

# ANSYS Fluent Tutorial Part 1

This tutorial introduces you to the ANSYS workbench and Fluent environments. Upon completing this tutorial you will understand:

- ANSYS workbench environment
  - Create a new project, create geometry, mesh the domain, identify and name boundary conditions, grid adaptation
- Flow simulation in Fluent
  - Export mesh to Fluent, apply boundary conditions, iterate toward the solution, examine the flow fields, obtain numerical results, examine and validate the results

## Project 1 description:

Simulate fluid flow through a rectangular channel shown below.

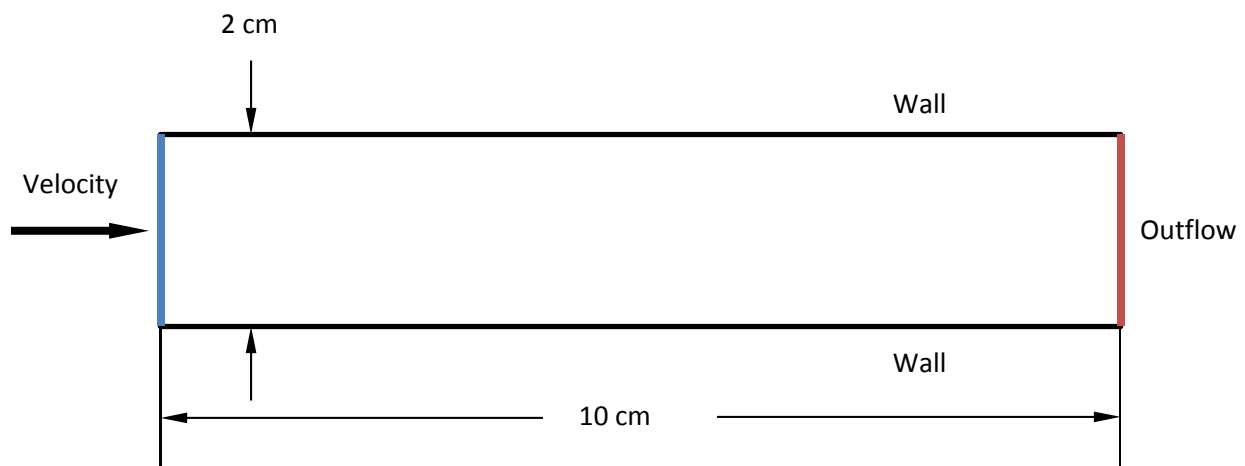


Figure 1.1

Consider air flowing through a rectangular channel of 2cm height and 10cm length. Assume that the channel has an infinite width. Inlet air velocity is  $0.1 \text{ m/s}$  at the inlet and outlet is at the atmospheric pressure.

# 1 ANSYS Workbench

## 1.1 Open Workbench

1. Open the Start Menu
2. Type **Workbench** into the search
3. Select the icon for **Workbench 17.1**
4. Alternately, you can find it in the **ANSYS 17.1** folder of the start menu

## 1.2 Create a Fluid Flow (Fluent) Project

1. Under the **Toolbox** tab (left menu), *drag* the **FLUENT module** over to the **Workspace** (white space) or *double-click*.
2. You will see your new **Project** appear. Name it *Project 1 Laminar Flow Coarse Mesh*. This will differentiate the project file from some of the other fluent simulations you will be creating.
3. **Save** the project in your **U:/** drive.

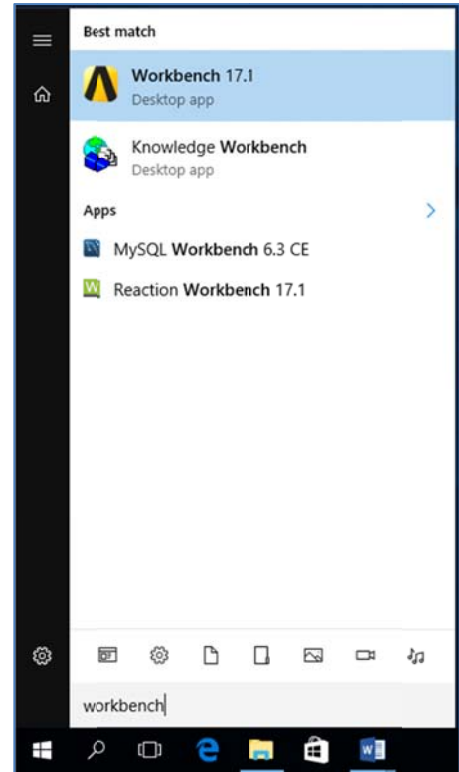


Figure 1.1

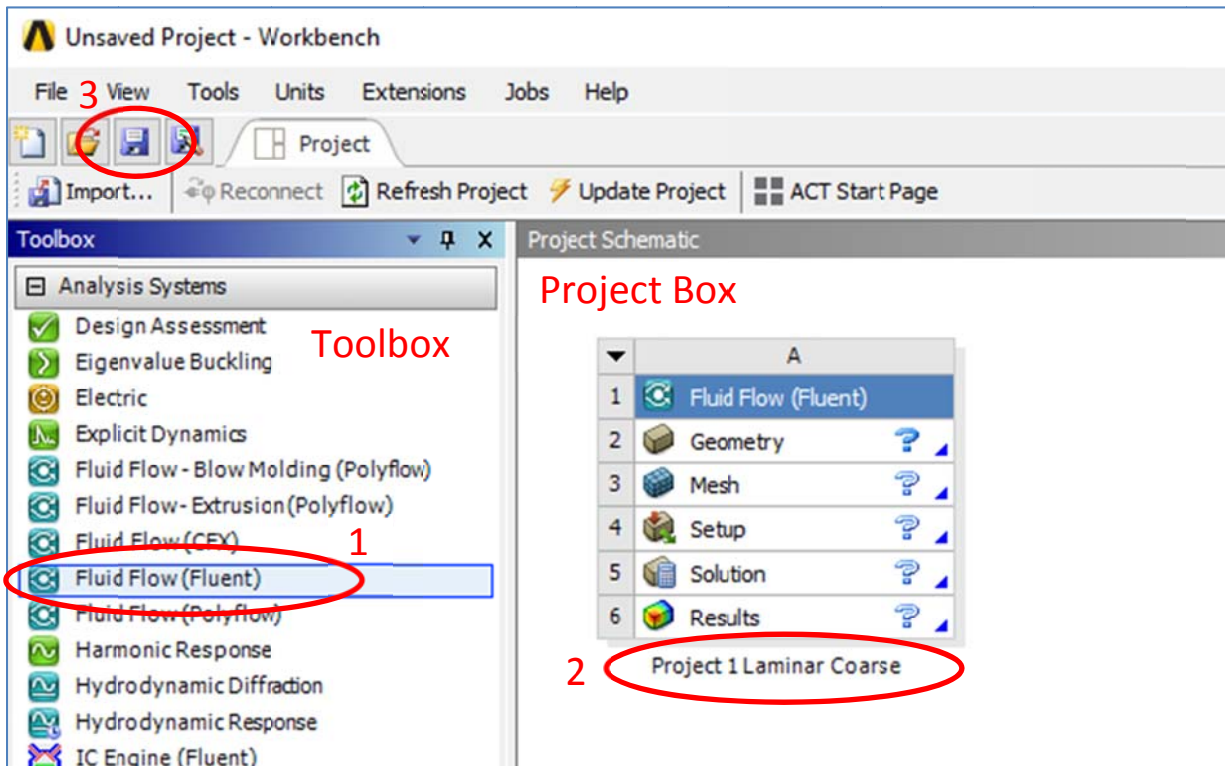


Figure 1.2

### 1.3 Set up Workbench Information

1. On the top menu, *click View*
2. *Select Files*. A menu will pop up at the bottom. This can tell you where the files are stored. There will be a bunch created as you progress and it may be important to note the location.
3. On the top menu again, *click View*, and *select Properties*. This time, a menu will appear on the right side of the screen. The **Properties** menu will change as you select different tabs on your project.

