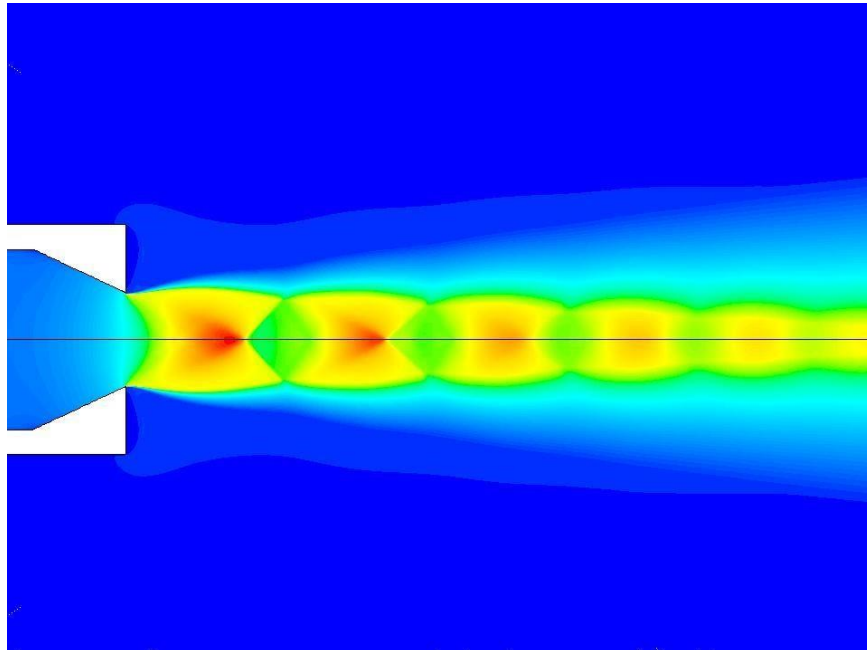


**SPC 407**  
**Supersonic & Hypersonic Fluid Dynamics**  
**Ansys Fluent Tutorial 4**

**Compressible Flow through Convergent Conical**  
**Nozzle**

---



## Introcuccion

The accurate numerical prediction of nozzle flows can be a valuable tool in the evaluation of designs of propulsion systems of a turbojet aircraft. Broad-spectrum, easy-to-implement numerical models of turbulent subsonic, transonic and supersonic flows from turbojet engine exhaust nozzles are being sought to augment the design process of nozzles, which are often of a convergent-conical type, operating at various nozzle pressure ratios. The flow of an imperfectly expanded supersonic jet exhausted from a convergent conical nozzle is also complicated by the presence of a quasi-periodic shock-cell structure that can significantly affect radiated jet noise. It is essential for the analysis to accurately resolve shock-cell structures of supersonic jets for a comprehensive exhaust flow analysis including jet expansion, mixing, and associated jet noise.

This tutorial discusses the pressure-based coupled algorithm implemented in the general purpose CFD code ANSYS Fluent, and evaluates its effectiveness in solving axisymmetric problems of steady and transient flows through convergent conical nozzles at different nozzle pressure ratios.

A pressure-based coupled solver formulation with weighted second-order central-upwind spatial discretizations (QUICK scheme) is applied to calculate the numerical solutions. A hierarchy of axisymmetric hybrid computational meshes is constructed to evaluate grid independence. Effects of turbulence modeling are evaluated by comparing SST  $k-\omega$  and Renormalization Group (RNG)  $k-\varepsilon$ . Numerical predictions of discharge and thrust coefficients, and Mach numbers inside and at the nozzle exit are compared with the experimental data and excellent agreements are presented.

## APPLICATIONS

Aircraft propulsion.

## KEY WORDS

CFD, nozzle, propulsion, compressible flow

# **Outline**

1. Problem description
2. Numerical model
  - Pressure-based coupled solver
  - Physical models and boundary conditions
  - Computational mesh
3. Numerical results and comparison with test data
  - Steady-state axisymmetric results
  - Transient solution of vortex shedding off the splitter trailing edge
4. Summary

# Problem Description

Benchmark problem of the **1<sup>st</sup> AIAA Propulsion Aerodynamics Workshop** (2012)

- nozzle geometry and flow conditions from experiments of Thornock and Brown<sup>1</sup>



## **Axisymmetric nozzle:**

- 25 deg cone half angle
  - Nozzle exit diameter = 3.0 inches
  - Conditions correspond to a stationary nozzle discharging into a quiescent ambient air at standard sea level ambient pressure (14.696 psi)
  - Cold jets with total temperature and ambient static temperatures of 518.7 R
  - Nozzle pressure ratio (NPR) varied from 1.4 to 7.0
    - fully expanded jet Mach numbers are subsonic for NPRs 1.4 to 1.8 and supersonic for NPRs 2.0 to 7.0.
  - Nozzle coefficients of thrust,  $C_v$ , and discharge,  $C_d$ , are determined on the basis of axisymmetric flow computations
  - Noise radiation by an axisymmetric supersonic jet at NPR = 2.5 is considered
-

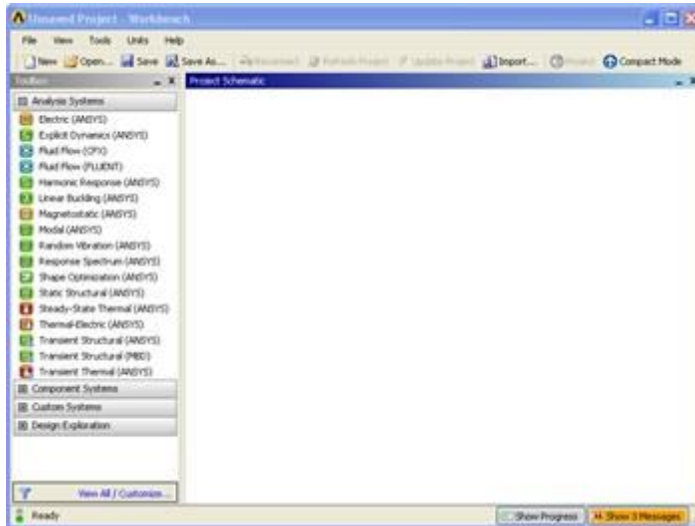
## **References (included in tutorial Zip File)**

1. Thornock, R. L., and Brown, E. F., "An Experimental study of Compressible Flow Through Convergent-Conical Nozzles, Including a Comparison With Theoretical Results," Transactions of the ASME: J. Basic Engineering, 1972, pp. 926-932.
2. Vance F. Dippold, NASA Report, "Computational Simulations of Convergent Nozzles for the AIAA 1st Propulsion Aerodynamics Workshop", 2014

# Simulation Steps

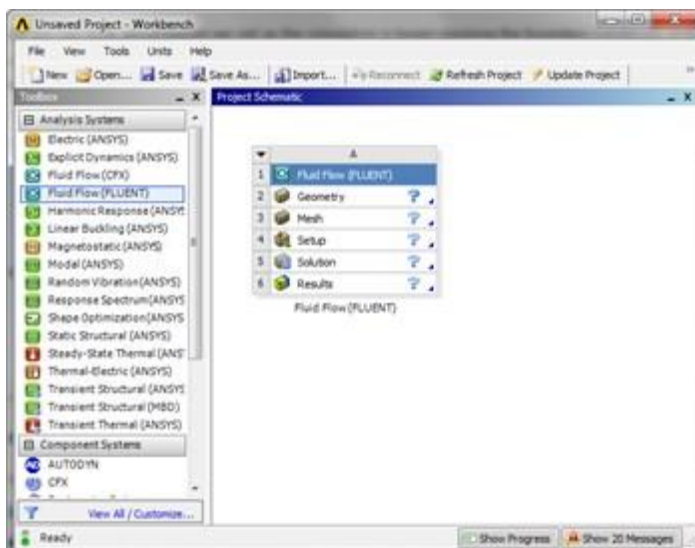
## Open ANSYS Workbench

We are ready to do a simulation in ANSYS Workbench! Open ANSYS Workbench by going to Start > ANSYS > Workbench. This will open the startup screen seen as seen below




## Setup Project

To begin, we need to tell ANSYS what kind of simulation we are doing. If you look to the left of the start up window, you will see the Toolbox Window. Take a look through the different selections. We will be using FLUENT to complete the simulation. Load the **Fluid Flow (FLUENT)** box by dragging and dropping it into the Project Schematic.



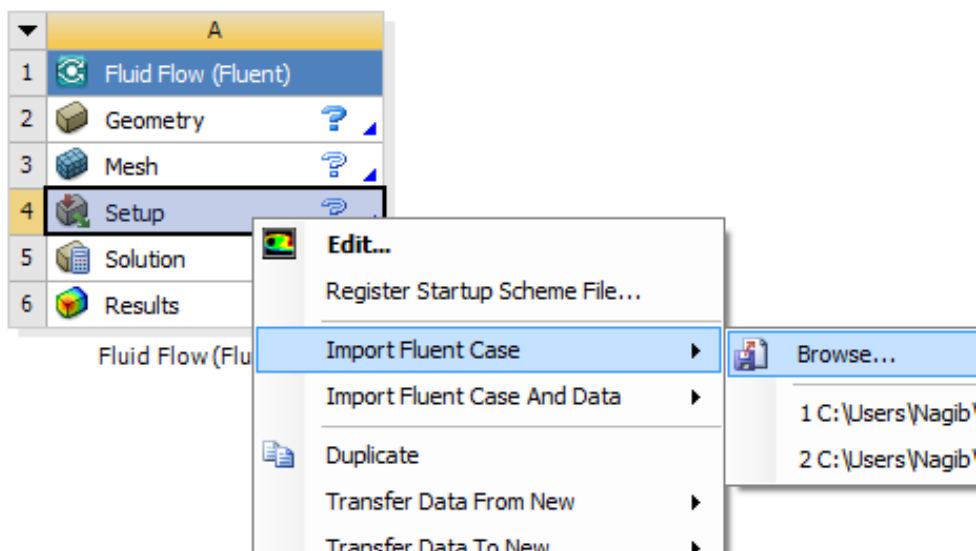
Right click the top box of the project schematic

1  Fluid Flow (FLUENT)

and go to **Rename**, and name the project “Conical Divergent Nozzle”. You are ready to import the case file which include the geometry and the mesh.

## Set Up Problem in FLUENT

Right Click on cell “A4” setup >>Import Fluent Case >> Browse>> locate Tut\_4\_case\_file.cas file (located at the same folder of downloaded files)

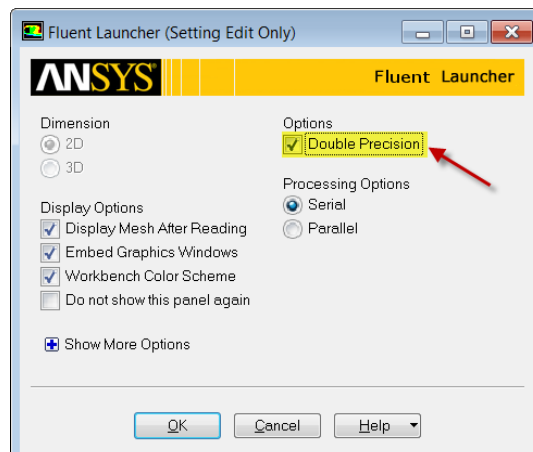


When the project updates, double click Setup  Setup to open FLUENT.

## Initial Settings

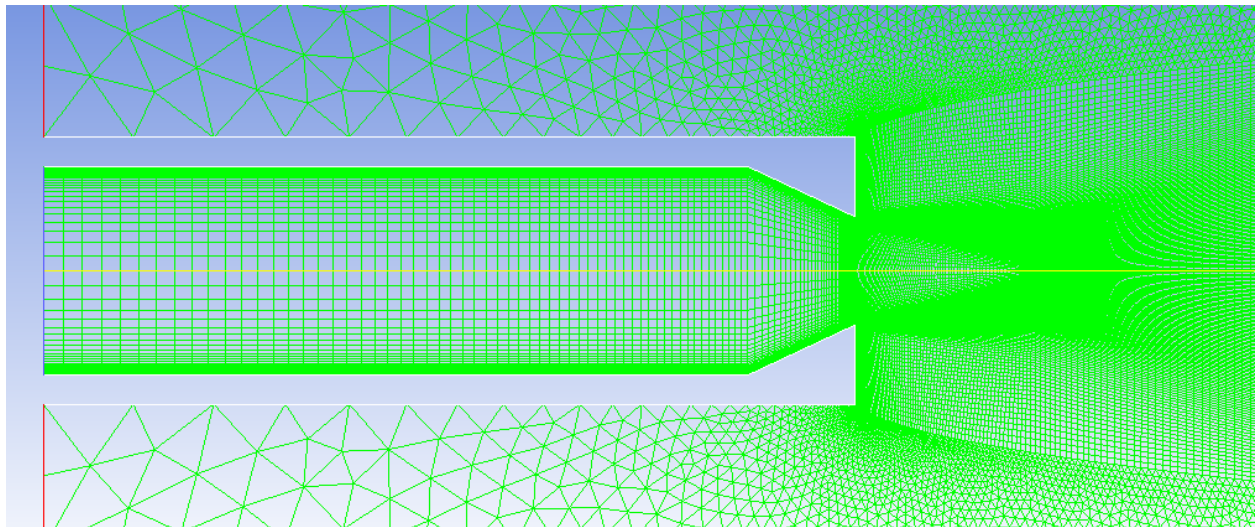
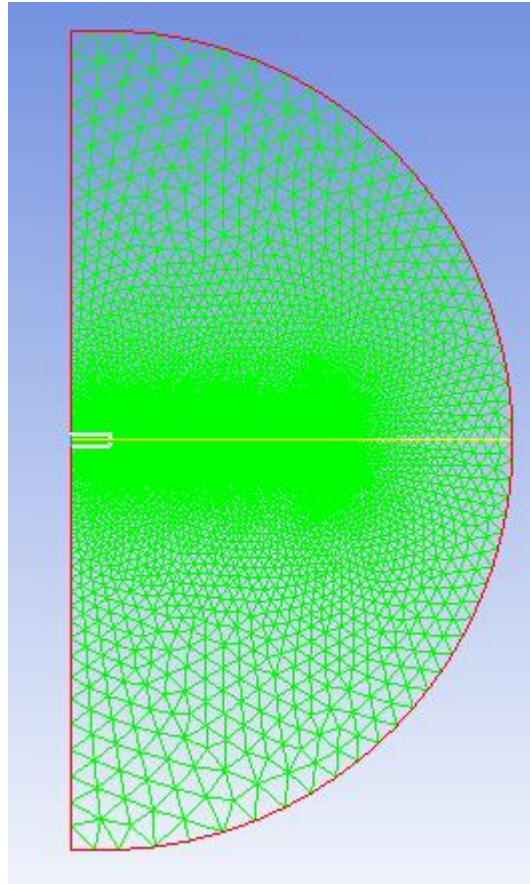
(Double Click) Setup in the *Workbench Project Page*.

When the **FLUENT Launcher** appears change options to "Double Precision", and then click **OK** as shown below.



