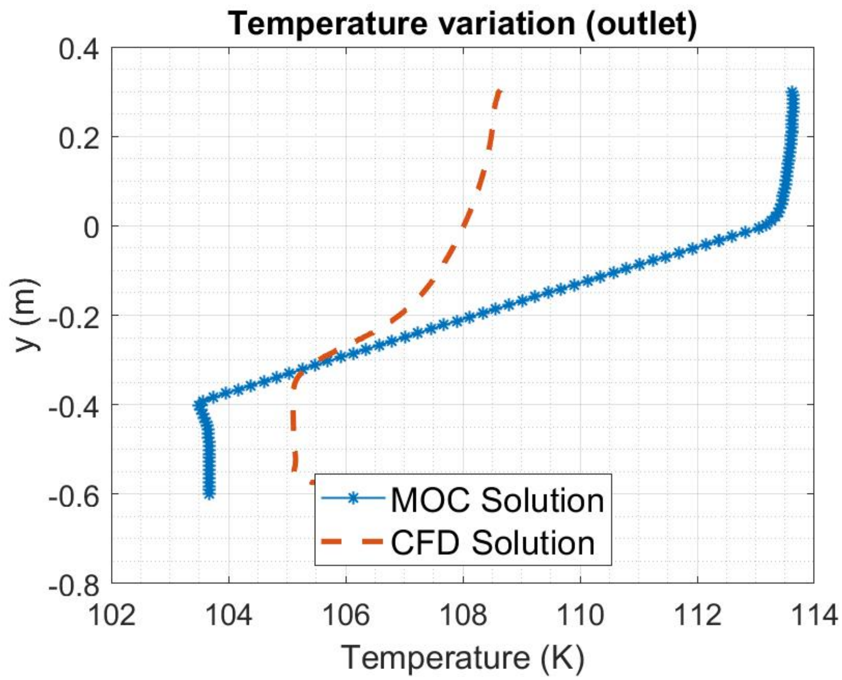


Fig. 32 Temperature variation at outlet of the nozzle

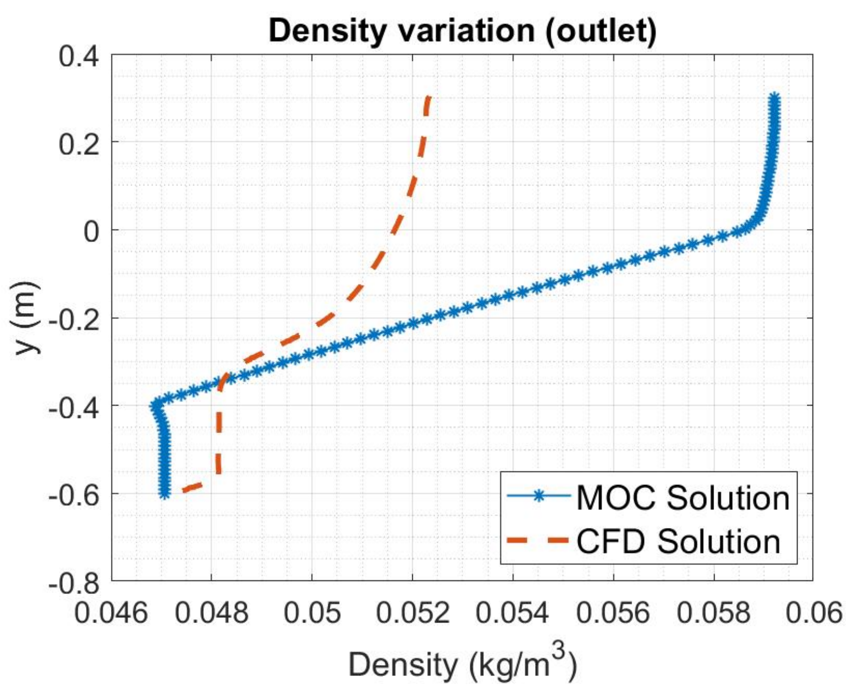


Fig. 33 Density variation at outlet of the nozzle

Contour Plots

Contour Plot obtained using MOC

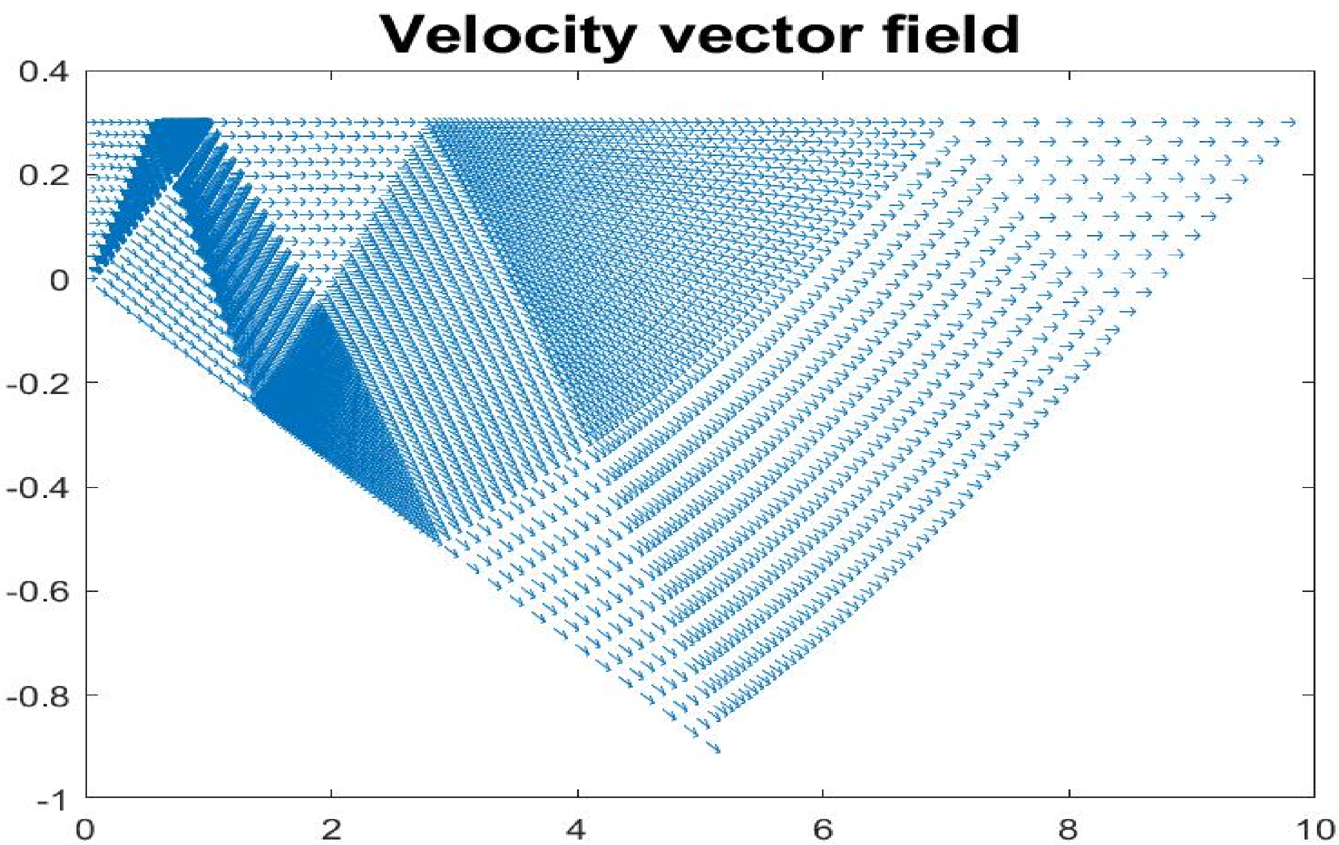


Fig. 34 Velocity vector plot using MOC

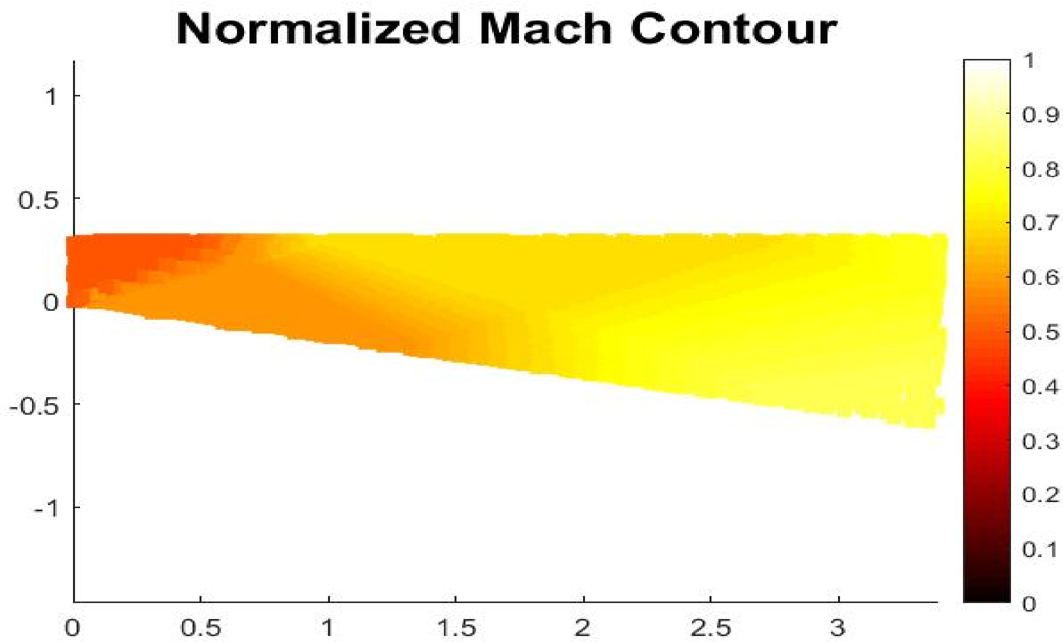


Fig. 35 Normalized mach contour plot using MOC