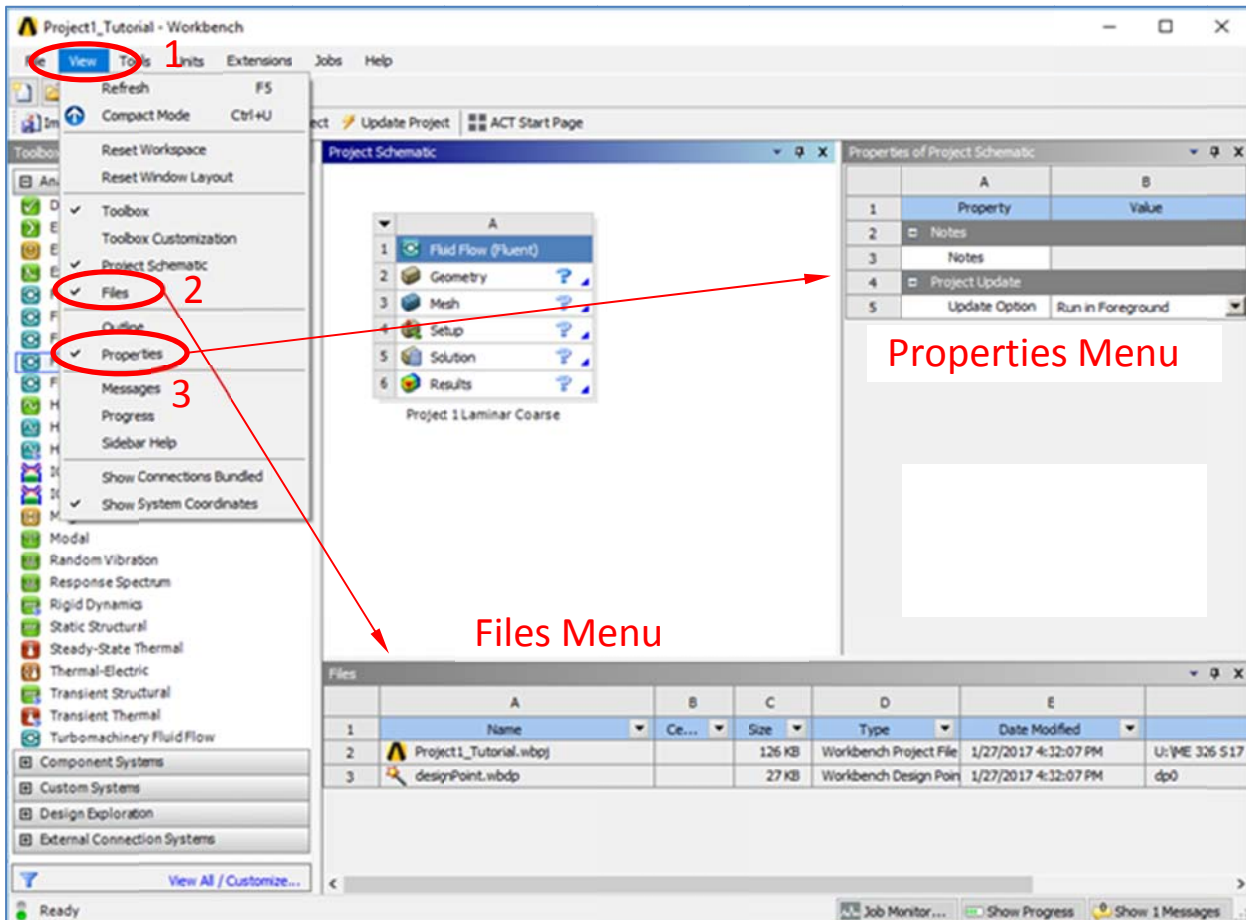


Figure 1.3

### 1.4 Set Analysis type

1. Within your **Project** box, *single-click* **Geometry**. Your **Properties** menu will change.
2. Under **Advanced Geometry Options**, set **Analysis Type** to 2D.