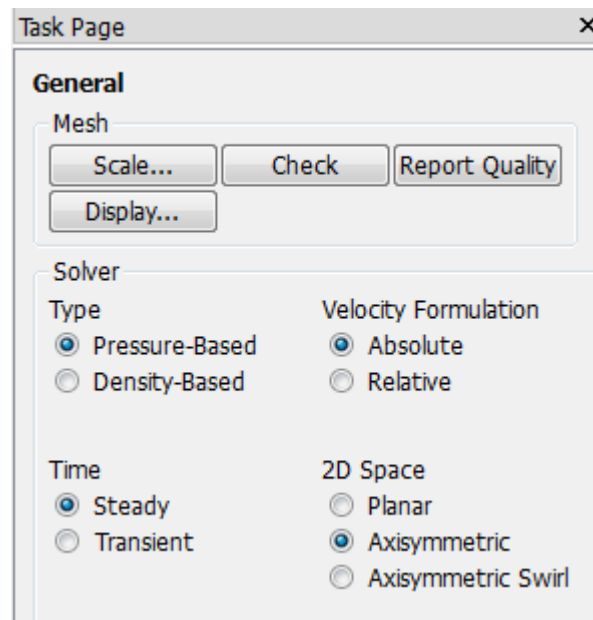
### **Problem Setup - Fluent**

Now, FLUENT should open. We will begin setting up some options for the solver. The mesh should be similar to the two figures below



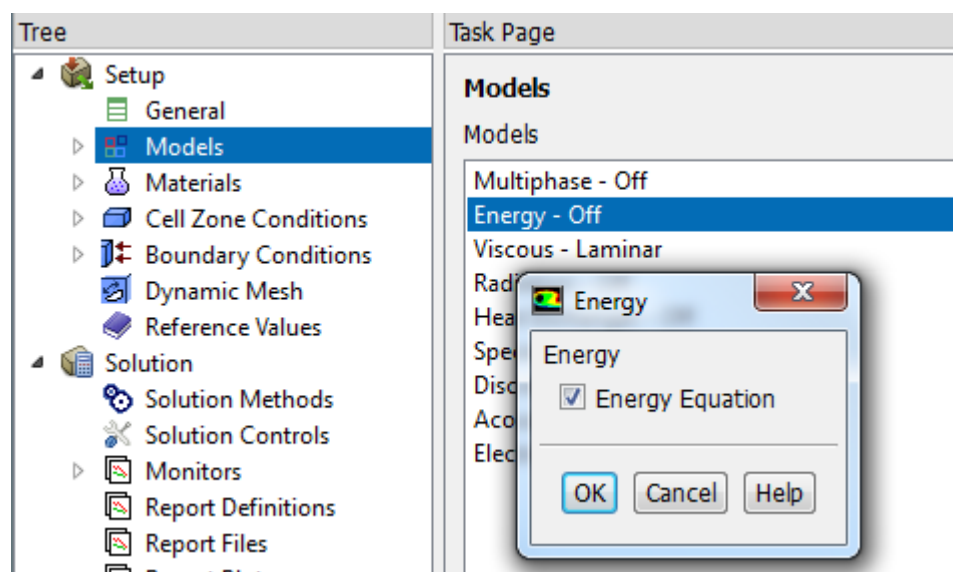


In the left hand window (in what I will call the *Outline* window), under *Problem Setup*, select *General*. The only option we need to change here is the type of 2D space. In the *Solver* window, select *Axisymmetric*.

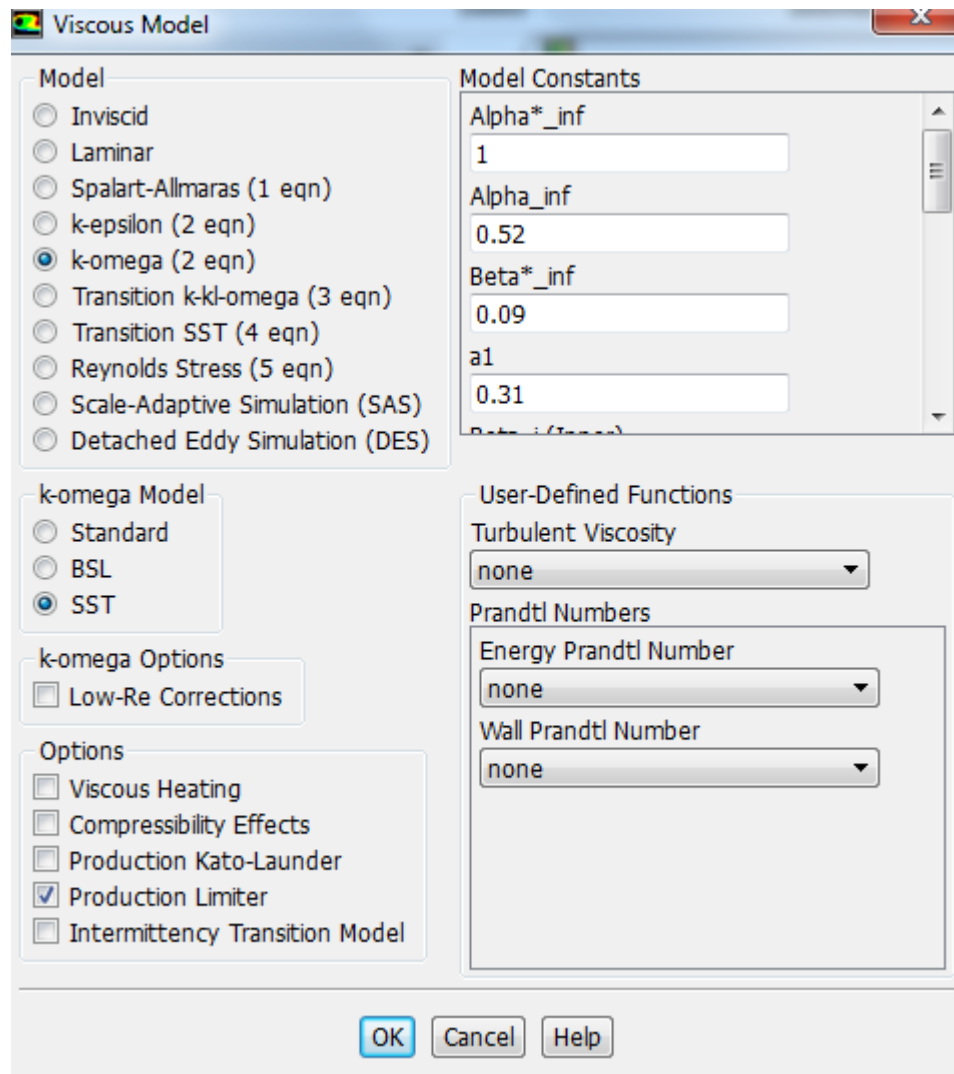


## Models

In the outline window, click *Models*. We will need to utilize the energy equation in order to solve this simulation. Under *Models* highlight *Energy-Off* and click *Edit*. Now, the *Energy* window will launch. Check the box next to *Energy Equation* and hit OK. Doing this turns on the energy equation.



We also need to change the type of viscosity model. Select *Viscous - Laminar* and click *Edit...* Choose the *k-omega and SST under K-omega model* option and press *OK*.



## **Materials**

In the *Outline* window, highlight *Materials*. In the *Materials* window, highlight *Fluid*, and click *Create/Edit...* this will launch the *Create/Edit Materials* window; here we can specify the properties of the fluid. Set the *Density* to *Ideal Gas*, *Viscosity* to *Sutherland* and the default values for  $C_p$  (1006.43), and the Molecular Weight (28.966) are used. When you have updated these fields, press *Change/Create*.

Create/Edit Materials

Name	air	Material Type	fluid
Chemical Formula		Fluent Fluid Materials	air
		Mixture	none

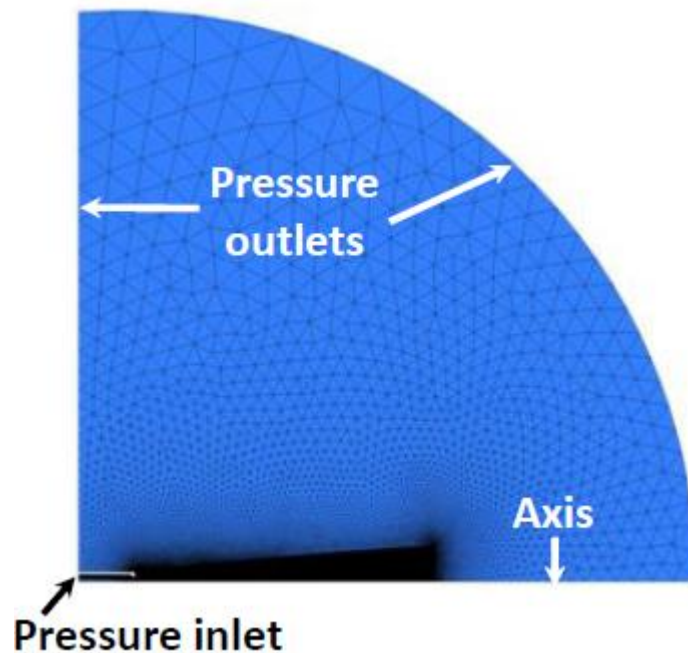
Properties

Density (kg/m3)	ideal-gas	Edit...
Cp (Specific Heat) (j/kg-k)	constant	Edit...
	1006.43	
Thermal Conductivity (w/m-k)	constant	Edit...
	0.0242	
Viscosity (kg/m-s)	sutherland	Edit...

- Air modeled as single-species ideal gas
- Molecular viscosity defined as a function of temperature by Sutherland's viscosity law

## **Boundary Conditions**

In the *Outline* window, select *Boundary Conditions*. We will now specify each boundary condition for the simulation.



*Pressure inlet condition at the nozzle inlet*

*static and total pressure, and total temperature*

*Pressure outlet at the ambient boundaries*

*specifies  $p_s$  if  $M_{local} < 1.0$ , extrapolates  $p_s$  if  $M_{local} > 1.0$*

*Turbulence models*

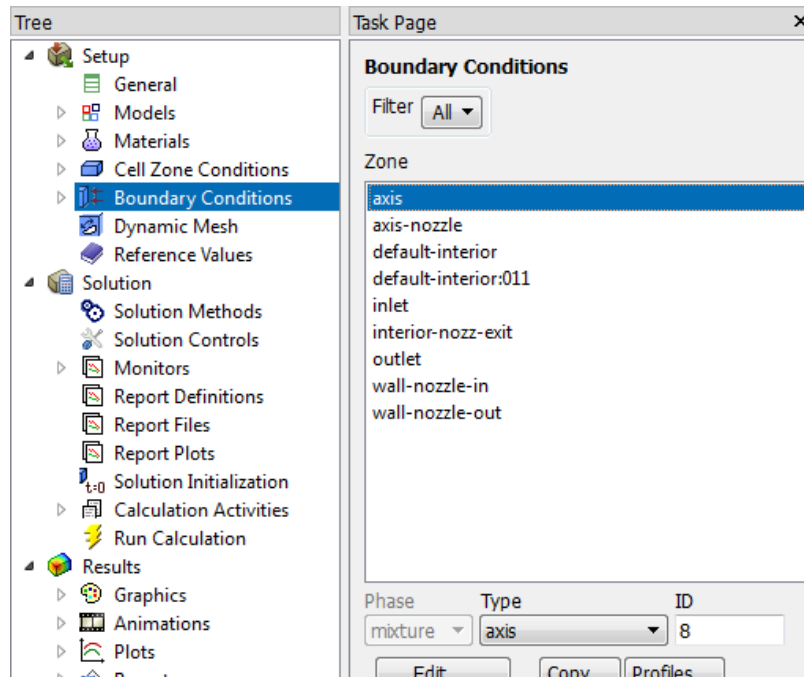
*SST  $k-\omega$*

*Adiabatic no-slip walls*

*$k$  and  $\omega$  at nozzle inlet derived from turbulence intensity  $I = 1.0\%$  and inlet diameter  $D$ ,  $k = (3/2)(U \cdot I)^2$ ,  $\omega = k^{1/2}/(0.07DC\mu^{1/4})$*

## Axis

In the *Boundary Conditions* window, select *axis*. Use the drop-down menu to make sure that its type is *axis*.

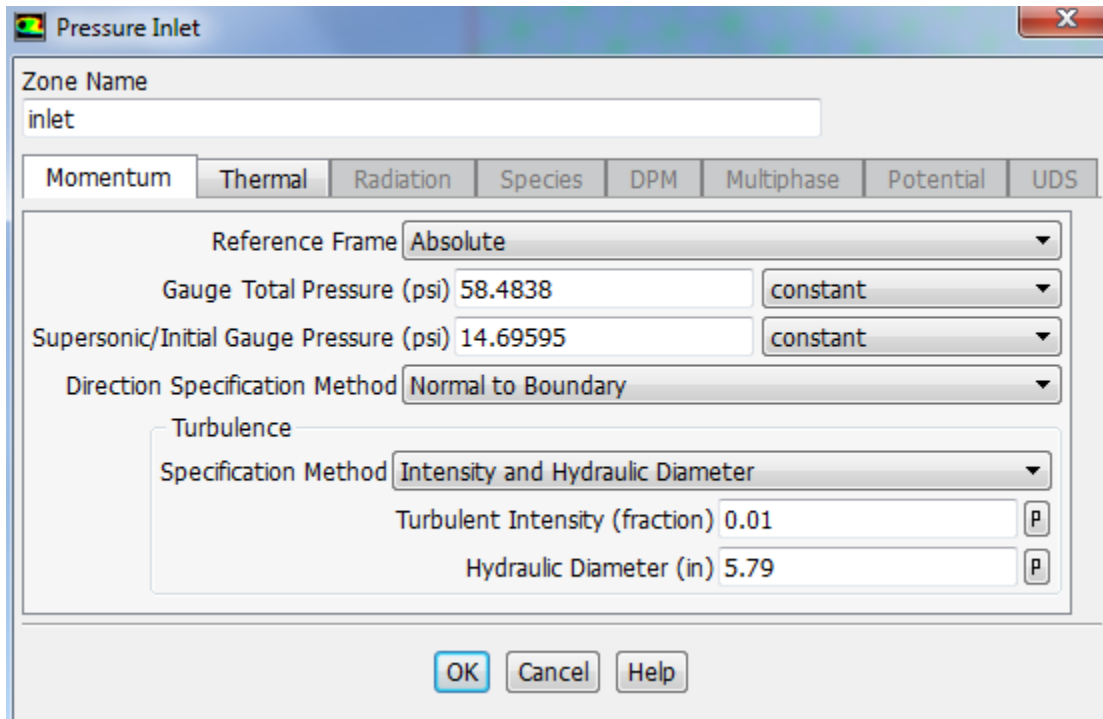


## Axis-Nozzle

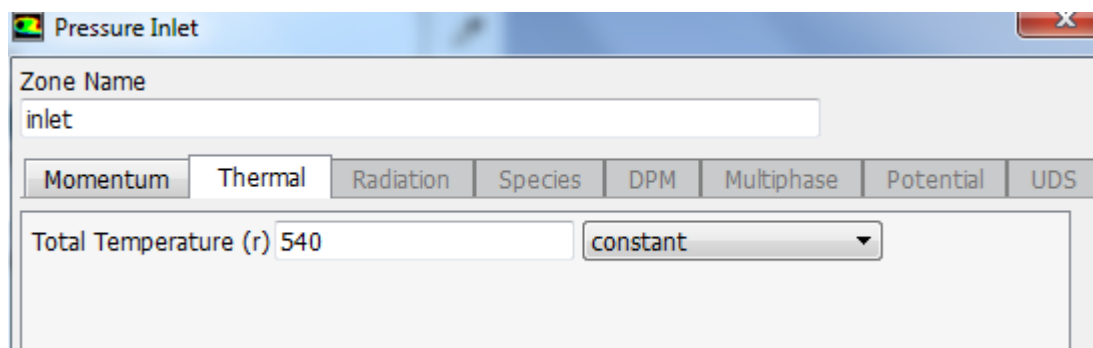
In the *Boundary Conditions* window, select *axis-nozzle*. Use the drop-down menu to make sure that its type is *axis*.