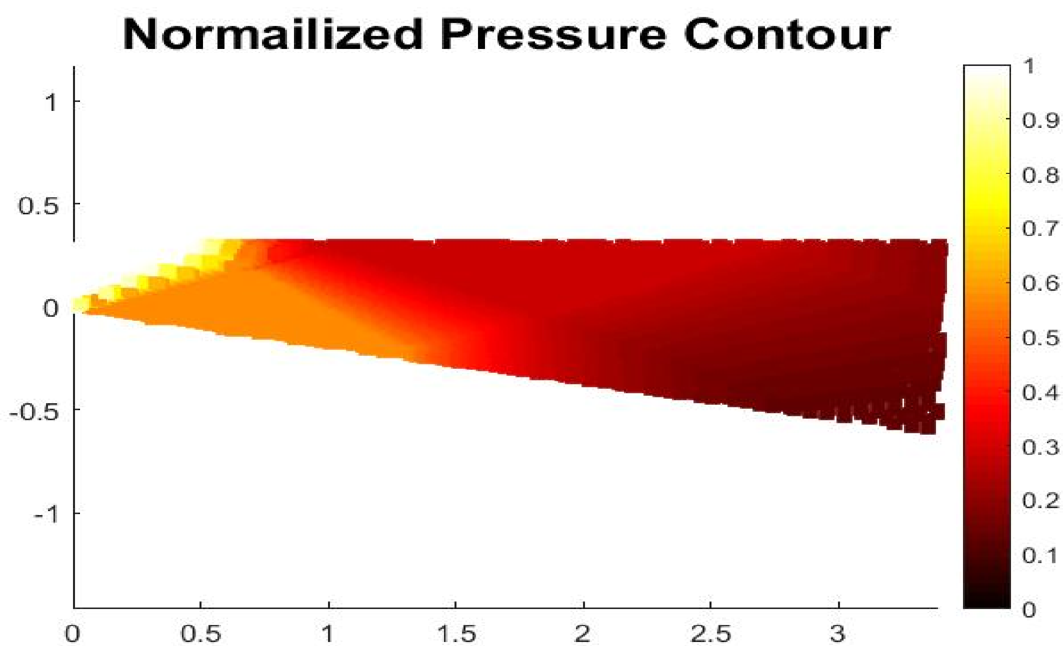


Fig. 36 Normalized pressure contour plot using MOC

Contour Plot obtained using CFD

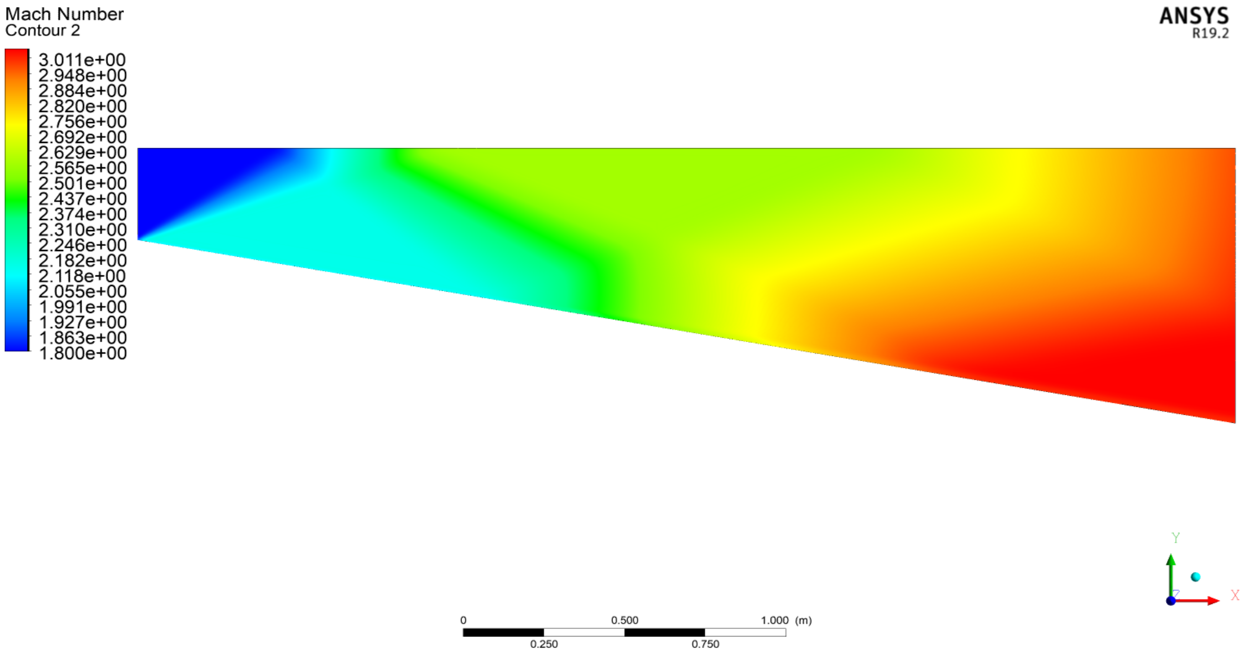


Fig. 37 Mach contour in ansys for same inlet conditions

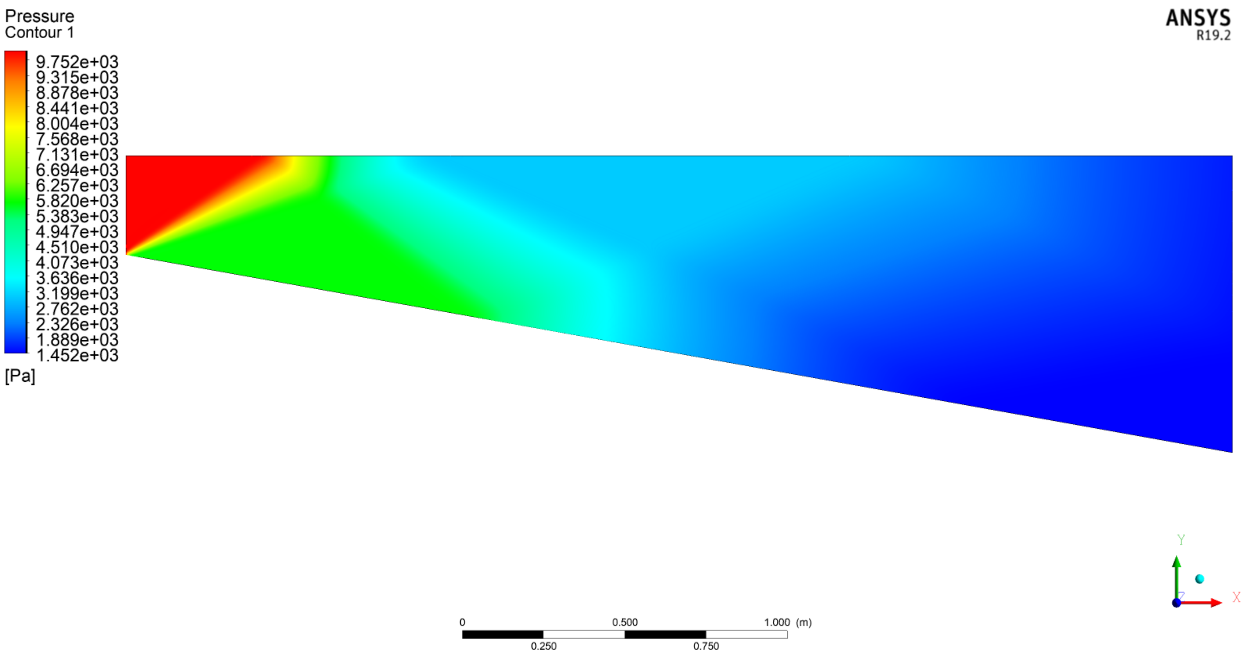


Fig. 38 Pressure contour in ansys for same inlet conditions

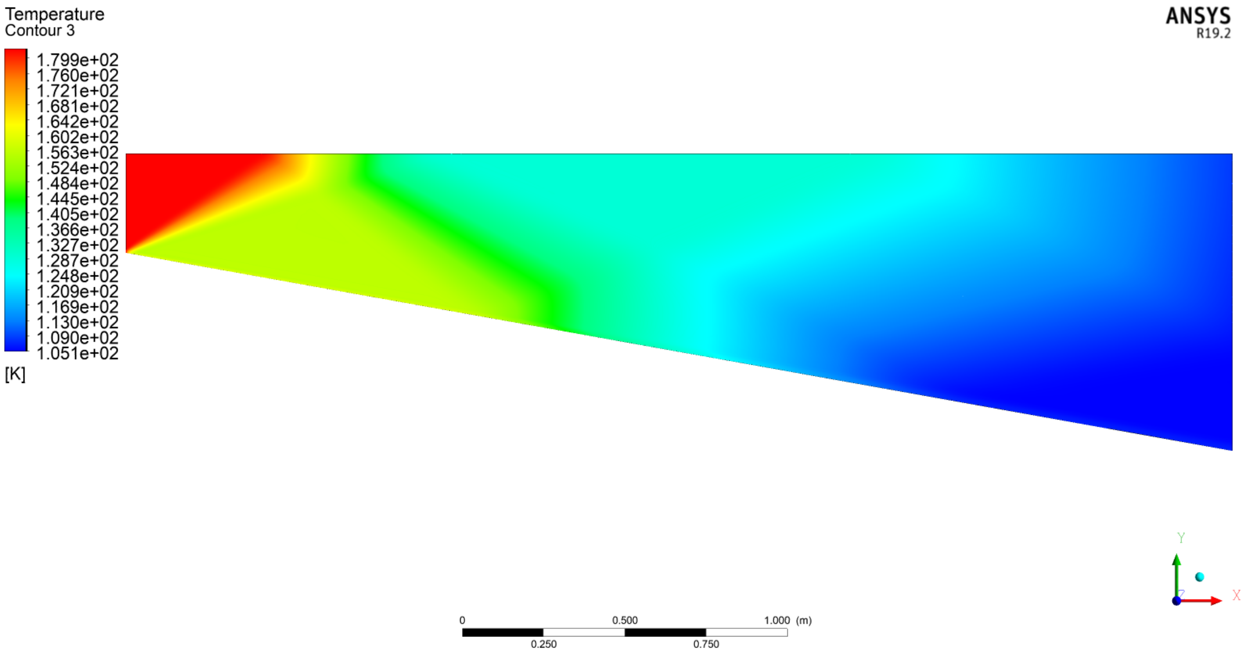