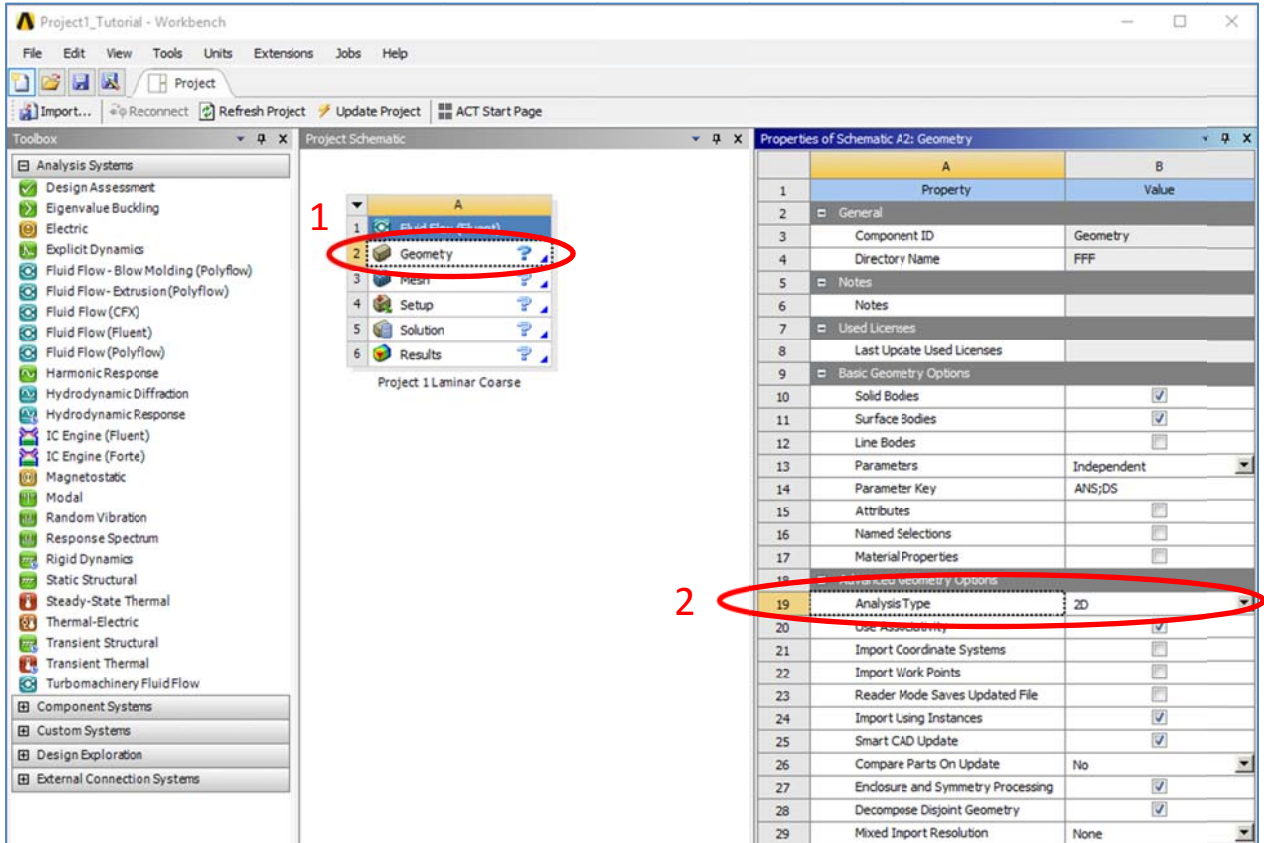


Figure 1.4

### 1.5 New Geometry

1. In the Project Box, *right-click* **Geometry**.
2. Select **New Design Modeler Geometry...**
3. ANSYS **Design Modeler (DM)** will then open.

Note that you can also **Import Geometry** if you made the geometry in a CAD program. Just save the CAD file as a universal format, such as .STP

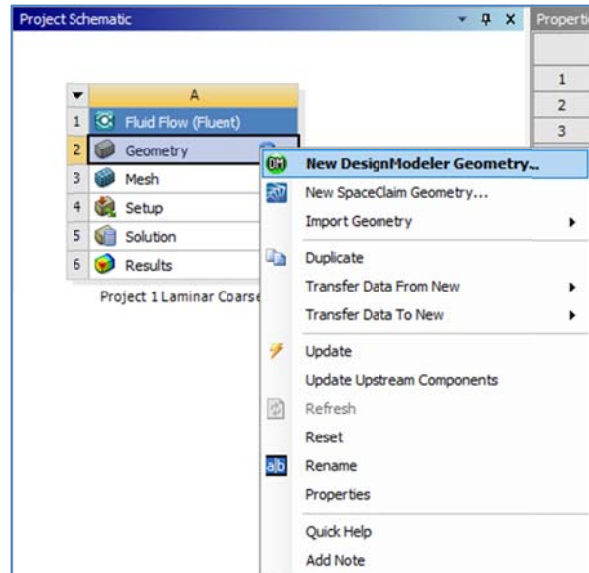


Figure 1.5

## 2 ANSYS Design Modeler

### 2.1 Creating a New Sketch

1. In the **Tree Outline** (left menu), *click XYPlane*
2. In the **Top Menu**, *click the New Sketch icon*. **Sketch 1** will appear in your **Tree Outline** under **XYPlane**
3. On the **Coordinate System** in the bottom-right corner, *click the +Z-axis* to snap your view to 2D in the XY plane. You can also familiarize yourself with the zooming and panning functions.
4. To begin drawing the geometry, select the **Sketching** tab at the bottom of the **Tree Outline**. Note that the **Sketching Toolboxes** menu will replace your **Tree Outline**.

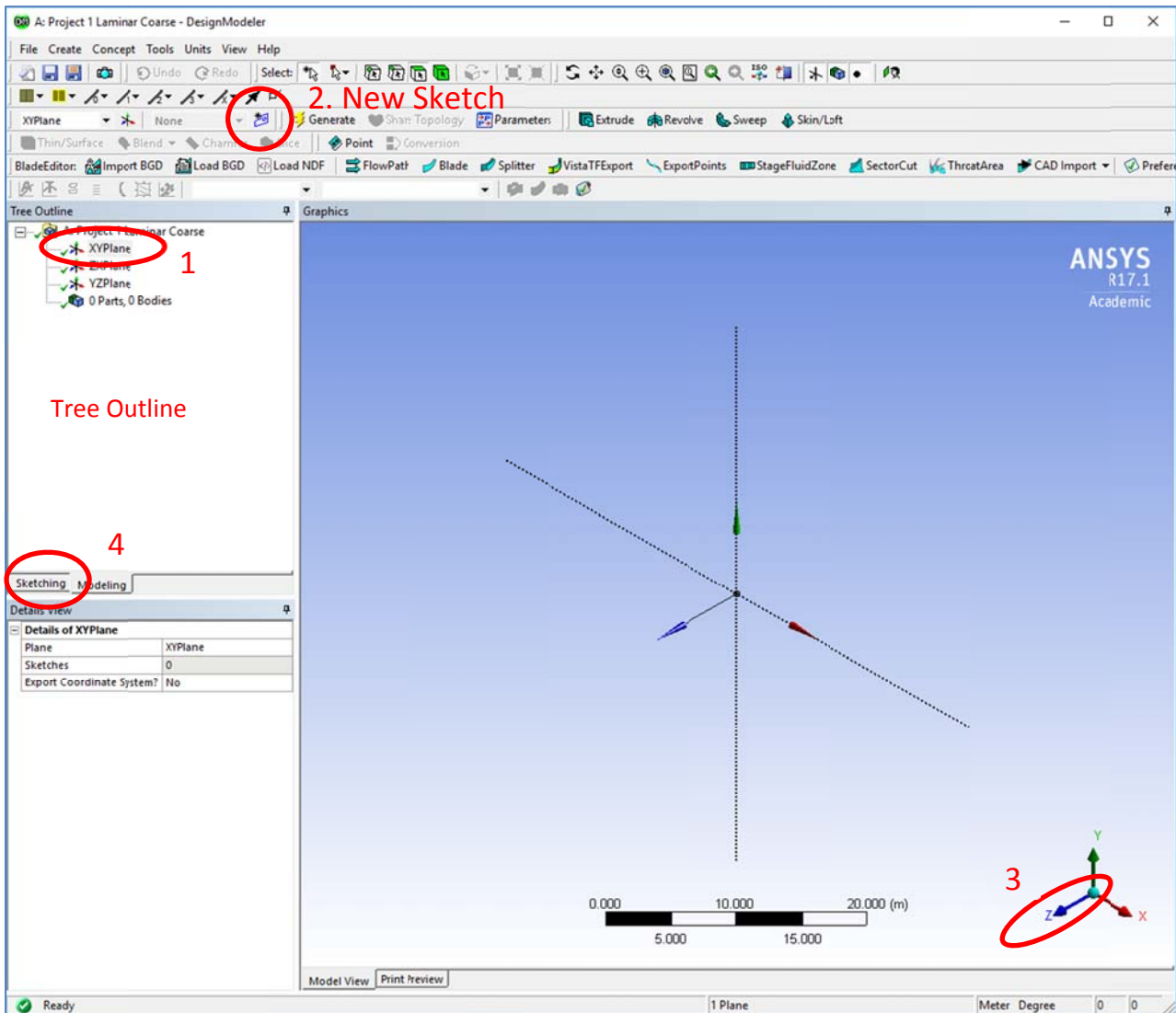


Figure 2.1

## 2.2 Drawing the Geometry

1. In the **Sketching Toolbox**, under **Draw**, click **Rectangle**. Your cursor will become a pencil.
2. Create the rectangle of any size anywhere in Quadrant 1 of your drawing canvas. We will constrain the location and size next.