## Inlet

In the *Boundary Conditions* window, select *inlet*. Use the drop-down menu to change the *Type* to *pressure-inlet*. You will be asked to confirm the change, and do so by pressing *OK*. Next, a dialogue box will open with some parameters we need to specify. Change the *Gauge Total Pressure (psi)* to *58.4838* (equivalent to pressure ratio of 4) and make sure that Initial gauge pressure and Turbulent intensity are 14.6959 and 0.01 respectively.

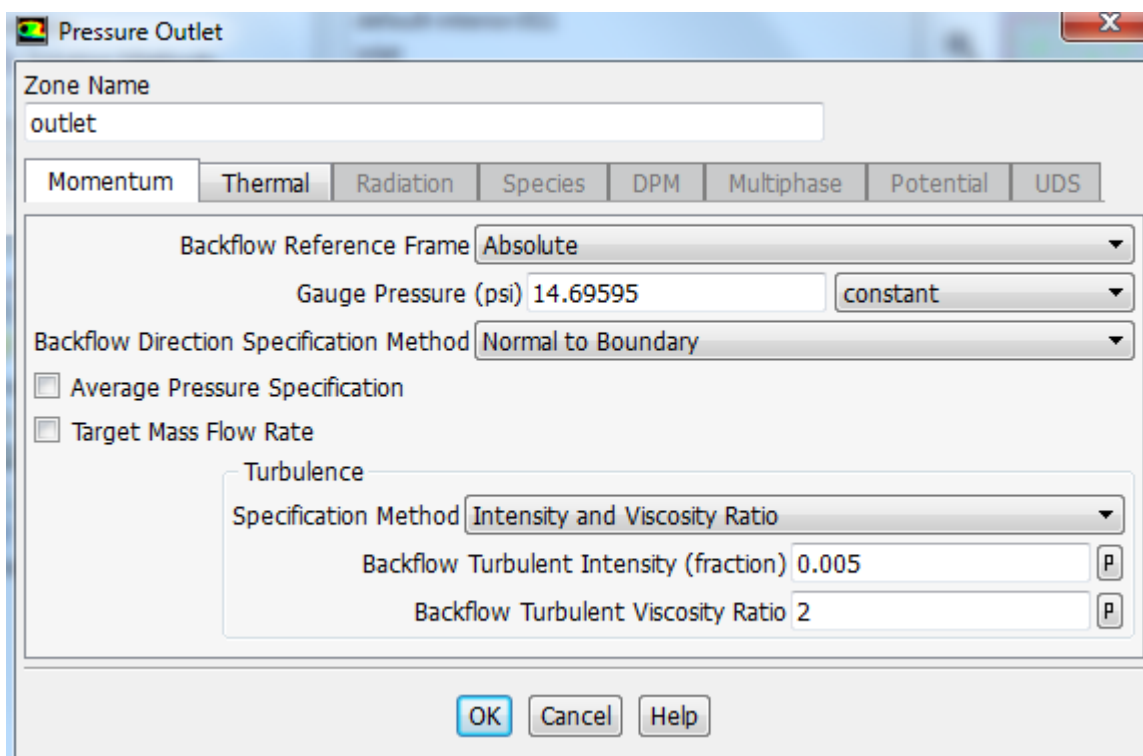


Also, select the *Thermal* tab, and ensure that the temperature correctly defaulted to *540 R*. When you are finished, press *OK*.



### Outlet

In the *Boundary Conditions* window, select *outlet*. Use the drop-down menu to change the *Type* to *pressure-outlet*. Keep the gauge pressure at 14.6959 psi, press *OK*.



The screenshot shows the 'Pressure Outlet' dialog box. The 'Zone Name' field contains 'outlet'. The 'Momentum' tab is selected, showing options for 'Backflow Reference Frame' (set to 'Absolute'), 'Gauge Pressure (psi)' (set to 14.6959), and 'Backflow Direction Specification Method' (set to 'Normal to Boundary'). There are checkboxes for 'Average Pressure Specification' and 'Target Mass Flow Rate', both of which are unchecked. A 'Turbulence' section is expanded, showing 'Specification Method' (set to 'Intensity and Viscosity Ratio'), 'Backflow Turbulent Intensity (fraction)' (set to 0.005), and 'Backflow Turbulent Viscosity Ratio' (set to 2). At the bottom are 'OK', 'Cancel', and 'Help' buttons.

### Wall-nozzle-in

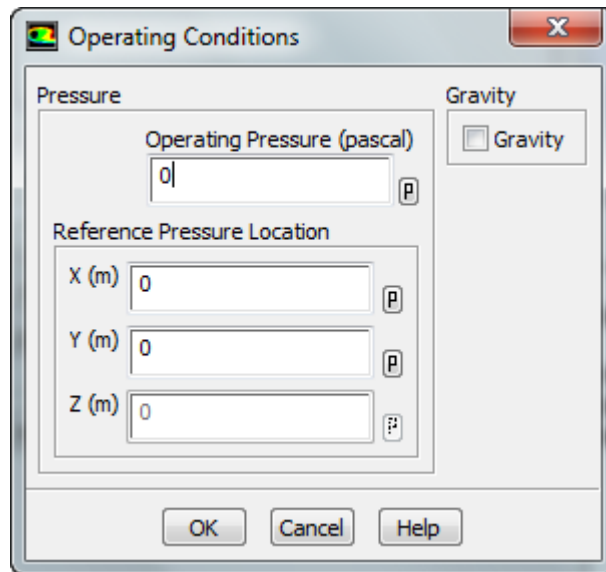
In the *Boundary Conditions* window, select *wall*. Use the drop-down menu to change the *Type* to *wall*.

### Wall-nozzle-out

In the *Boundary Conditions* window, select *wall*. Use the drop-down menu to change the *Type* to *wall*.

### Operating Conditions

In the *Boundary Conditions* window, select the *Operating Conditions* button. Change the *Gauge Pressure* to 0. Then press *OK*



It is important to check the operating conditions. When setting the density in materials to ideal gas, FLUENT calculates the density using the absolute pressure. However, the pressure we specify is the gauge pressure, not the absolute pressure. FLUENT will use the absolute pressure to compute the density therefore if we do not set the operating pressure to 0 our density will be incorrect for the flow field.



## Reference Values

In the *Outline* window, select *Reference Values*. Change the *Compute From* parameter to *inlet*. Check that the values are accurate. The reference values are used when calculating the non-dimensional results such as the drag coefficient.

Task Page

**Reference Values**

Compute from  
inlet

Reference Values

Area (m2)	1
Density (kg/m3)	1.353345
Depth (m)	1
Enthalpy (j/kg)	1861.91
Length (in)	39.37008
Pressure (psi)	14.69595
Temperature (r)	469.5063
Velocity (m/s)	280.7668
Viscosity (kg/m-s)	1.7894e-05
Ratio of Specific Heats	1.4

Reference Zone  
fluid-out

## Solution Methods

In the *Outline* window, select *Solution Methods* to open the *Solution Methods* window. Under *Scheme*, choose *coupled*, Under *Spatial Discretization*, ensure that the following options are selected:

Gradient: *Least Squares Cell Based*.

Pressure: *Second Order*.

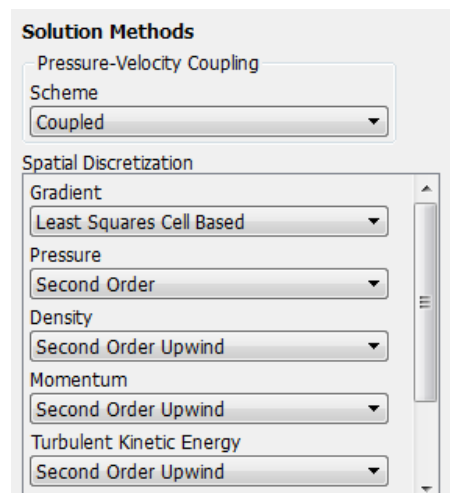
Density: *Second Order Upwind*.

Momentum: *Second Order Upwind*.

Turbulent Kinetic Energy: *Second Order Upwind*.

Specific Dissipation Rate: *Second Order Upwind*.

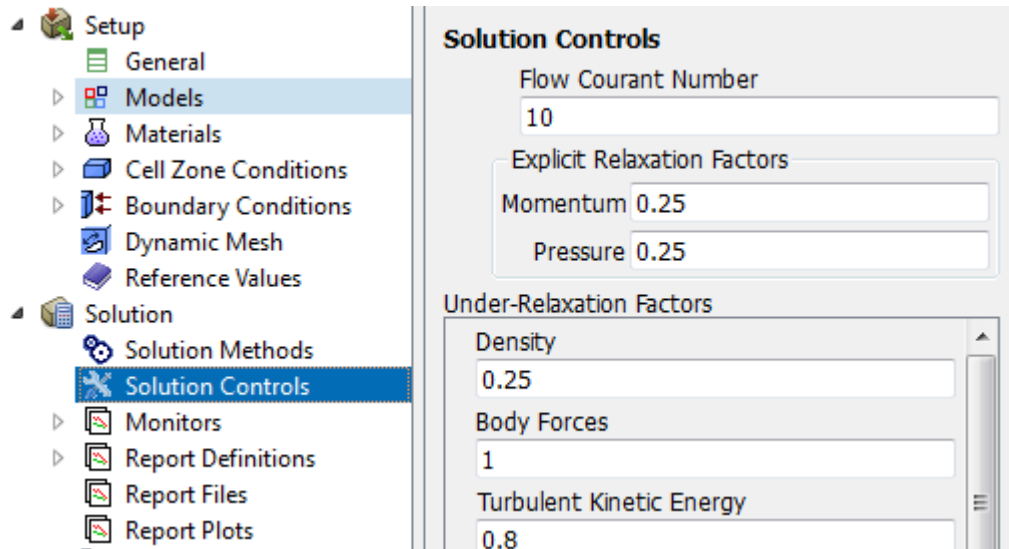
Energy: *Second Order Upwind*.



- Pressure-based coupled solver
  - pressure field is extracted by solving a pressure correction equation obtained by manipulating continuity and momentum equations
  - full implicit coupling between momentum and continuity equations
  - coupled system solved using coupled algebraic multigrid (AMG) scheme
  - Incomplete Lower Upper (ILU) smoother to smooth residuals between levels of AMG
- QUICK scheme for interpolating face values from cell centers in the continuity, momentum and energy equations
  - weighted average of 2nd order upwind and 2nd order central differencing
  - less diffusive than a traditional 2nd order upwind
- Pressure face values interpolated by a 2nd scheme similar to 2nd-order upwinding
- Least Squares cell-based gradient evaluation
- 2nd order spacial accuracy preserved

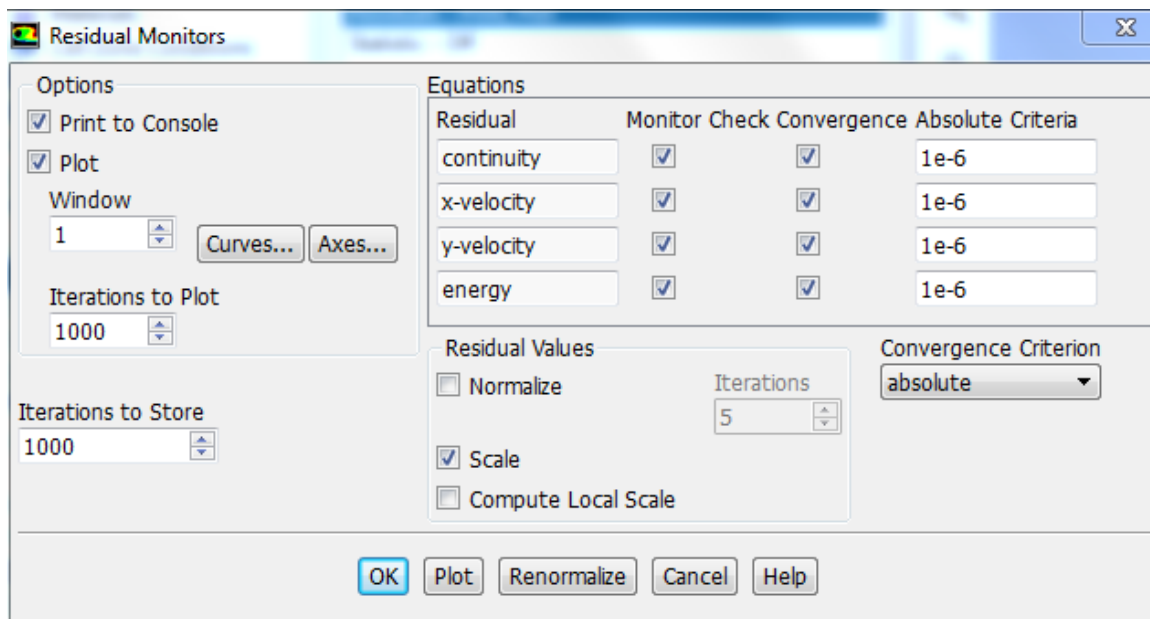
## Solution Controls

In the *Outline* window, select *Solution Controls* to open the *Solution Controls* window. Ensure that the *Courant Number* is set to *10.0*.