Fig. 39 Temperature contour in ansys for same inlet conditions

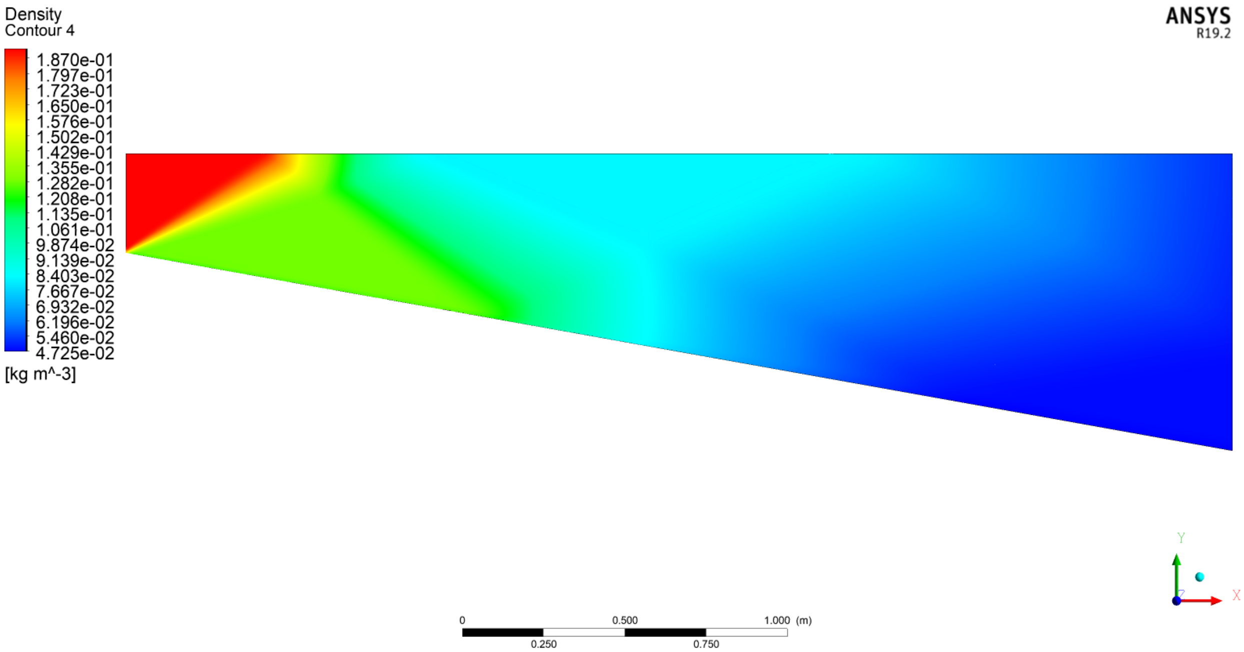


Fig. 40 Density contour in ansys for same inlet conditions

II. Curved and Smooth Walls

Characteristic grid network in channel

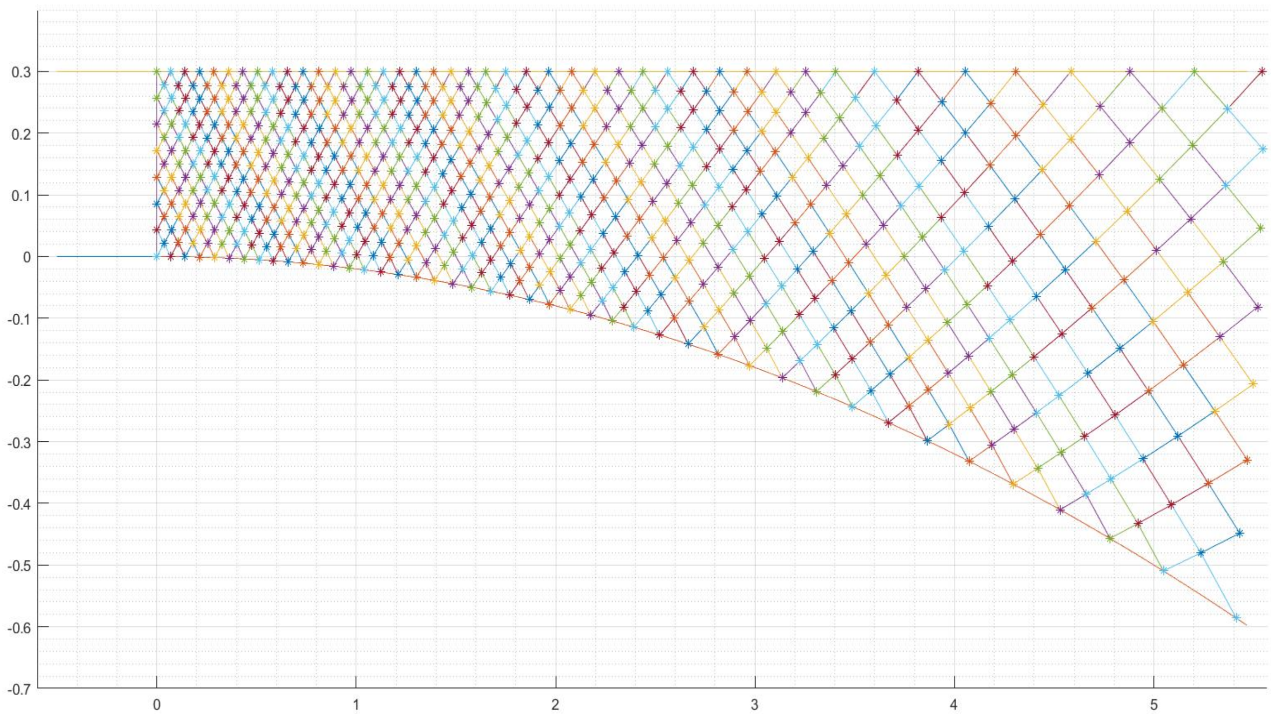


Fig. 41 Characteristic grid network plot for curved wall

Matlab and CFD Results

The results of CFD and MoC almost coincide with each other. The difference arises at the outlet where the density of nodes is less when compared to that at the inlet of the nozzle in MoC. Due to less number of nodes the interpolation (adopted for extracting the results at different sections) leads to approximation errors.

Mach

The variation of mach number at y = 0.15 m of the nozzle is shown in figure **??**. It shows that overall mach number increases along the direction of flow. This is because whenever the flow crosses the expansion fan its mach number increases, as according to equation 6, for $d\theta$ being negative, dM should be positive to satisfy the equation. The mach number in the figure is constant in some region as in those regions the flow doesn't encounter any expansion fan.