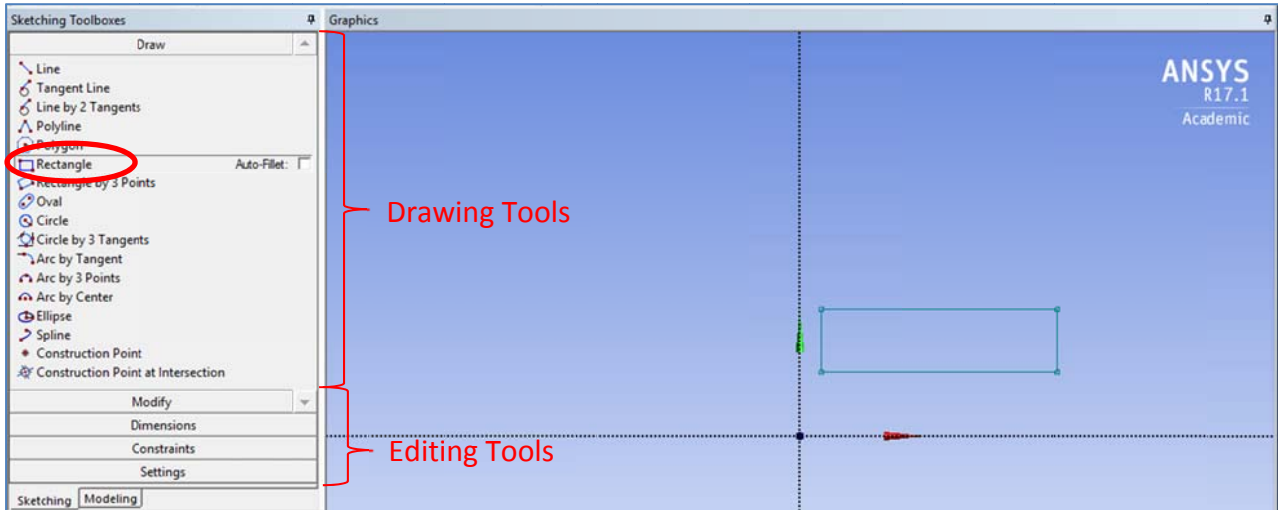


Figure 2.2

## 2.3 Constraining Geometry to the Axis

1. In the **Sketching Toolbox**, select the **Constraints** dropdown.
2. Select **Coincidence**. This will set a rule for an edge to coincide with another.
3. Click the edge of the rectangle you want to constrain and then select the axis you want to lock it to. The edges of the rectangle will change color if they are fully constrained.
4. Do this for both edges of the rectangle, so that its lower left corner is at the origin (0, 0).

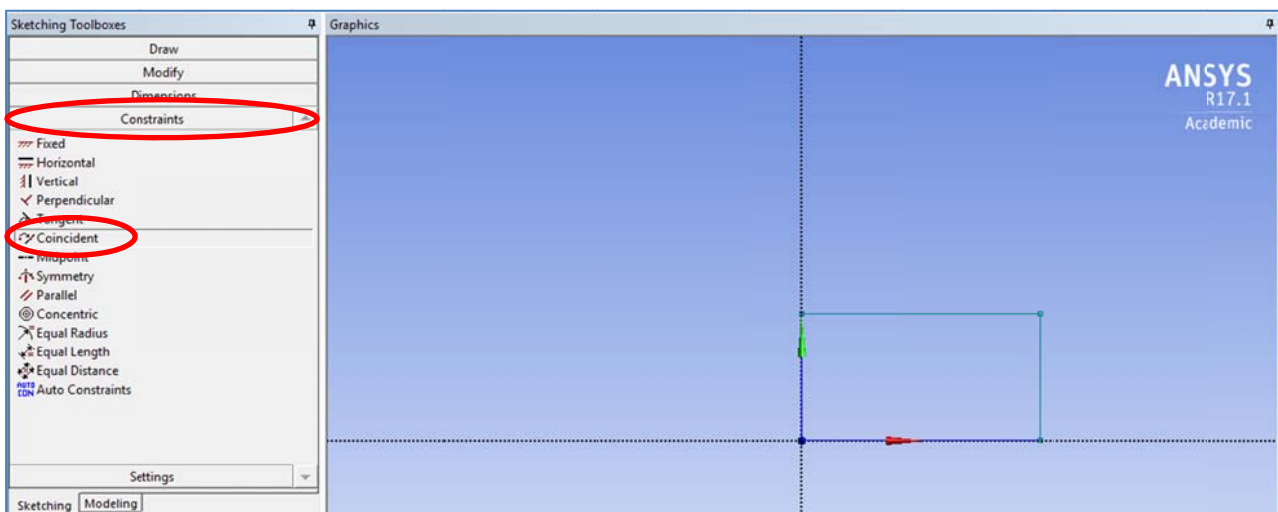


Figure 2.3

## 2.4 Dimensioning the Geometry

1. In the Sketching Toolbox, select the Dimensions dropdown
2. Select **General**. You can also use **Horizontal** or **Vertical** for their respective lines.
3. Select the edge of the rectangle you want to dimension. *Click* above or next to that edge to place the dimension label.
4. When you create a dimension, it will show up in the **Details View** menu. You can edit these values and it will adjust the size of the rectangle.
5. Create a dimension for the horizontal and vertical lines of the rectangle and set their values to  $H1 = 0.1\text{ m}$  and  $V2 = 0.02\text{ m}$ , respectively.