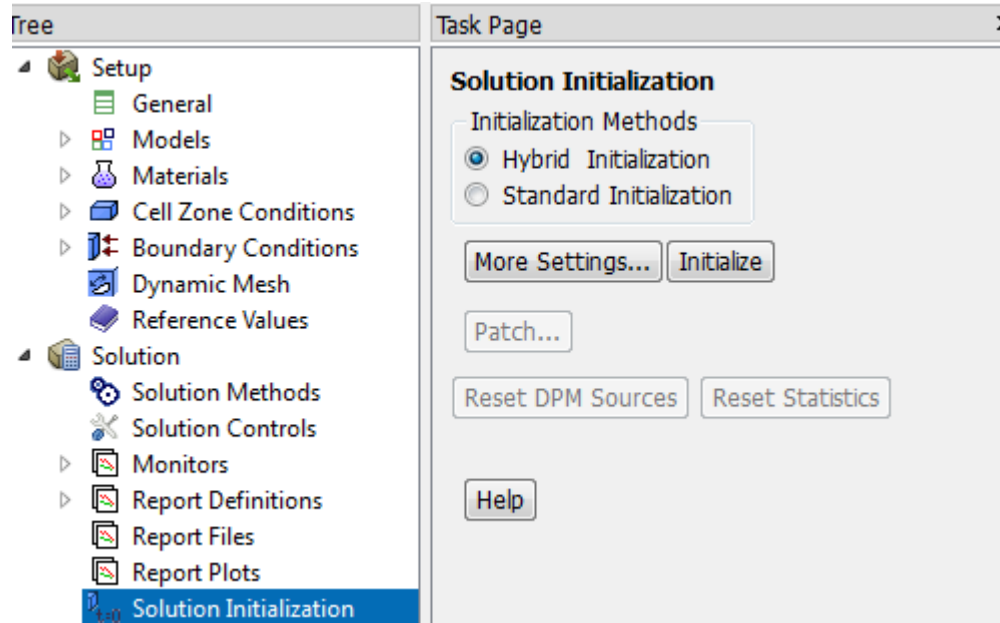
## Monitors

In the *Outline* window, click *Monitors* to open the *Monitors* window. In the *Monitors* window, select *Residuals - Print, Plot* and press *Edit...* This will open the *Residual Monitors* window. We want to change the convergence criteria for our solution. Under *Equation* and to the right of *Continuity*, change the *Absolute Criteria* to *1e-6*. Repeat for *x-velocity*, *y-velocity*, and *energy*, then press *OK*.



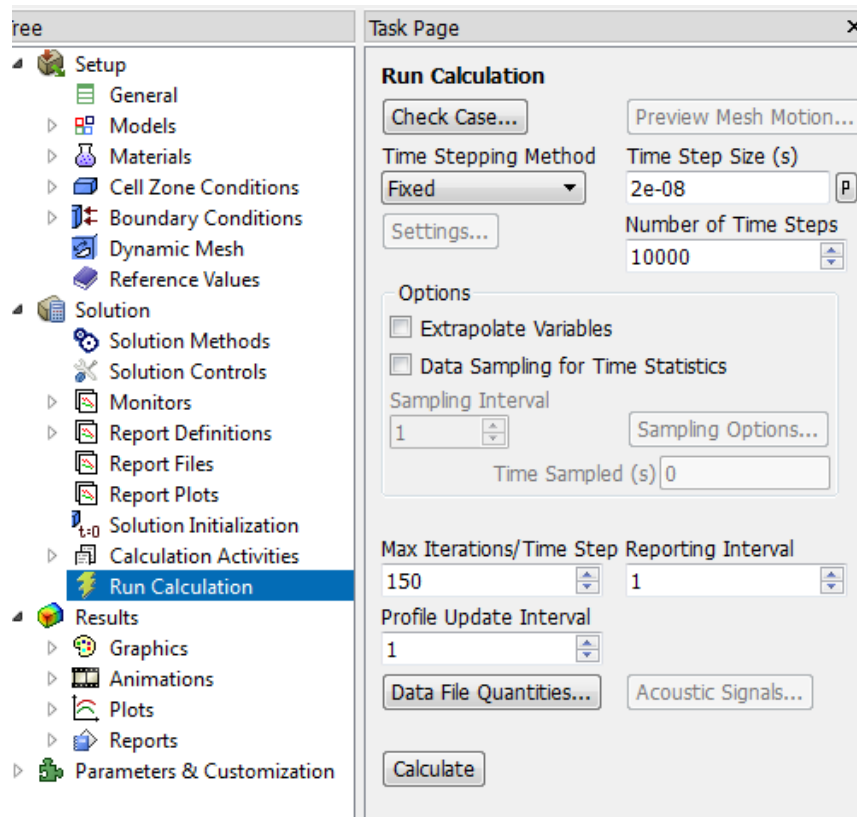
## Solution Initialization

In the *Outline* window, select *Solution Initialization*. We need to make an "Initial Guess" to the solution so FLUENT can iterate to find the final solution. In the *Solution Initialization* window, select *Hybrid Initialization*, then press *Initialize*.



## Run Calculation

In the *Outline* window, select *Run Calculation*. Change the *Number of Iterations* to *2000*. Double click *Calculate* to run the calculation. It should a few minutes to solve. After the calculation is complete, save the project. Do not close FLUENT.

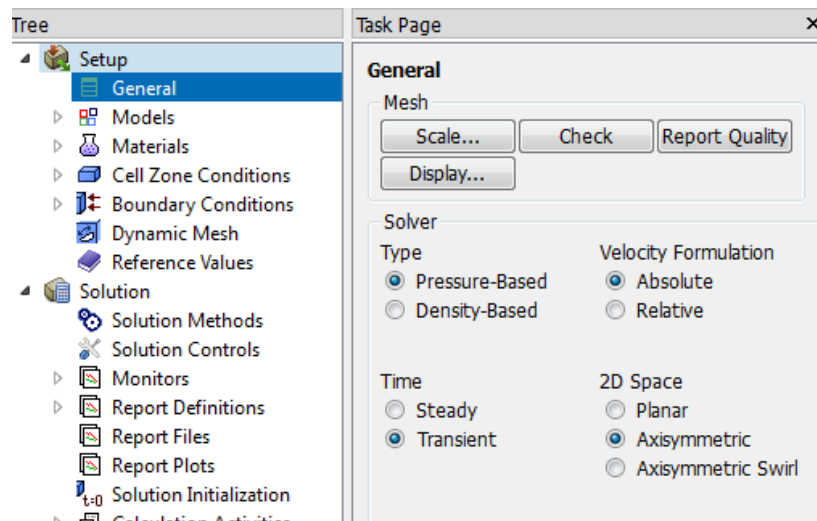


## Note:

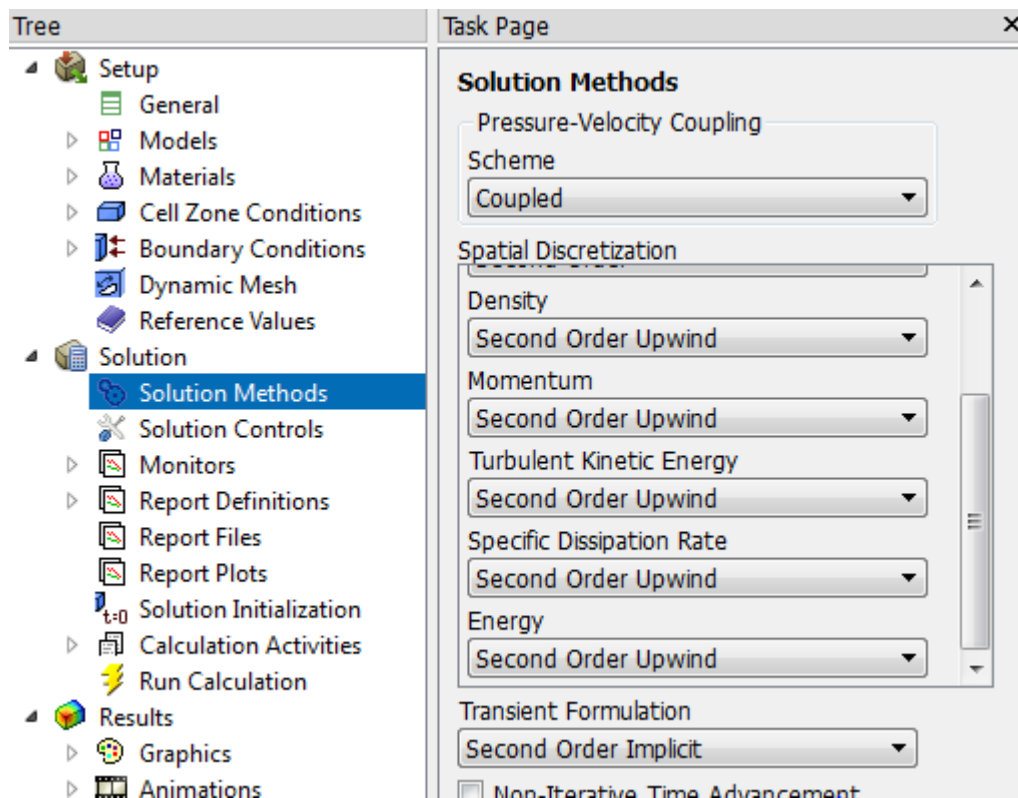
1. You need to run the simulations again by using Realizable K- $\epsilon$  turbulence model and compare the results with K-w SST Turbulence model.

## For the Transient Case

1. Under General, choose Transient



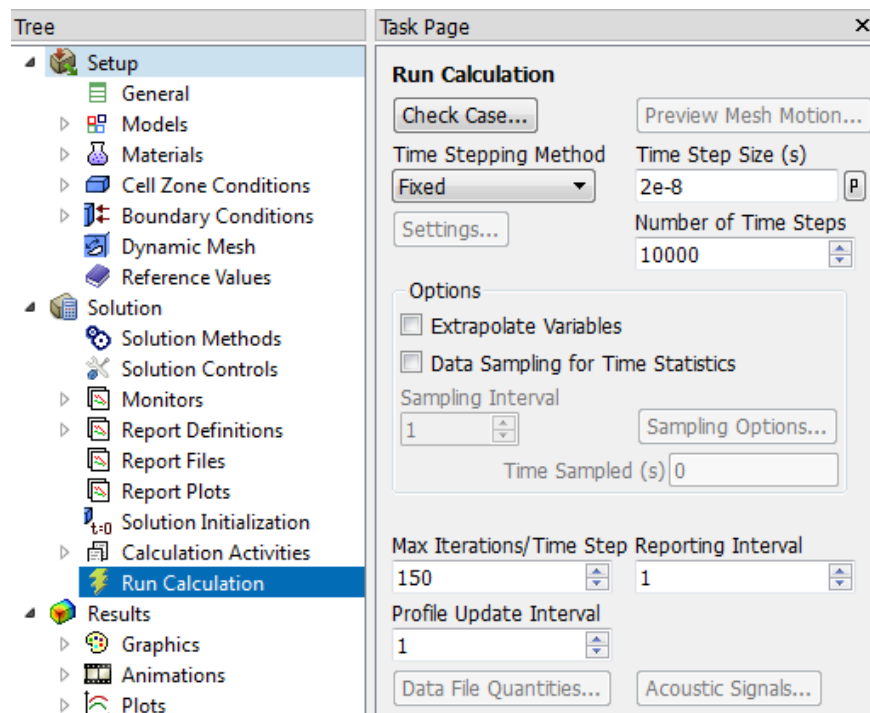
2. Under Solution Methods>> Transient Formulation, use, Second Order Implicit



3. Initialize the flow by using hybrid initialization

4. Under Calculations

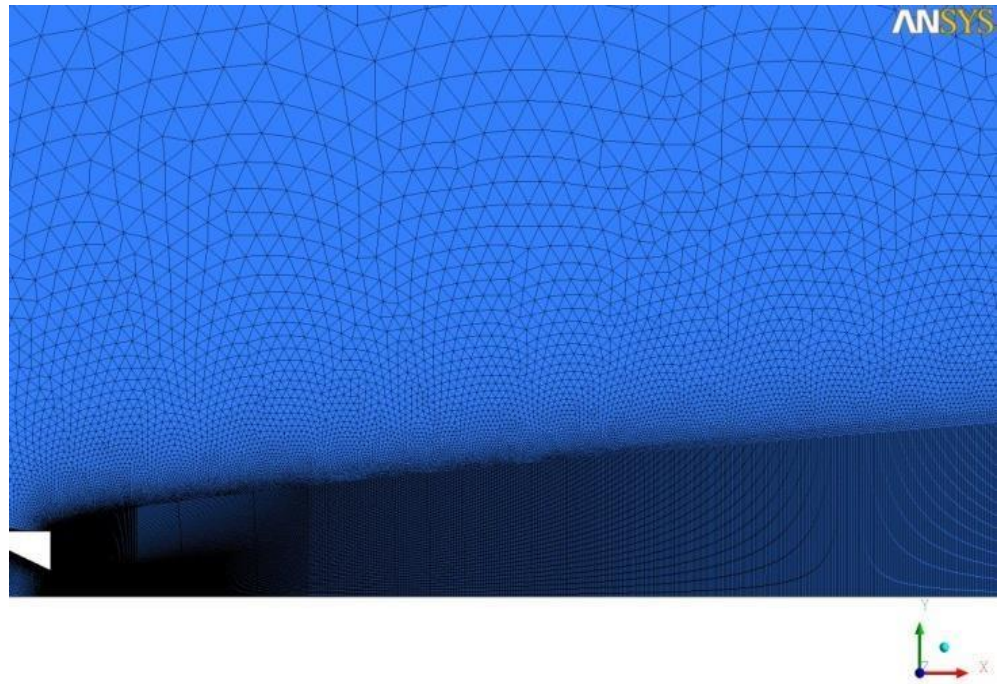
- Use a time step of  $2 \times 10^{-8}$  sec and iterate for 10000 time step with max iterations/time step of 150



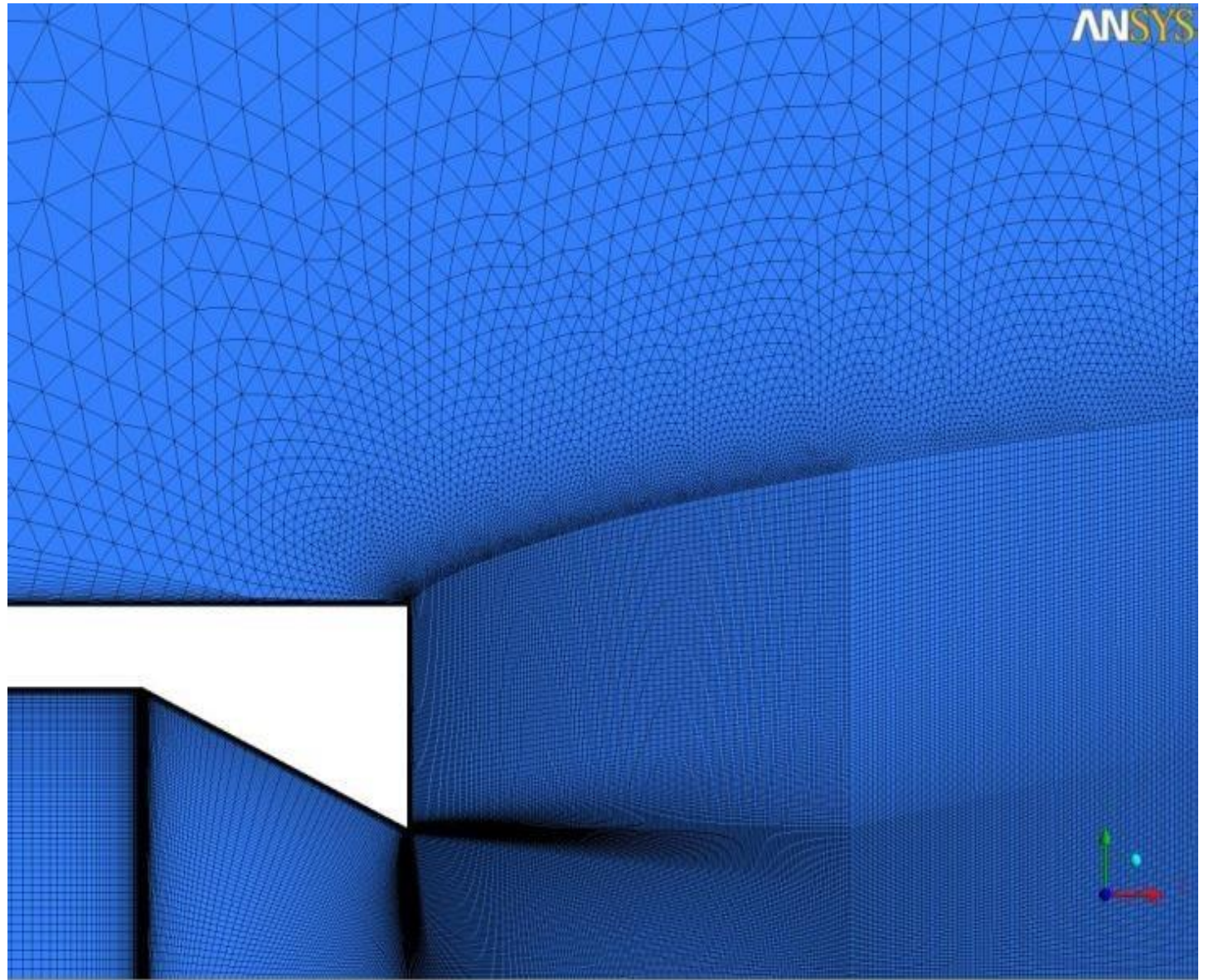
4. Run Calculations

# Computational Meshes

- A hybrid axisymmetric meshes is constructed using ANSYS pre-processing tools
  - block-structured quad mesh in the jet flow region, unstructured tri mesh in far field – boundary layer  $y^+ \approx 1$  mesh at the nozzle walls