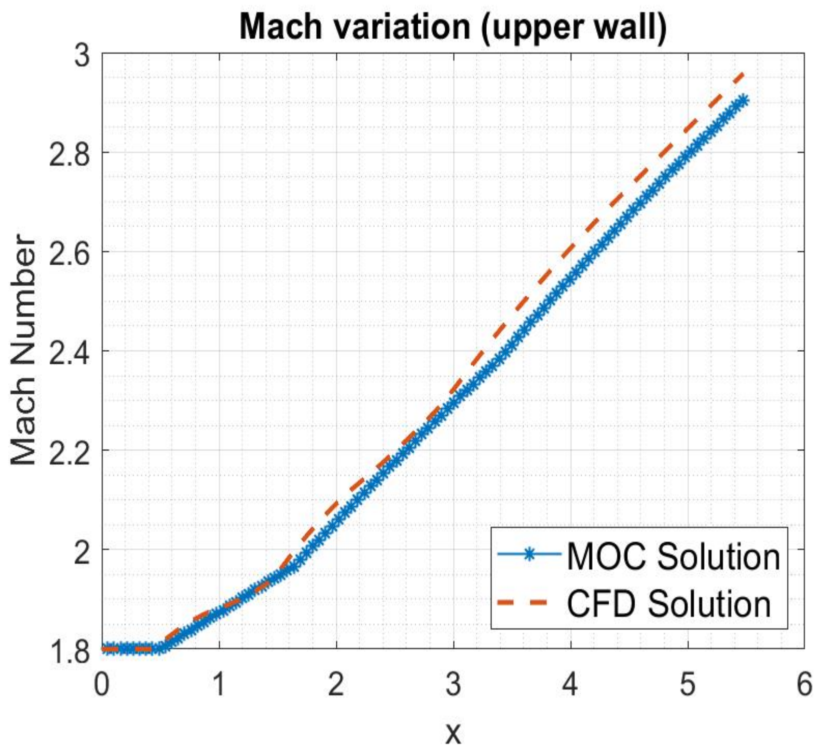


Fig. 42 Mach variation at upper wall of the nozzle

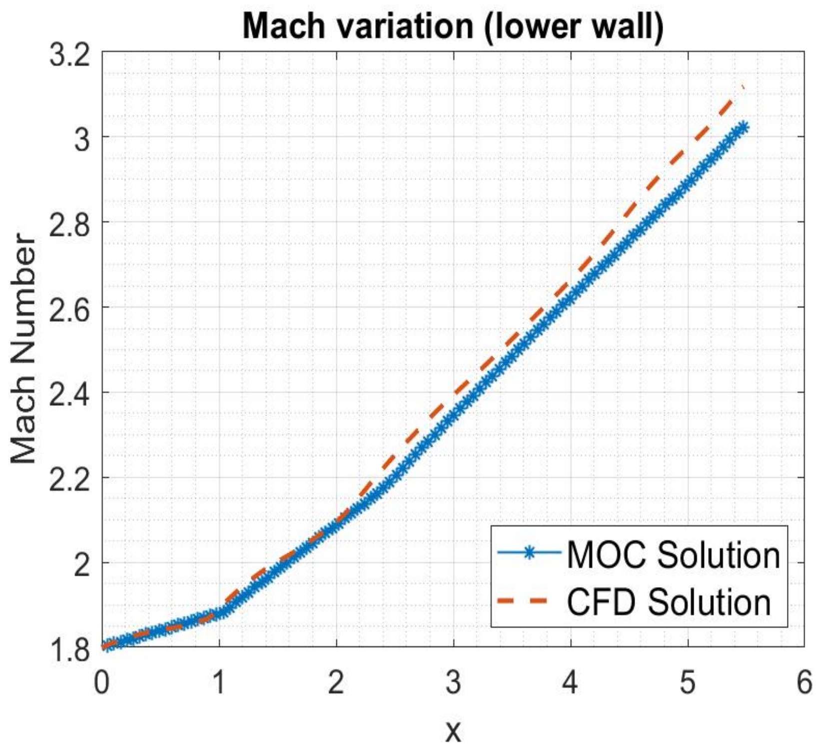


Fig. 43 Mach variation at lower wall of the nozzle

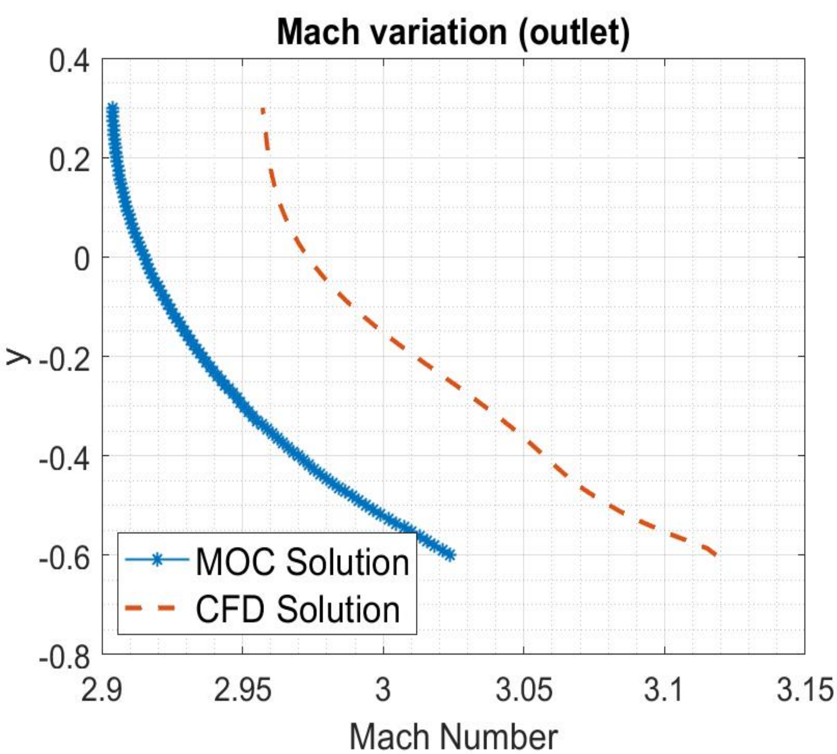


Fig. 44 Mach variation at outlet of the nozzle

Contour Plots

Velocity Vector Plot

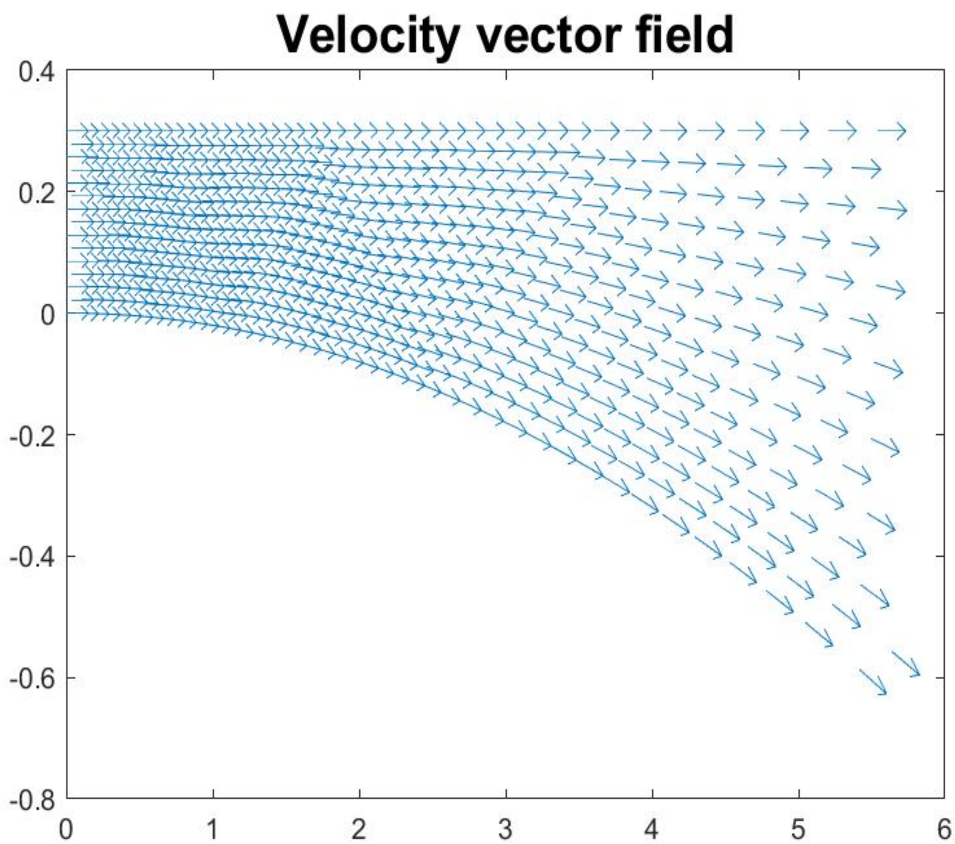


Fig. 45   Mach contour in Ansys for same inlet conditions

Contour plot obtained using CFD

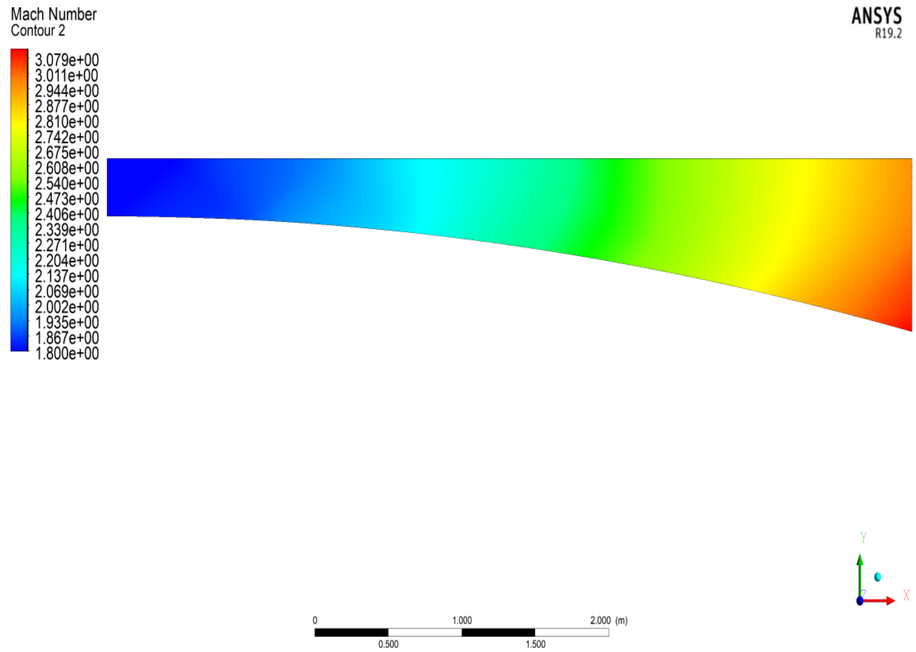


Fig. 46 Mach contour in Ansys for same inlet conditions

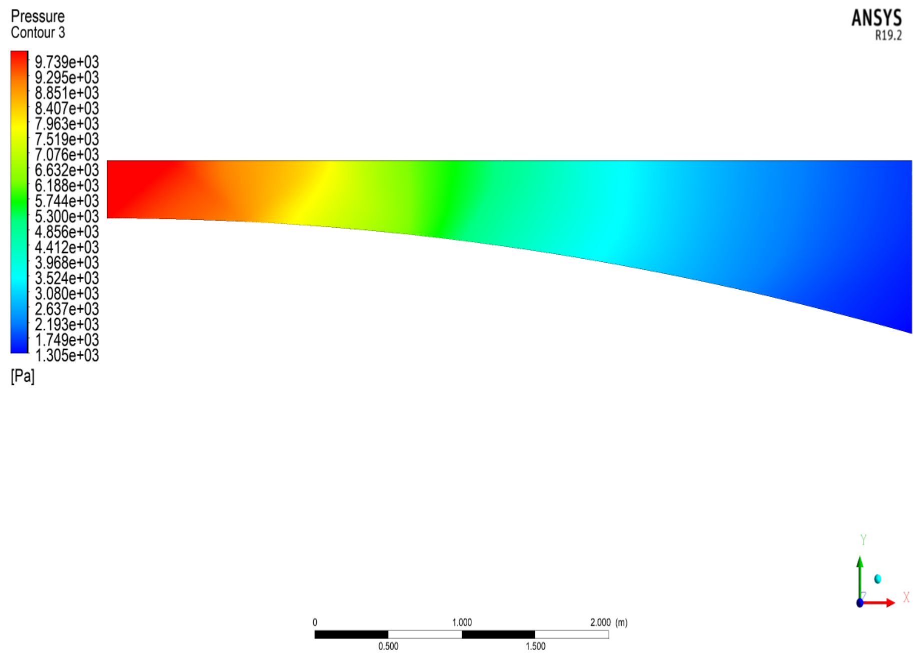


Fig. 47 Pressure contour in Ansys for same inlet conditions

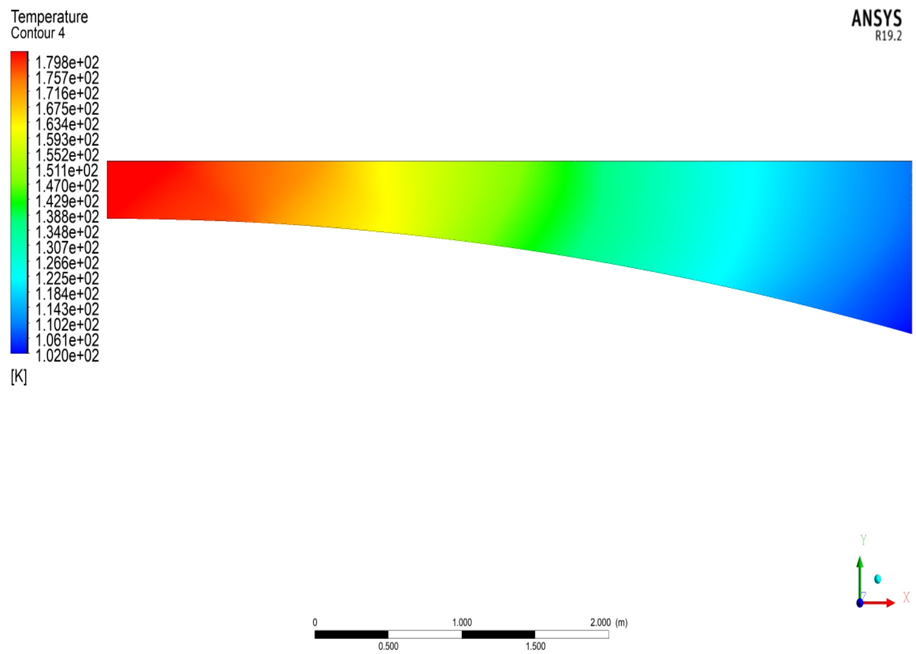


Fig. 48 Temperature contour in ansys for same inlet conditions

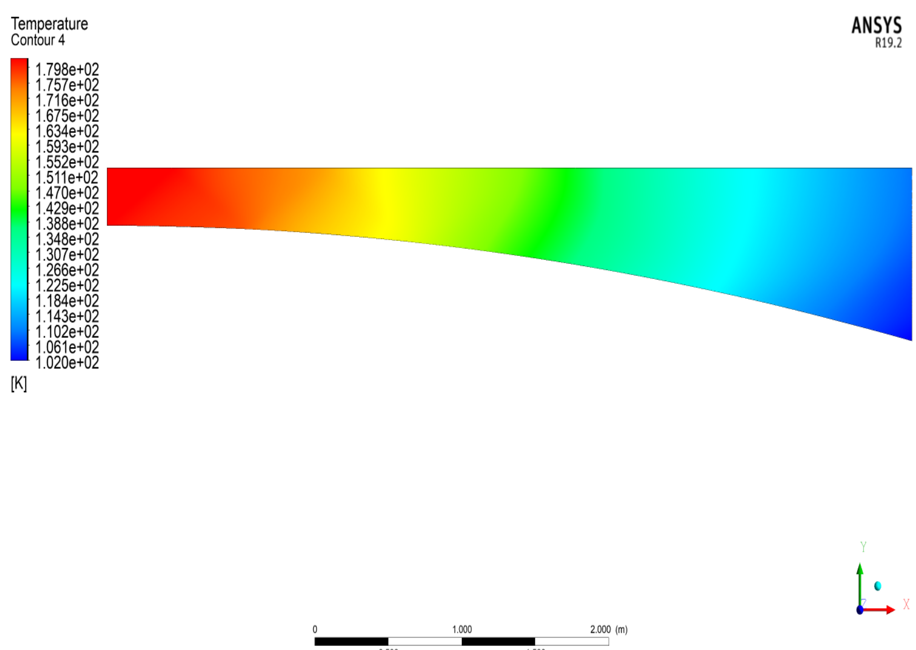


Fig. 49 Density contour in ansys for same inlet conditions

Run Time Analysis

This is run time chart along with result generation without saving the result plots.