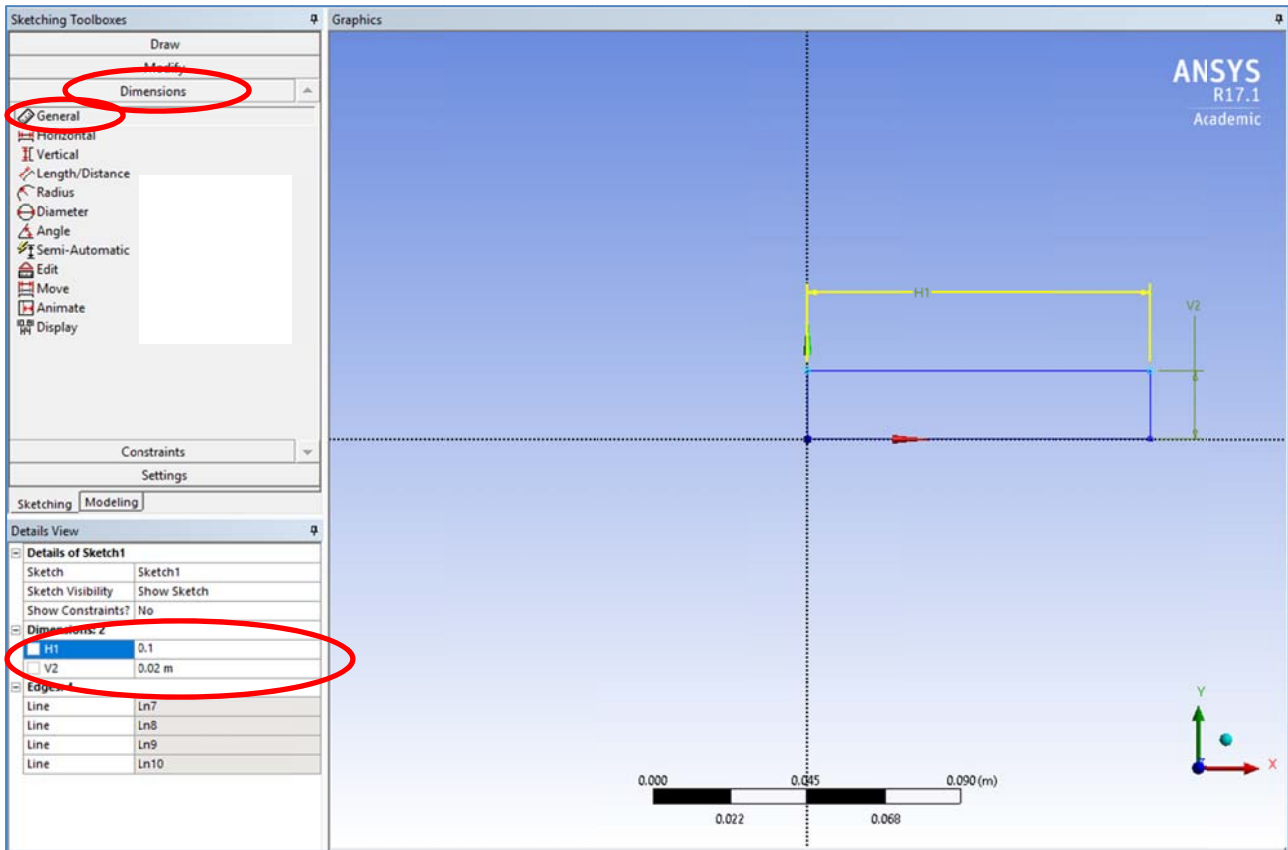


Figure 2.4

## 2.5 Creating a Surface

1. Click back to the **Modeling Tab** at the bottom of the **Sketching Toolbox**.
2. On the top menu, open the **Concept** dropdown and select **Surface from Sketches**.
3. Select your sketch and then click **Apply** in the **Details View** (bottom left menu).
4. Click the **Generate** button in the top menu. Your part will fill gray when the surface is generated.
5. You are now done with creating your geometry. Click **Save** and close **Design Modeler**.

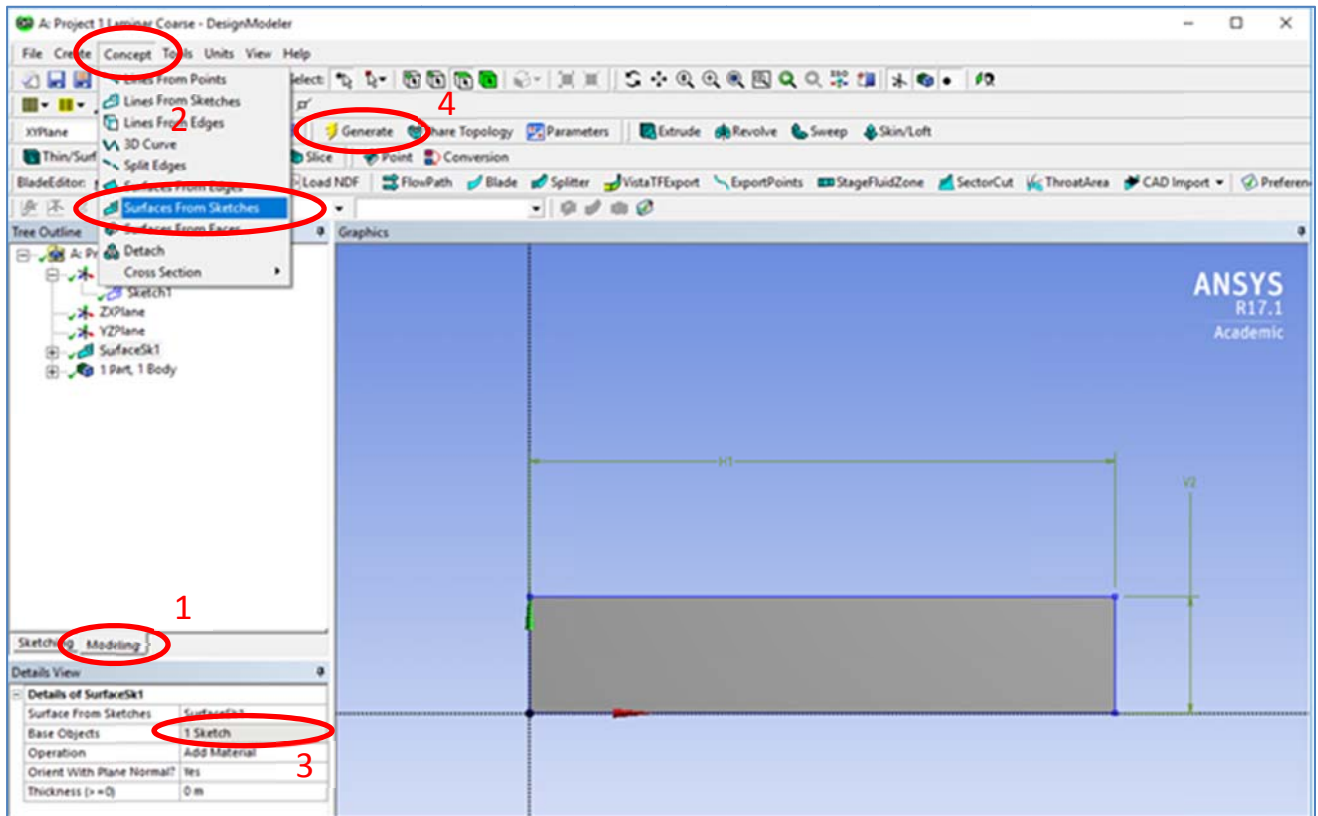


Figure 2.5



## 3 Meshing

### 3.1 Open Meshing

1. Go back into **ANSYS Workbench**.
2. Note that you should see a *checkmark* next to **Geometry** in your **Project Box**.
3. **Save** the Project.
4. In the **Project Box**, *right-click* **Mesh** and select **Edit...**
5. The **Mesh Editor** will open and your **Geometry** should automatically load.