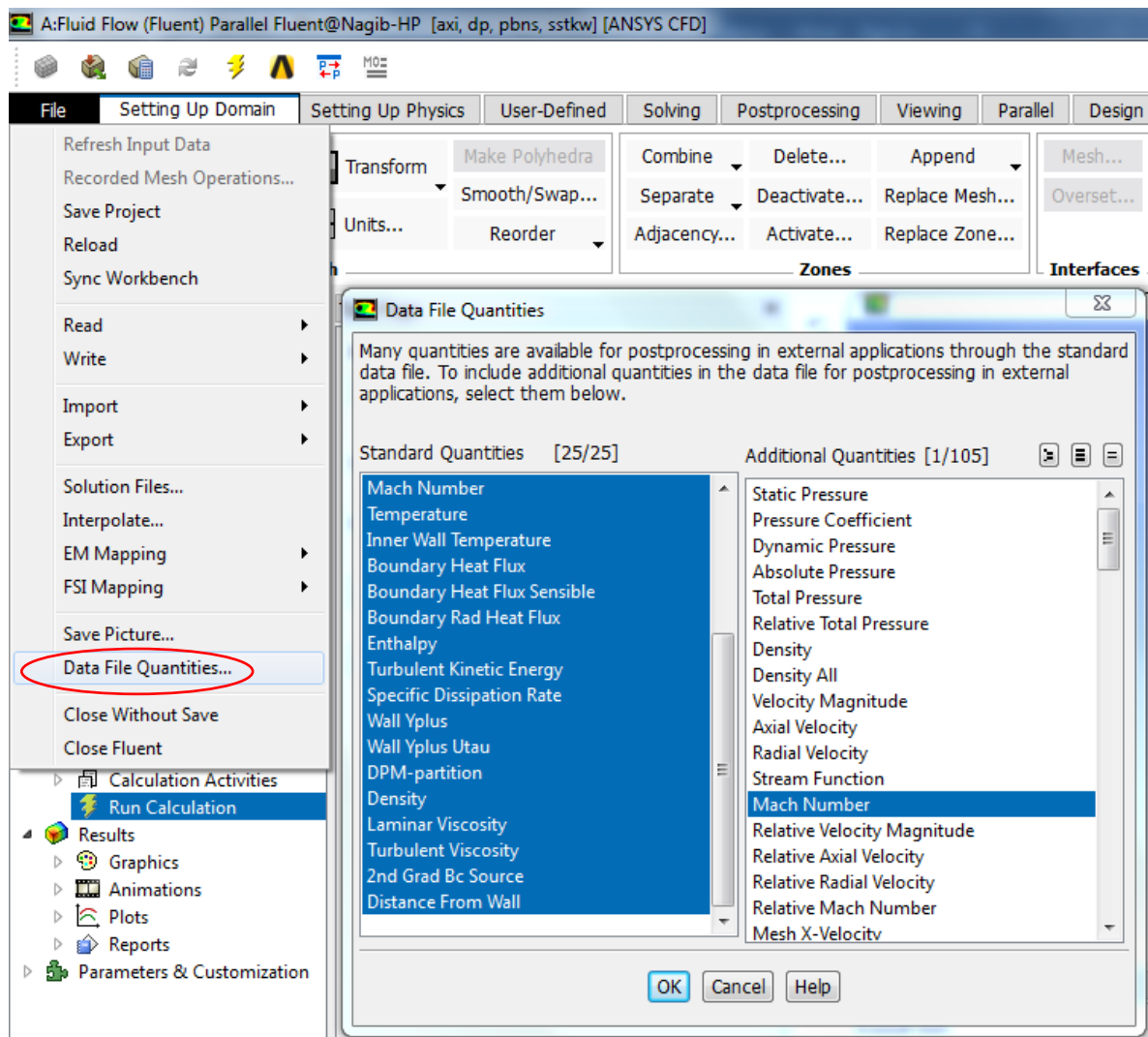


# Results

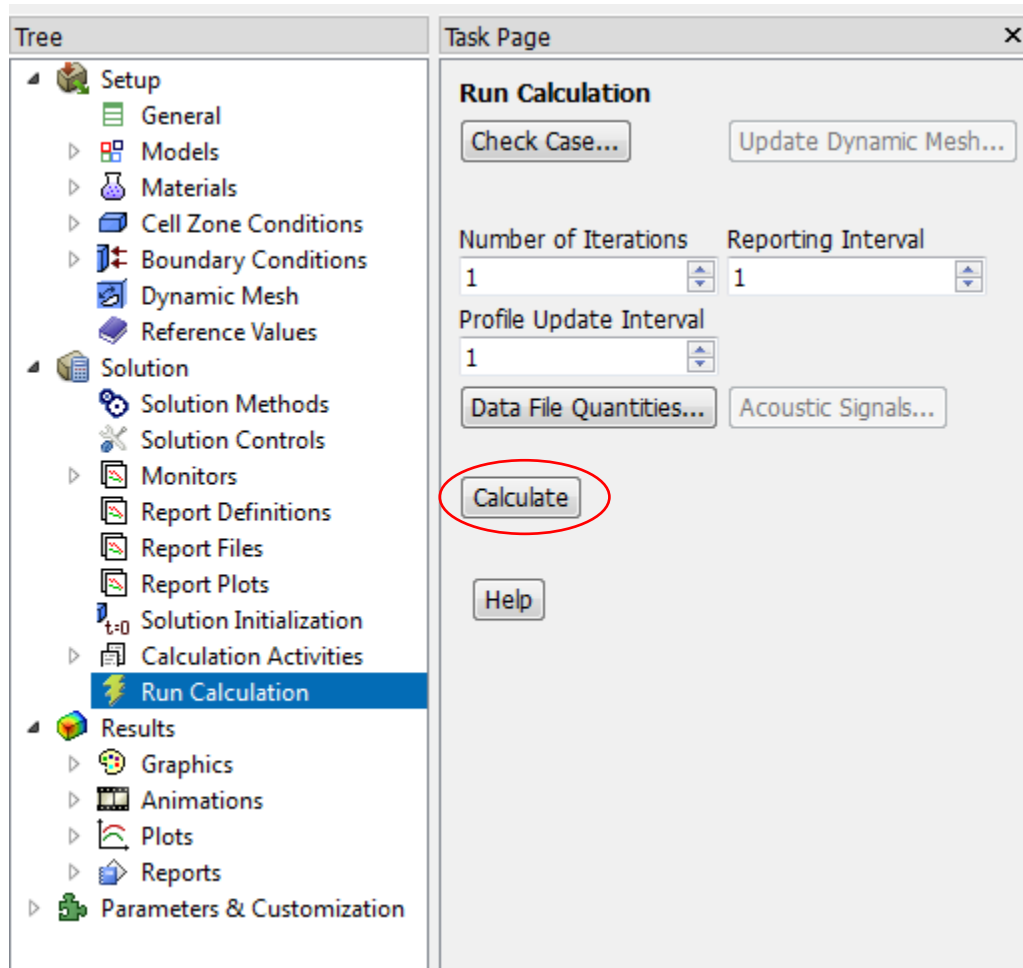
## Effects of NPR, Nozzle Pressure Ratio (Nozzle Total pressure / $P_{\text{ambient,static}}$ ) on Mach Number

### Mach Number Contours

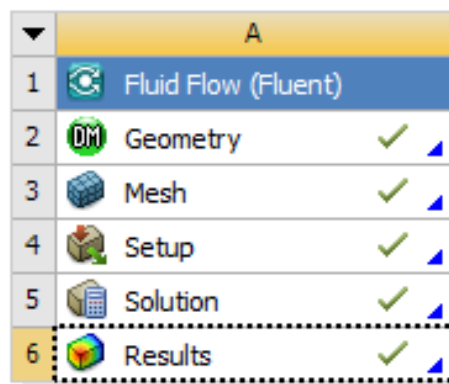
1. Some calculated parameters are not by default carried over into CFD-post. We are interested in such quantities (i.e. Mach Number). To manually transfer a customized selection of quantities
  - a. Inside Fluent, Select File > Data File Quantities
  - b. Under Additional Quantities, Select Static Pressure, Total Pressure, Mach Number, and Total Temperature



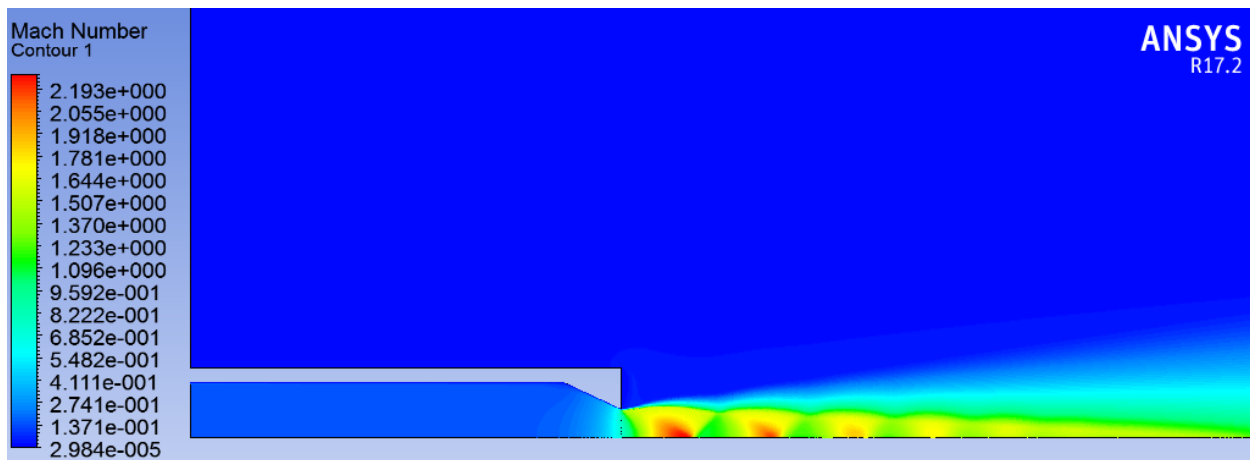
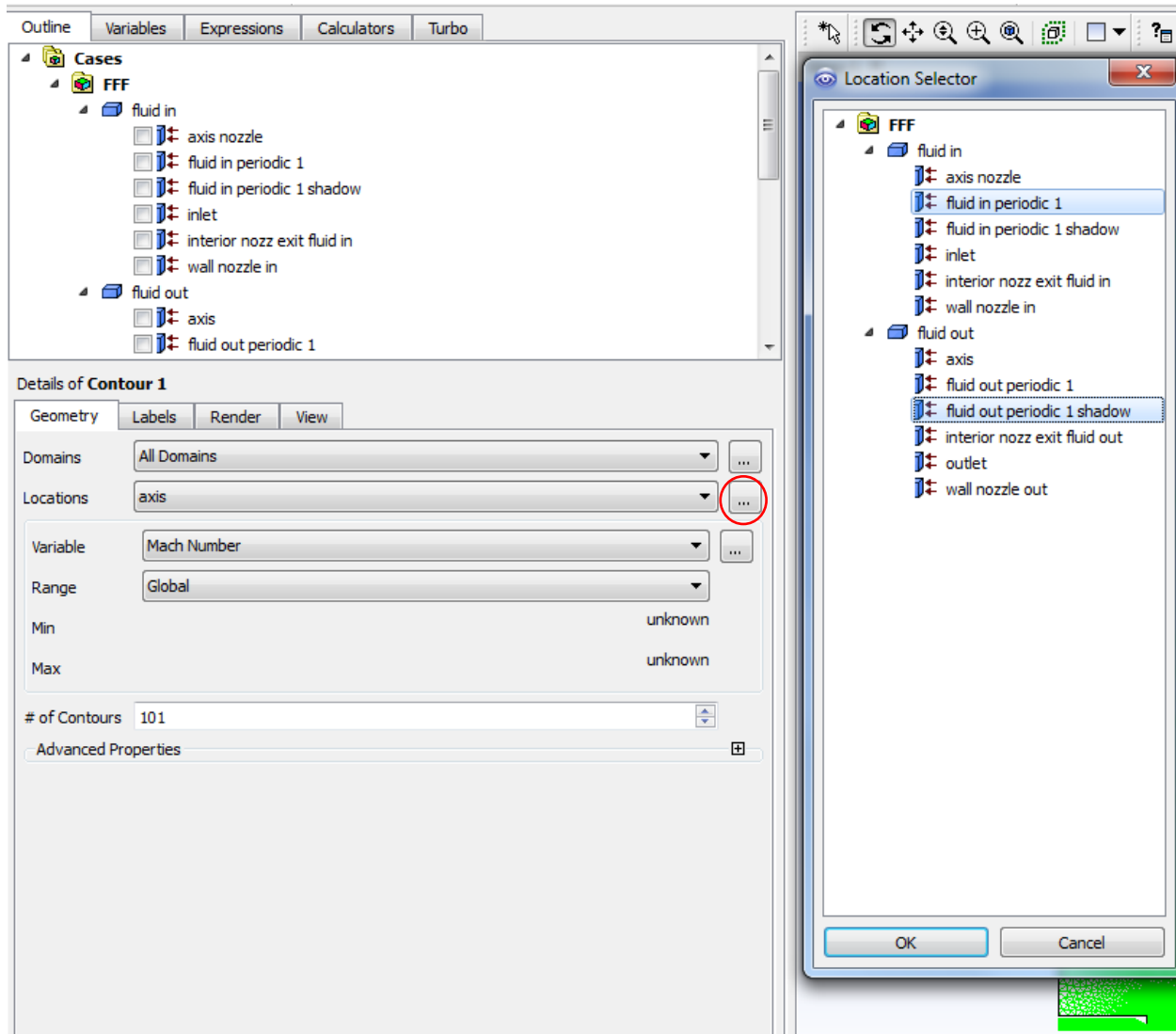
2. Run Fluent for one more iteration