Function Name	Calls	Total Time	Self Time*	Total Time Plot (dark band = self time)
trial	1	24.785 s	14.561 s	
xlsread	24	4.068 s	0.029 s	
iofun\private\xlsreadCOM	24	3.257 s	0.114 s	
newplotwrapper	6828	2.482 s	0.450 s	
iofun\private\openExcelWorkbook	24	2.125 s	1.988 s	
newplot	6829	2.039 s	0.646 s	
legend	24	1.686 s	0.028 s	
legend>make_legend	24	1.655 s	0.047 s	
fsolve	207	0.906 s	0.154 s	
getExcelInstance	24	0.761 s	0.027 s	
Legend.Legend>Legend.Legend	48	0.759 s	0.180 s	
gobjects	13658	0.688 s	0.688 s	
actxserver	1	0.680 s	0.680 s	
upper_wall_intersection	131	0.655 s	0.026 s	
onCleanup>onCleanup.delete	49	0.555 s	0.002 s	
...)xlsCleanup(Excel,file,workbookState)	24	0.550 s	0.003 s	
iofun\private\xlsCleanup	24	0.546 s	0.545 s	

Fig. 50 Run time Analysis for Matlab code

References

[1] Jegede, O. O., and Crowther, W. J., "Low order supersonic nozzle design using superimposed characteristics," *54th AIAA Aerospace Sciences Meeting*, 2016, p. 0805.

[2] Zhu, C., Qiu, R., Chen, R., and You, Y., "Inverse method of characteristics with given exit parameters for the computation of supersonic flows," *21st AIAA International Space Planes and Hypersonics Technologies Conference*, 2017, p. 2307.

[3] Yang, C., Li, X., and Zhang, C., "Numerical study of two-dimensional cylindrical underwater explosion by a modified method of characteristics," *Journal of Applied Physics*, Vol. 122, No. 10, 2017, p. 105903.

[4] Liepmann, H., and Roshko, A., *Elements of Gas Dynamics*, Dover Books on Aeronautical Engineering, Dover Publications, 2013. URL `https://books.google.co.in/books?id=IWrCAgAAQBAJ`.

[5] John, J., and Keith, T., *Gas Dynamics*, Pearson Prentice Hall, 2006. URL `https://books.google.co.in/books?id=VXgsAAAAYAAJ`.

[6] Anderson, J., *Modern Compressible Flow: With Historical Perspective*, Aeronautical and Aerospace Engineering Series, McGraw-Hill Education, 2003. URL `https://books.google.co.in/books?id=woeqa4-a5EgC`.