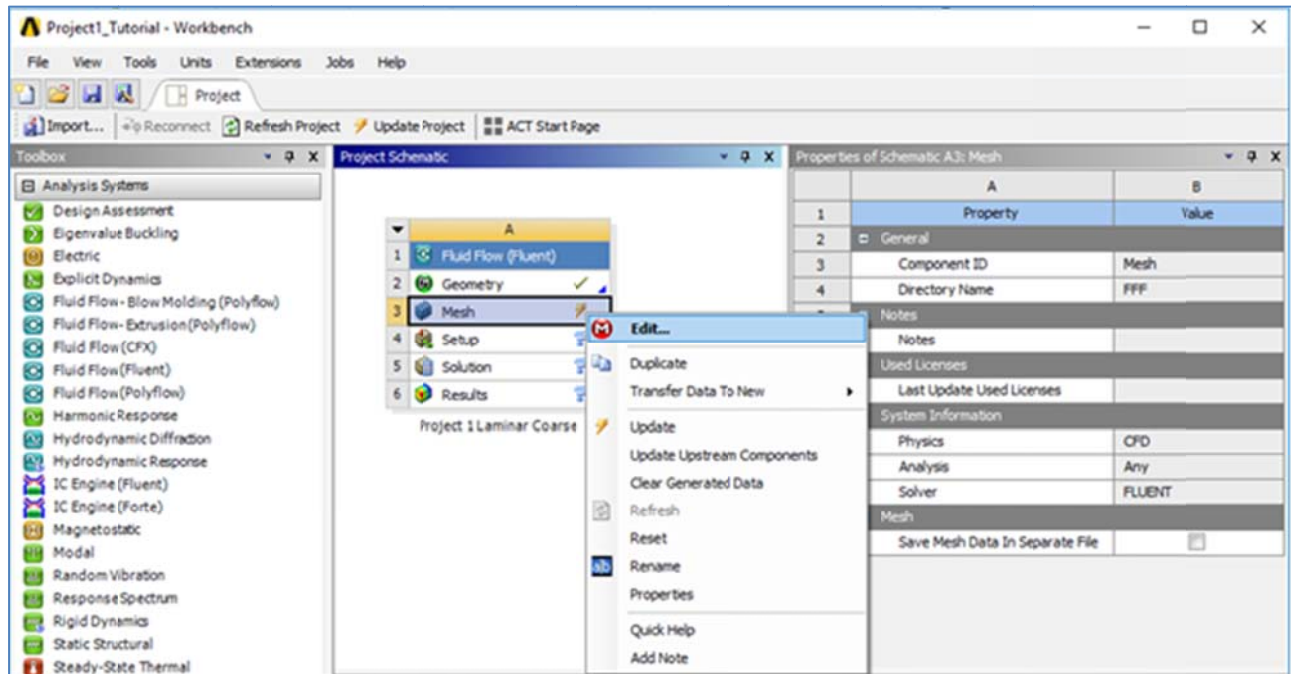


Figure 3.1

### 3.2 Set up Meshing

1. With the **Mesh Editor** open, orient your view, if necessary, with the **coordinate system** and **Zoom to Fit** (button in the Top Menu with a Cube inside a Magnifying Glass). You can also familiarize yourself with the zooming and panning functions.
2. In the **Outline menu** (left side), *expand* out the **Geometry** tree (plus sign) and *click* **Surface Body**
3. The **Details Menu** (bottom left) will change to properties of the **Surface Body**
4. Under the **Definition** tab, change the **Thickness** to **0 m**.
5. Under the **Material** tab, change the **Fluid/Solid** to **Fluid**.

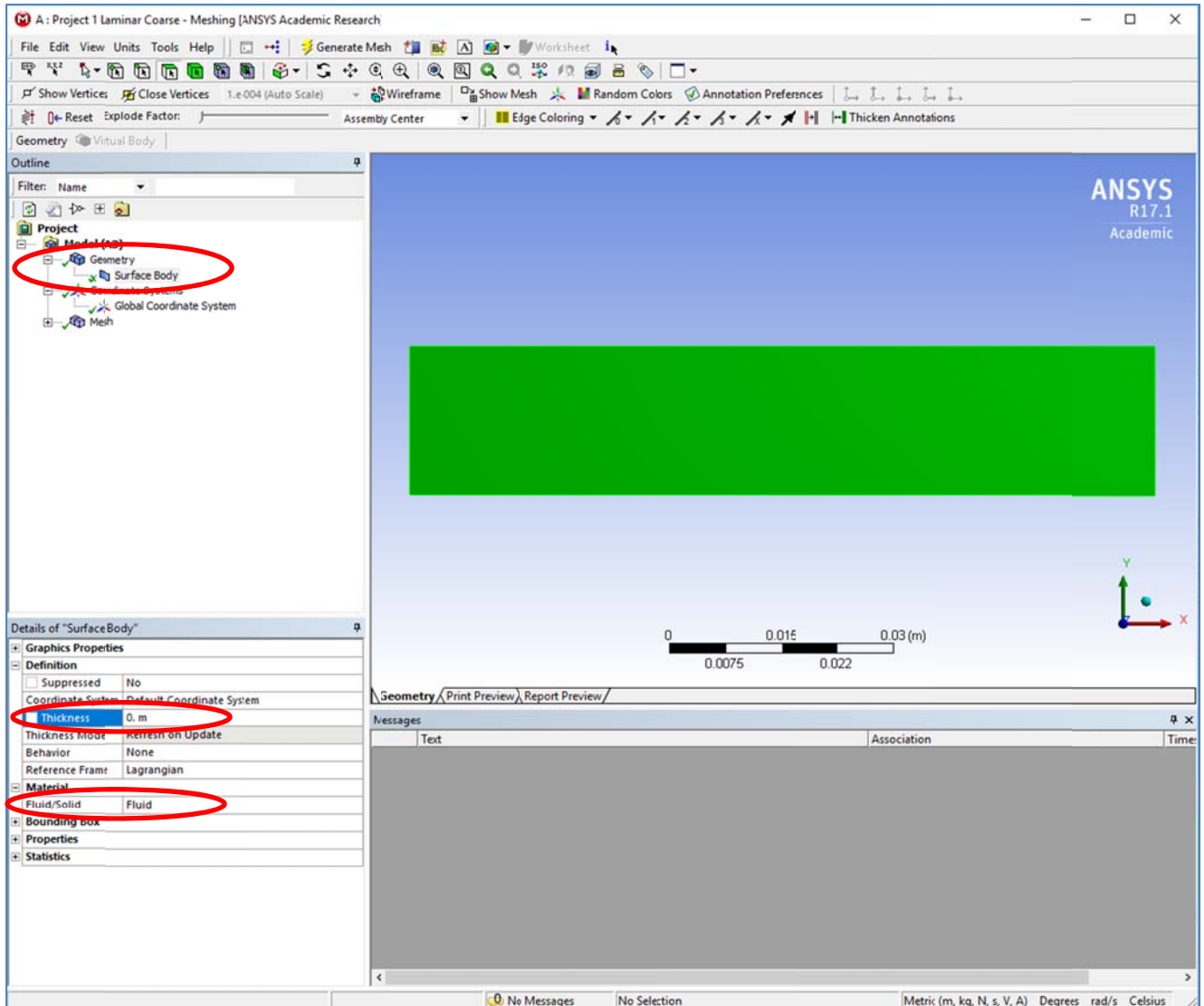


Figure 3.2

### 3.3 Set Mesh Resolution

1. In the **Outline** menu, *click* the **Mesh** tab.
2. The **Details** menu will change to properties of the **Mesh**.
3. Set **Min Size**, **Max Face Size**, and **Max Tet Size** all to  $1e - 3$  m.
4. This will set your grid size to 1 mm spacing.