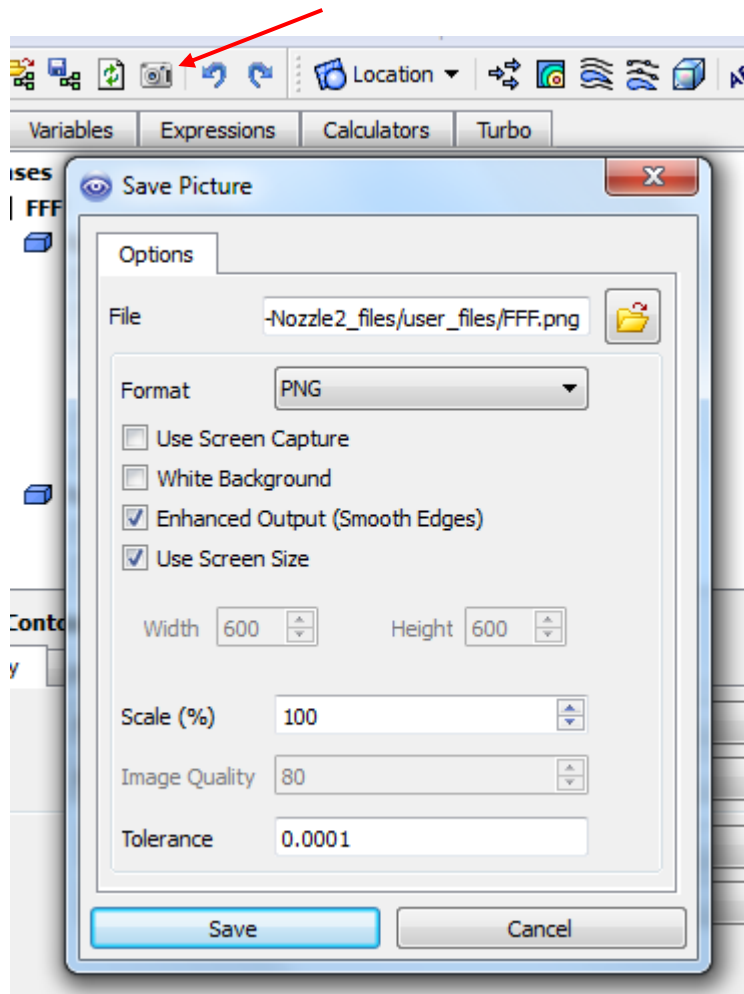
3. Post-processing will be done in CFD-post> Right Click Results in Workbench, then update
4. When update is done, Double Click Results in Workbench



5. We are interested in viewing contours of Mach Number in CFD-post
  - a. Select Insert > Contour > Name > Mach No.
  - b. Under Details of Mach No, select Locations > click on the box containing the 3 dots as shown below, and select fluid in periodic 1 and fluid out periodic 1.
  - c. Variable > Mach Number > No Contours = 101



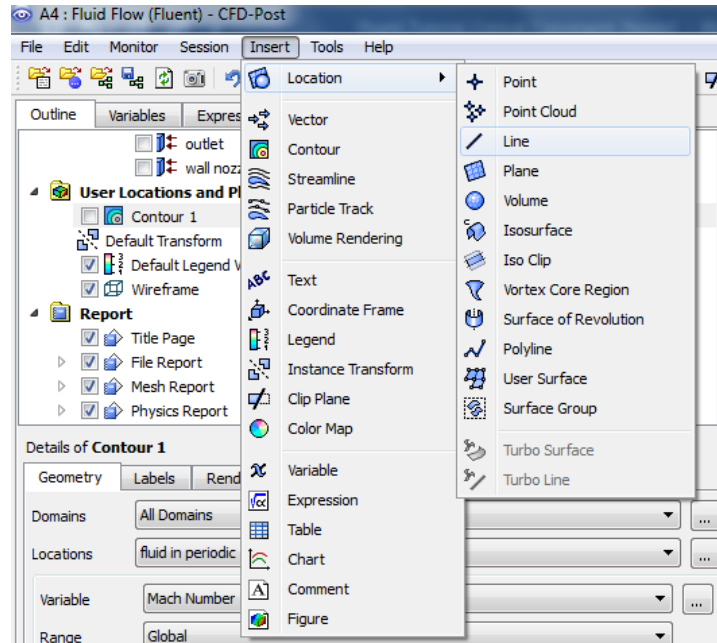
6. Save a copy of the figure in the graphics pane
  - a. Select the camera icon in the toolbar.



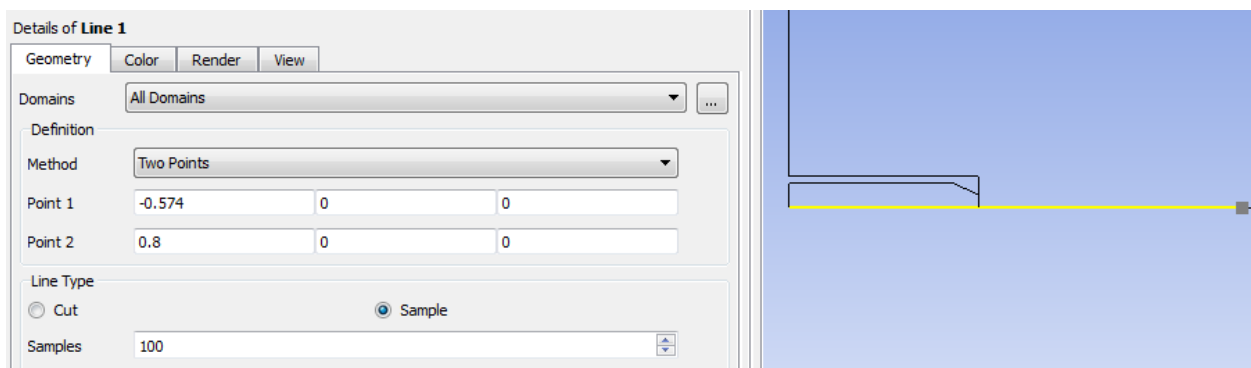
## Plot Mach Number Variation Along *the center line of the nozzle*

First, we'll create a line at *the center of the nozzle*. Then, we'll plot the Mach number variation along this line using the "Chart" facility in CFD Post.

1. In CFD-post, Insert line at the center of the nozzle.
  - a. Insert> Location > Line

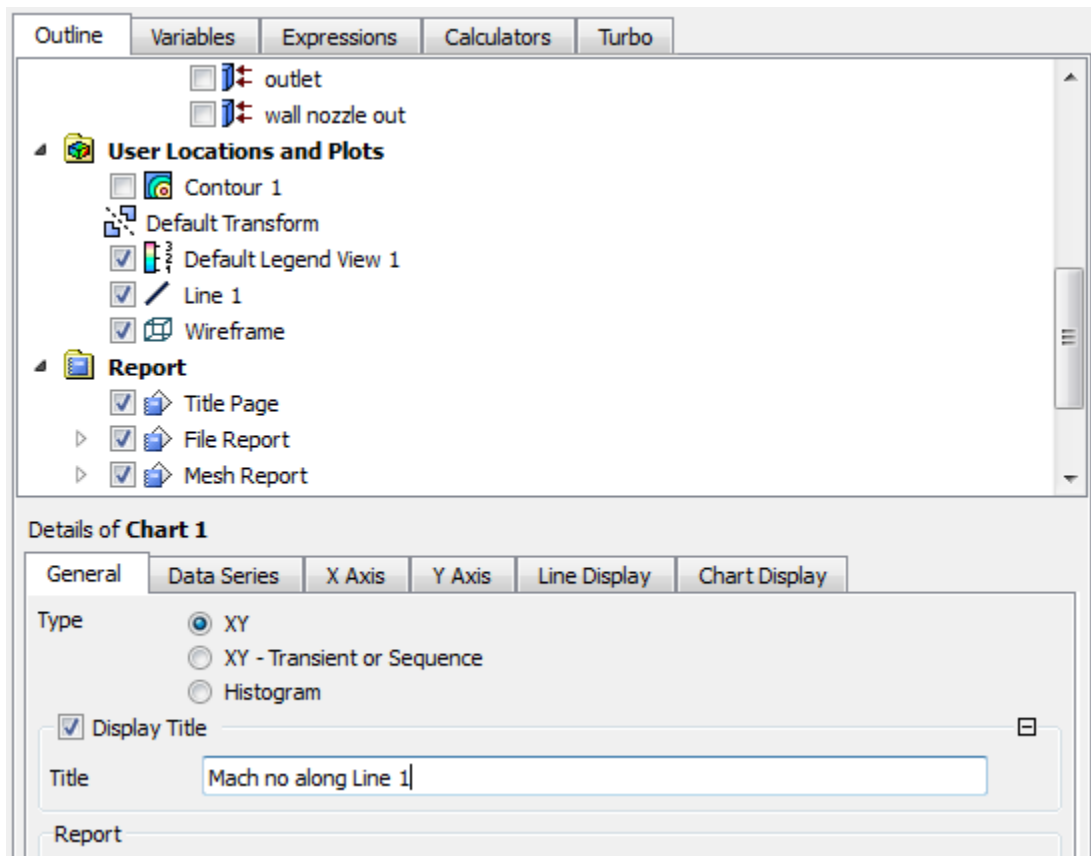


- b. Name Line 1
  - c. Under Details of Velocity vectors, type Point 1 coordinates (-0.574,0,0) and Point 2 coordinates (0.8,0.4,0).

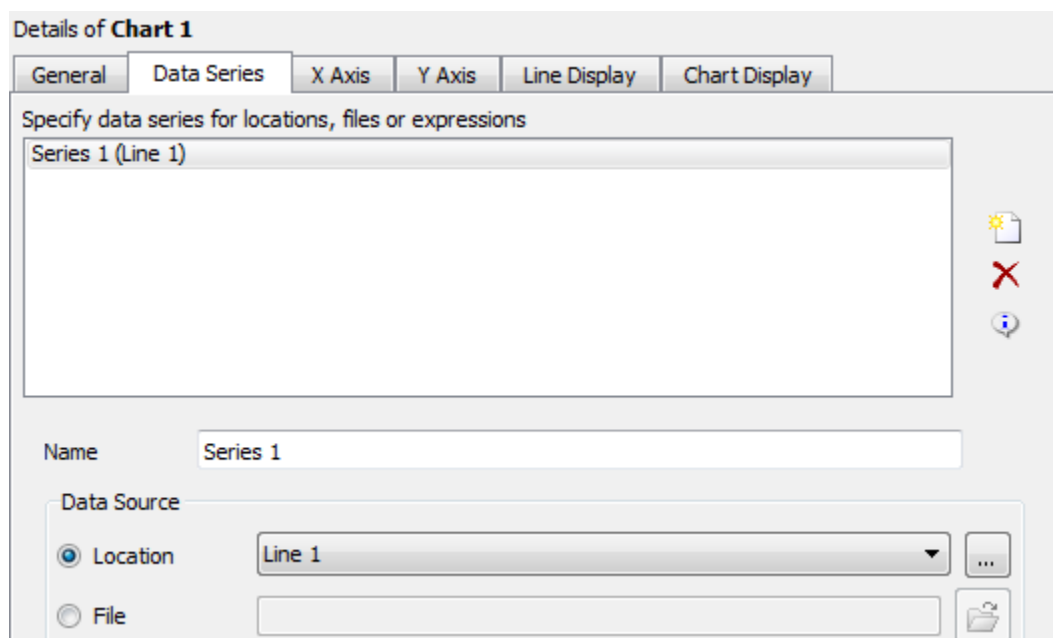




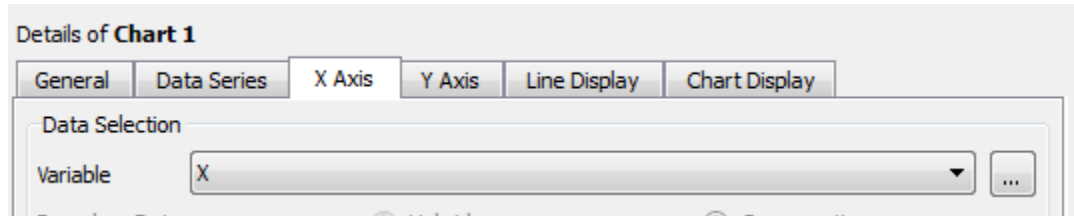
2. Plot Mach Number along the newly created line
  - a. Select Insert > Chart
  - b. Type "Mach no along Line 1" under Name



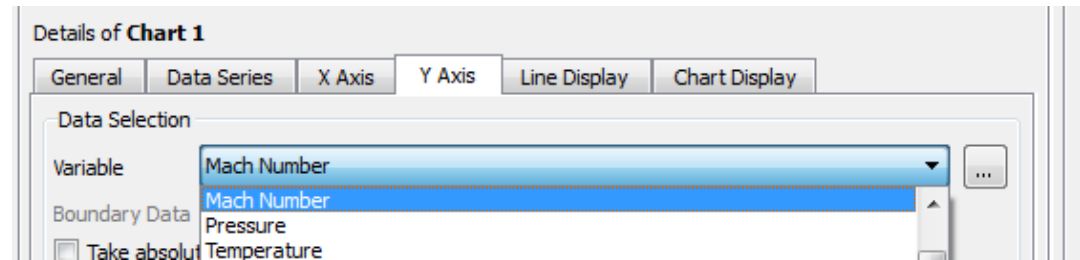
- c. Select the Data Series Tab. Select Location > Line 1.



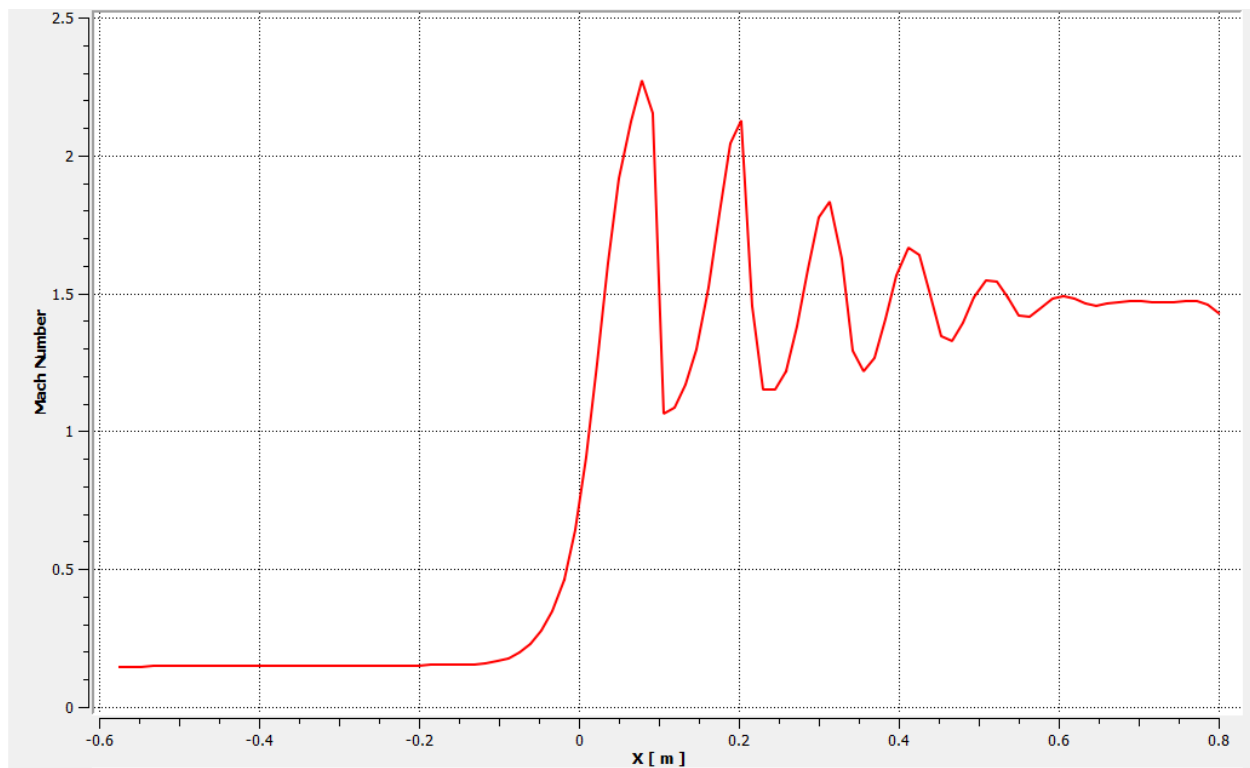
- d. select the X Axis tab. Select Variable > X.



- e. select the Y Axis tab. Select Variable > Mach Number.

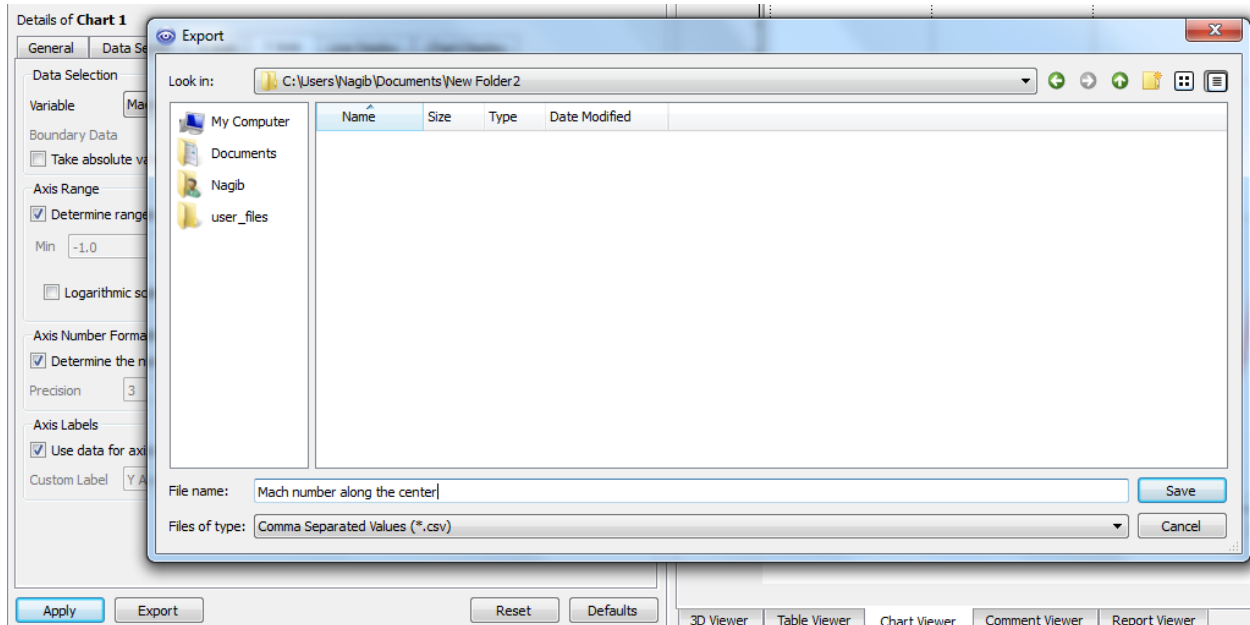


- f. Click Apply to Plot.



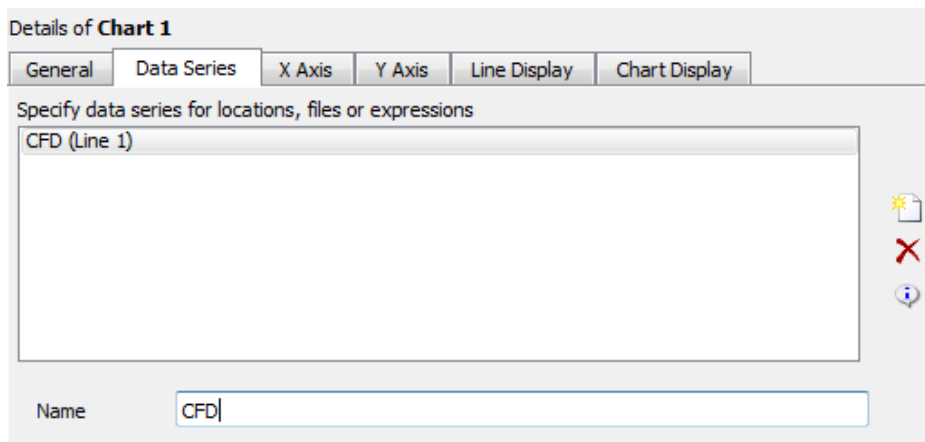


- g. Export the graph to excel sheet, by clicking export



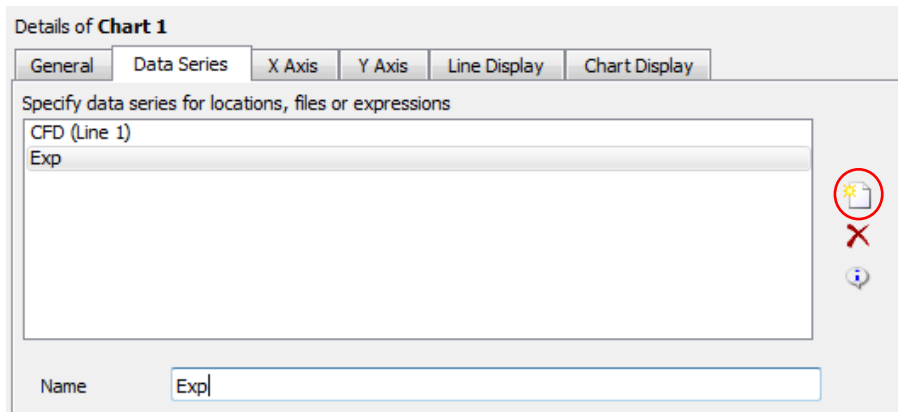
This file can be opened by excel to calculate the average value if needed.

- h. Select the Data Series tab> change the Name to CFD

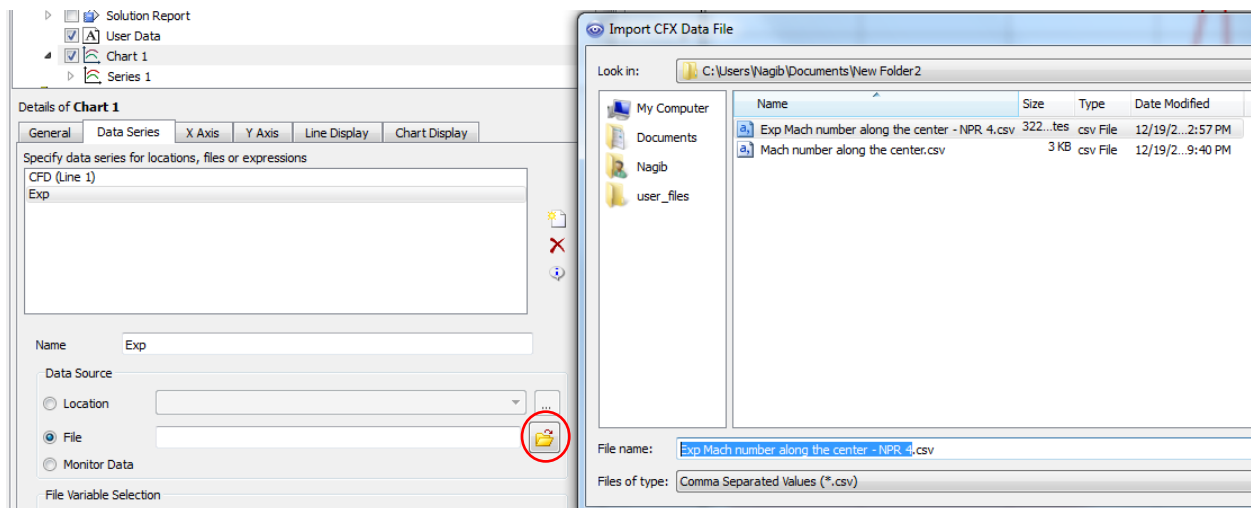


## Add the experimental data to the graph

Add new series by clicking on the new series icon, Name it EXP, Click Apply



Under data Source tab> Select File> import the file named (Exp Mach number along the center - NPR 4)



Under Data Series> line Style (None)> Symbols (Triangle)> Symbol Color Green

Details of **Chart 1**

General Data Series X Axis Y Axis Line Display Chart Display

Specify the properties of each line

Name	Data Series
<input checked="" type="checkbox"/> CFD	CFD (Line 1)
<input checked="" type="checkbox"/> Exp	Exp (C:/Users/Nagib/Documents/New Folder2/Exp Mach number along t...

Line Display

☒ Use series name for legend name

Line Style: None

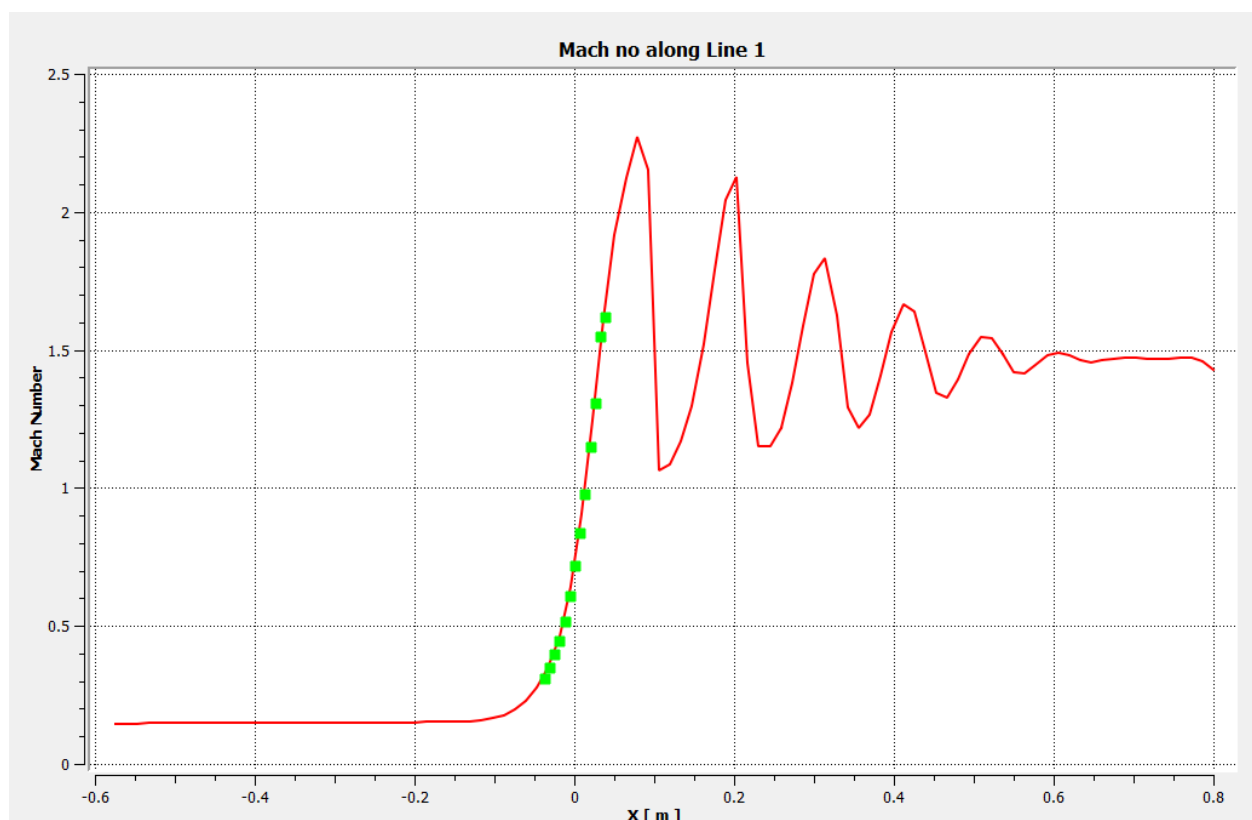
Symbols: Triangle

Symbol Color:  