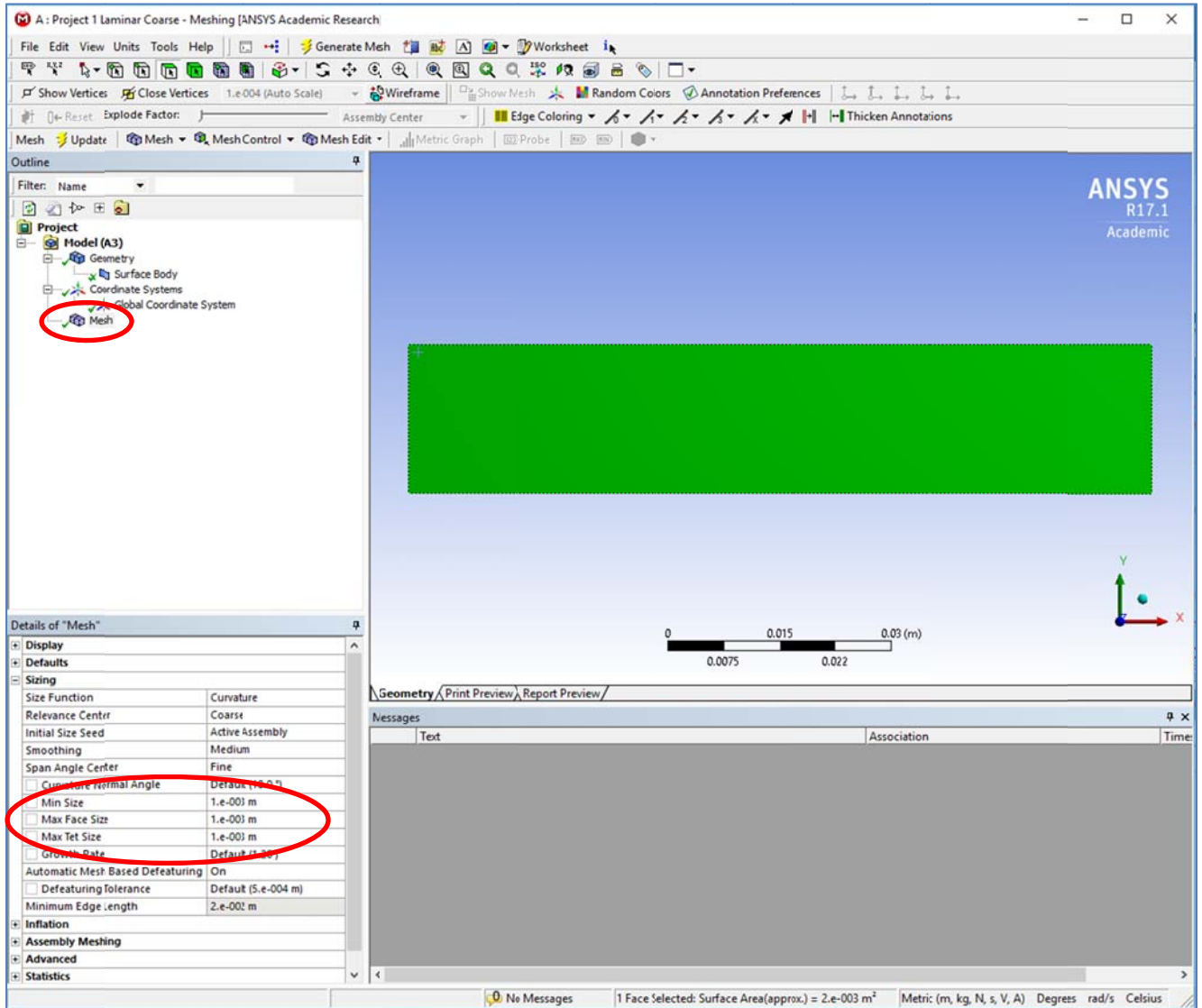


Figure 3.3

### 3.4 Set Mesh to Quadrilaterals

1. Familiarize yourself with the types of selection tools. **There is Vertex Selection, Edge Selection, Face Selection, and Body Selection.**
2. Use the **Face Selection** tool to select the face of the **geometry**
3. Once it is selected (the face turns green), *right-click* the **geometry** and **Insert Mapped Face Meshing**
4. A **Face Meshing** will appear in your **Outline** menu. Once it is selected, the **Details** menu will show its properties.
5. In the **Details Menu**, make sure **Method** is set to **Quadrilaterals**.
6. In the **top menu**, click **Generate Mesh**. If you click the **Mesh** icon in the **Outline menu** again, your mesh should appear over your geometry.