☒ Fill Area

☒ Automatic ☐ Always On

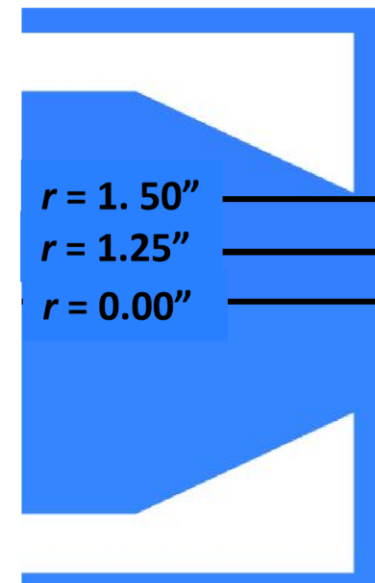
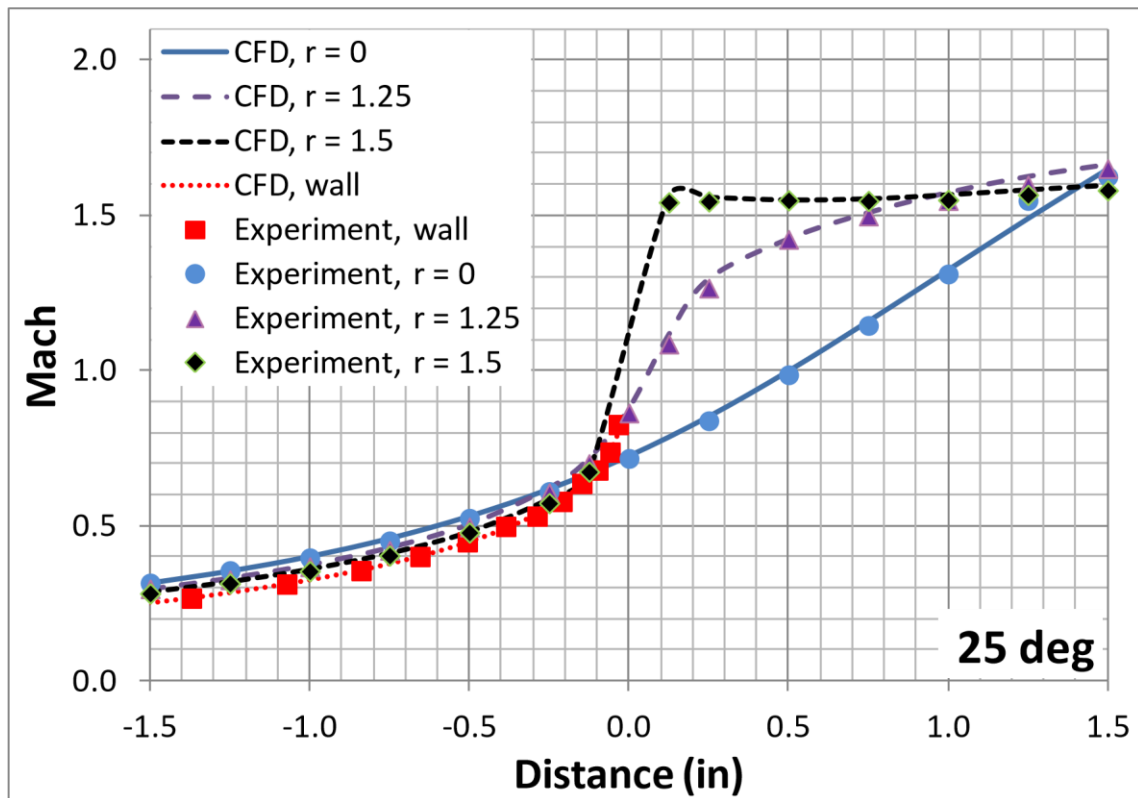
Click apply



Results from another paper for different locations are shown below:

## Comparison of Mach Numbers, $NPR = 4.0$

$$Mach = \sqrt{\left(\frac{2}{\gamma - 1}\right) \left( \left( \frac{p_t}{p_{local}} \right)^{\frac{\gamma}{\gamma - 1}} - 1 \right)}$$



# Calculating the discharge and Thrust coefficient of the nozzle

The discharge coefficient,  $C_d$  (The ratio of the actual mass flow to the ideal mass flow) and thrust coefficient,  $C_V$ , (the ratio of the actual thrust to the ideal thrust), were computed for each solution as follows:

$$C_d \equiv \frac{2\pi \cdot \int_0^{r_{jet}} \rho \cdot u \cdot r \cdot dr}{\rho_{jet} \cdot U_{jet} \cdot A_{jet}}$$

$$C_V \equiv \frac{2\pi \cdot \int_0^{r_{jet}} [\rho \cdot u^2 \cdot (p - p_\infty)] \cdot r \cdot dr}{U_{jet} \cdot \int_0^{r_{jet}} \rho \cdot u \cdot r \cdot dr}$$

The standard discharge coefficient,  $C_d$ , and thrust coefficient,  $C_V$ , are computed over the radius of the nozzle exit,  $r_{jet}$ , using the local density ( $\rho$ ), local streamwise velocity ( $u$ ), and local radius from the centerline ( $r$ ). The resultant nozzle exit area,  $A_{jet}$ , was 7.0686 in<sup>2</sup>. The ideal jet velocity,  $U_{jet}$ , is computed from the Mach number of the ideally expanded jet,  $M_{jet}$ , and the temperature of the jet,  $T_{jet}$ , all found in the following equations:

$$U_{jet} = M_{jet} \cdot \sqrt{\gamma \cdot R \cdot T_{jet}}$$

$$M_{jet} = \sqrt{\left(\frac{2}{\gamma-1}\right) \cdot \left(NPR^{\left(\frac{\gamma-1}{\gamma}\right)} - 1\right)}$$

$$T_{jet} = T_0 \cdot \left[1 + \left(\frac{\gamma-1}{2}\right) \cdot M_{jet}^2\right]^{-1}$$

In CFD Post, you should draw a line at the exit of the nozzle point (0,0,0) and point (0,0.038,0) and calculate the average velocity and average density by exporting the data to an excel file to calculate the average as shown before.

Calculate the drag and discharge coefficient as follows

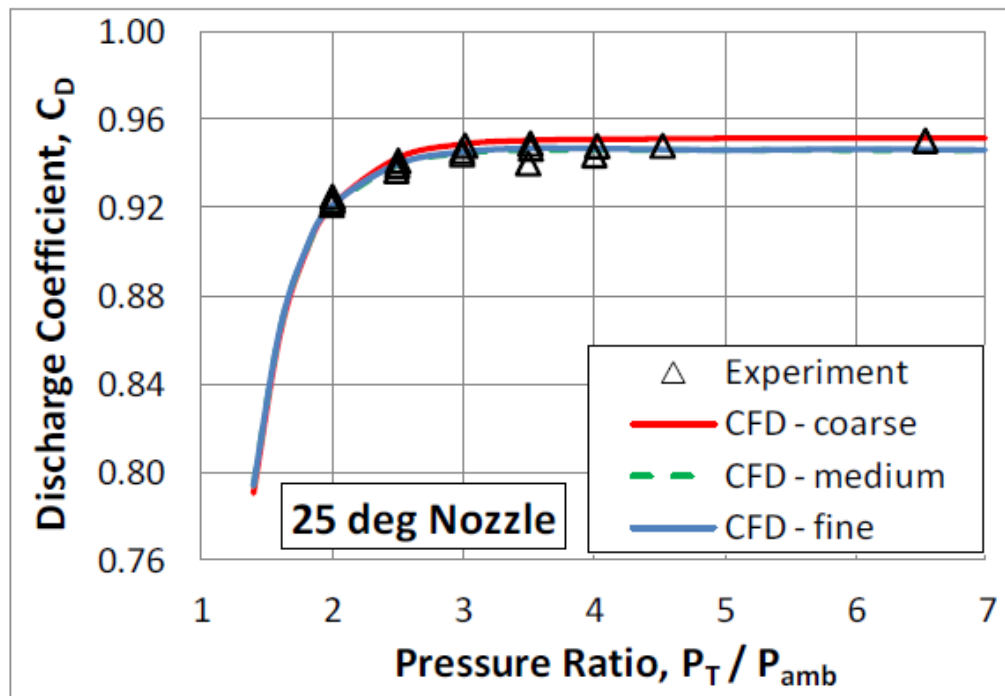
$$C_d = \frac{\rho_{av} \times u_{av}}{\frac{P_{jet}}{R T_{jet}} \times U_{jet}}$$

$$C_V = \frac{2\pi \times u_{av} \times (p_{av} - p_{amb})}{U_{jet}}$$

The values of  $M_{\text{jet}}$  and  $U_{\text{jet}}$  are shown for each NPR in the following Table

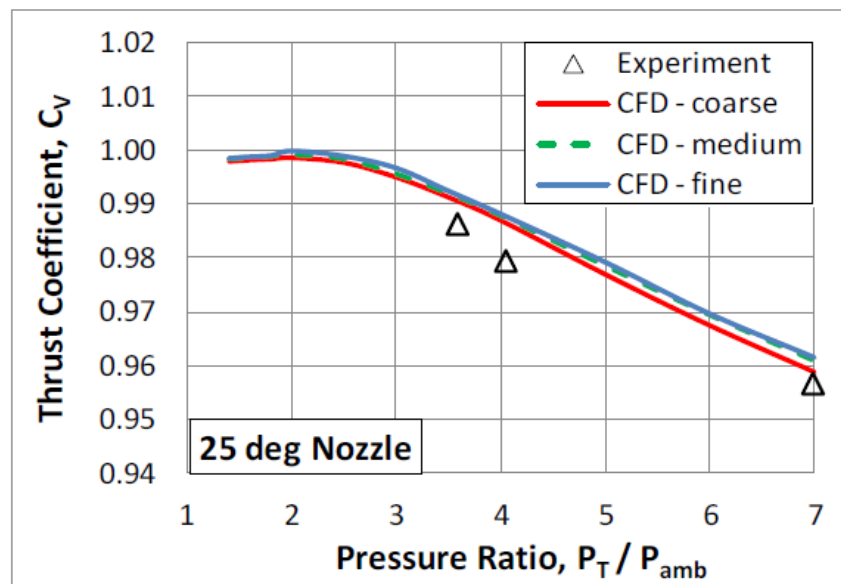
NPR	$M_{\text{jet}}$	$U_{\text{jet}}$ [ft/s]
1.4	0.710	742.0
1.6	0.848	868.7
1.8	0.956	963.6
2.0	1.046	1038.8
2.5	1.223	1176.2
3.0	1.358	1272.0
3.5	1.467	1344.3
6.0	1.559	1401.5
5.0	1.709	1487.9
6.0	1.828	1551.3
7.0	1.928	1600.5

## Comparison of Discharge Coefficient $C_d$



To Calculate the discharge coefficient  $C_d$ , you need to use CFD Post to calculate it. This step is left to the student.

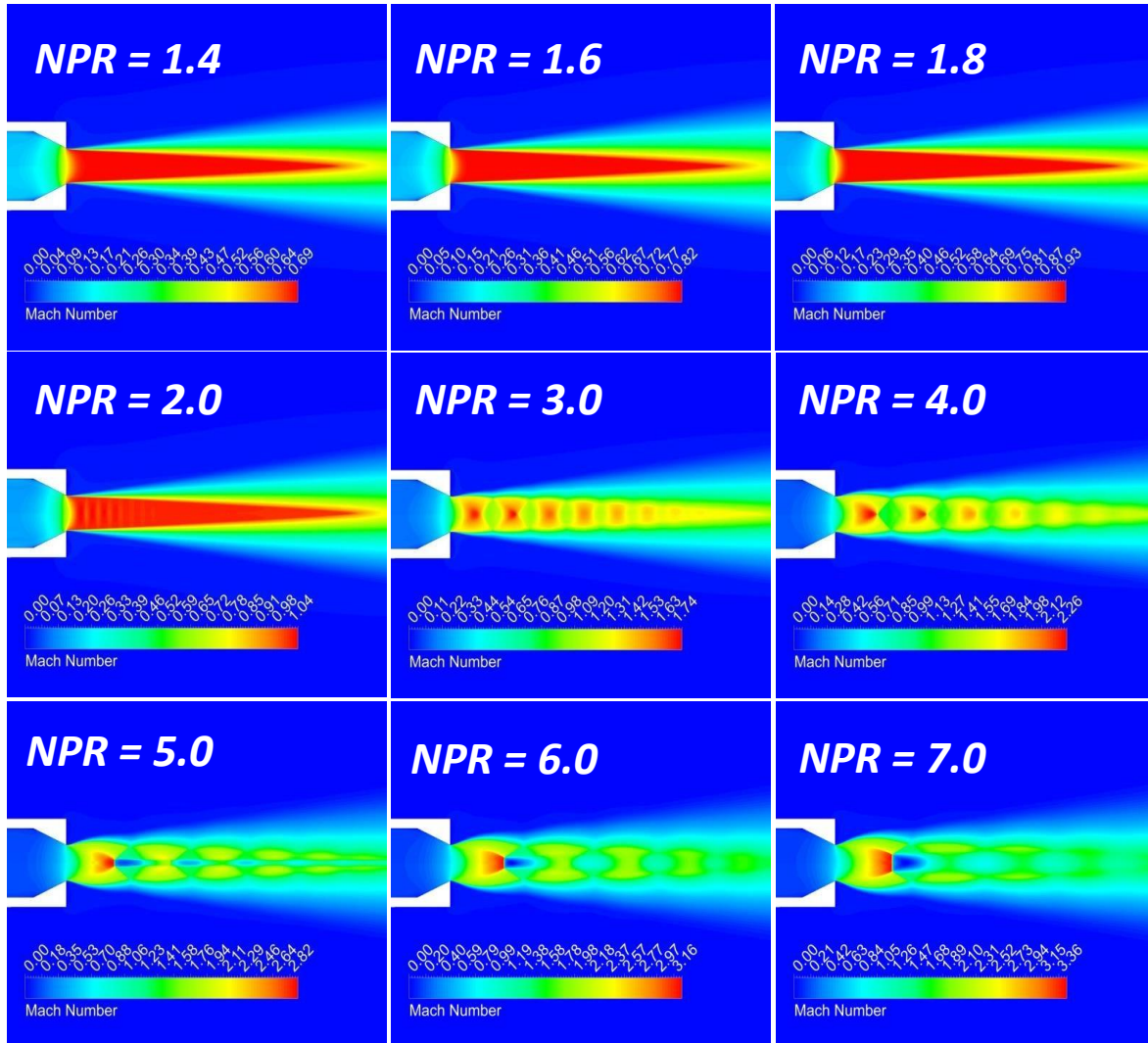
## Comparison of Thrust Coefficient $C_v$



To Calculate the Thrust coefficient  $C_v$ , you need to use CFD Post to calculate it. This step is left to the student.

# Tutorial Requirement

1. Run the simulation for different Nozzle Pressure Ratios, NPR (Nozzle Inlet Total pressure /  $P_{\text{ambient,static}}$ ) from 1.4 to 7 as shown below



2. Repeat the simulation by using Realizable K-epsilon turbulence model and compare the mach contours and Mach values across the center line of the nozzle results with the k-omega SST turbulence model results.
3. Run the transient case and create a video the video by using the method shown below in CFD Post:  
<https://youtu.be/xQBvYceTg8E?t=898>
4. Calculate the thrust and drag coefficients for NPR=4 and compare it with the experimental data ( $C_D=0.95$  and  $C_V=.978$ )



# Summary

- Numerical predictions of nozzle coefficients, Mach number distributions and shock location are in excellent agreement with experimental data as shown in the plotted data in this tutorial.
- Pressure-based coupled solver (PBCS) is a robust and effective method for solving subsonic and supersonic nozzle flow problems
  - adequately resolves physics and capture all essential features
  - less memory and CPU intensive than a traditional density-based approach
  - can be an economically attractive alternative to a density-based algorithm for obtaining steady-state solutions to subsonic and supersonic problems
- A convergent nozzle will not allow supersonic exit speeds of the combustion gasses, but due to their high temperature their speed of sound is considerably higher than that of the surrounding air. For example, at 700°C the speed of sound in air is 625 m/s. Since thrust is mainly determined by the difference in entry and exit speeds of the air flowing through an engine, a higher speed than flight speed is required for positive thrust. Low supersonic flight speeds are entirely possible with a convergent nozzle.
- If the design is meant to fly supersonically, which implicates a lot of design adaptations, it makes sense to go for higher supersonic speed; however, this requires both an adjustable intake and an adjustable convergent-divergent nozzle. Both increase efficiency, dramatically so at higher Mach numbers.