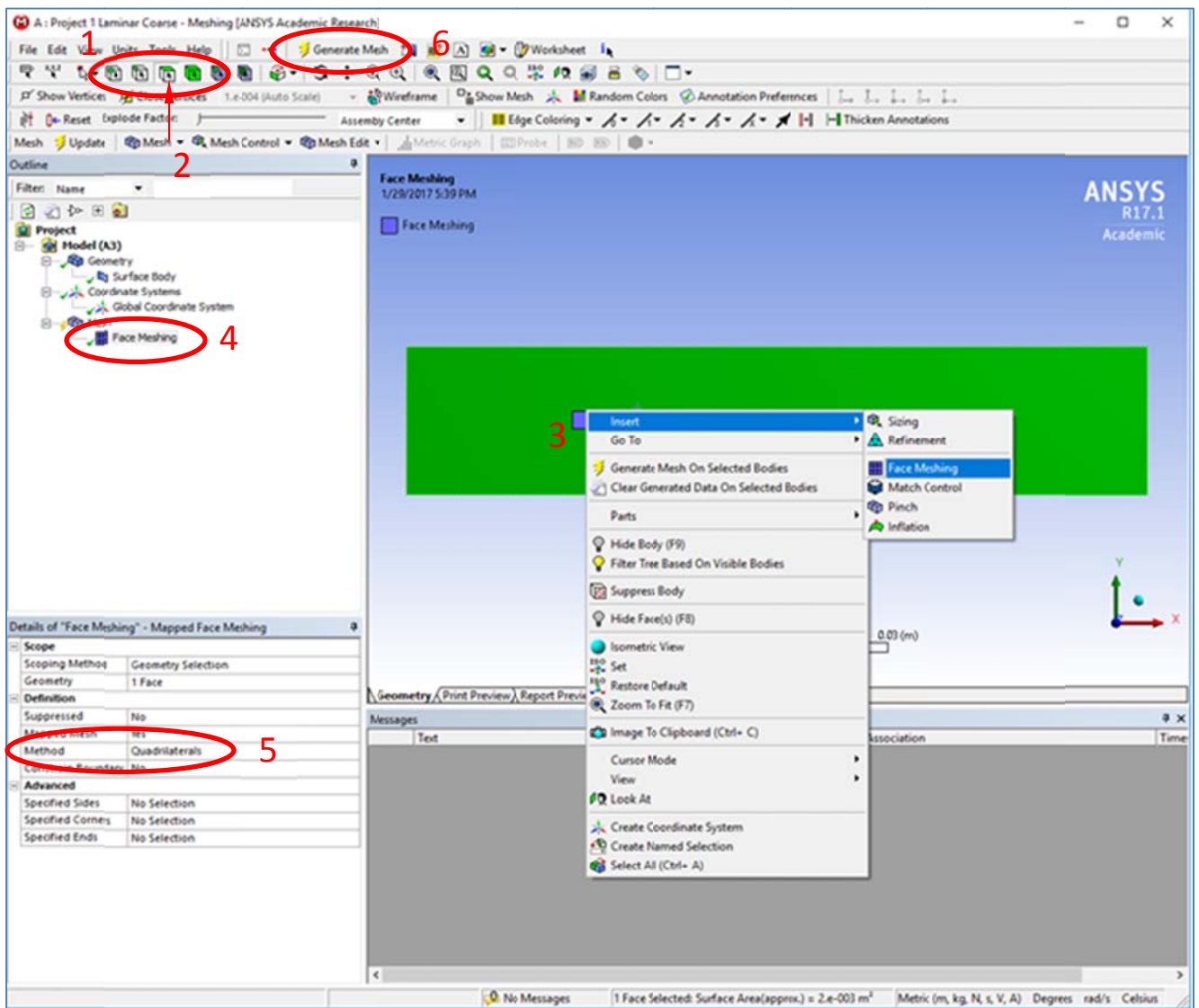


Figure 3.4

### 3.5 Boundary Labeling

1. In the **Top Menu**, use the **edge selection** tool to **select the left edge** of your **geometry**
2. *Right-click* the **left edge** and select **Create Named Selection**
3. **Name** this edge *inlet*.
4. **Repeat the process** for the other edges, naming the *wall\_top*, *wall\_bottom*, and *outflow*, respectively.
5. This naming process helps Fluent determine Boundary Conditions.
6. Click File, **Save Project**, and return to **ANSYS**. If you want to create a backup file of the Mesh, you can select File, Export, and change the file type to .msh. If for some reason, your Workbench Project becomes corrupted, you can resume from this point by importing the .msh file to a new Project. Note that you will not be able to edit your .msh file, so inspect that it is correct before exporting.

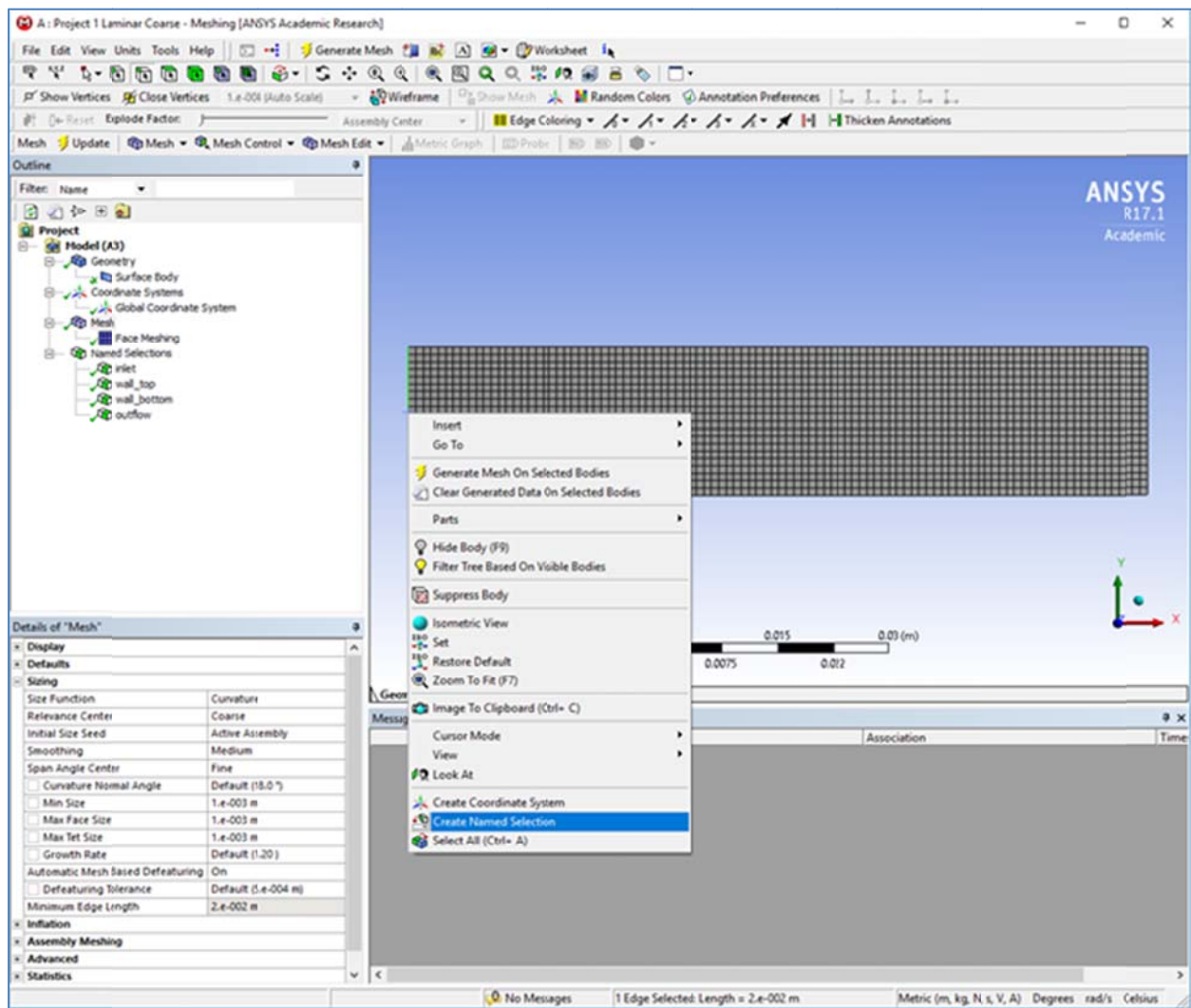


Figure 3.5

## 4 Fluent

### 4.1 Updating Project

1. Back in **Workbench**, you may notice a **Lightning Bolt** next to the **Mesh** in your **Project Box**. This means that your **Mesh** has not been updated yet.
2. To **update** the **Mesh**, *right-click Mesh* and select **Update**. Once it is done, a **green checkmark** should appear.

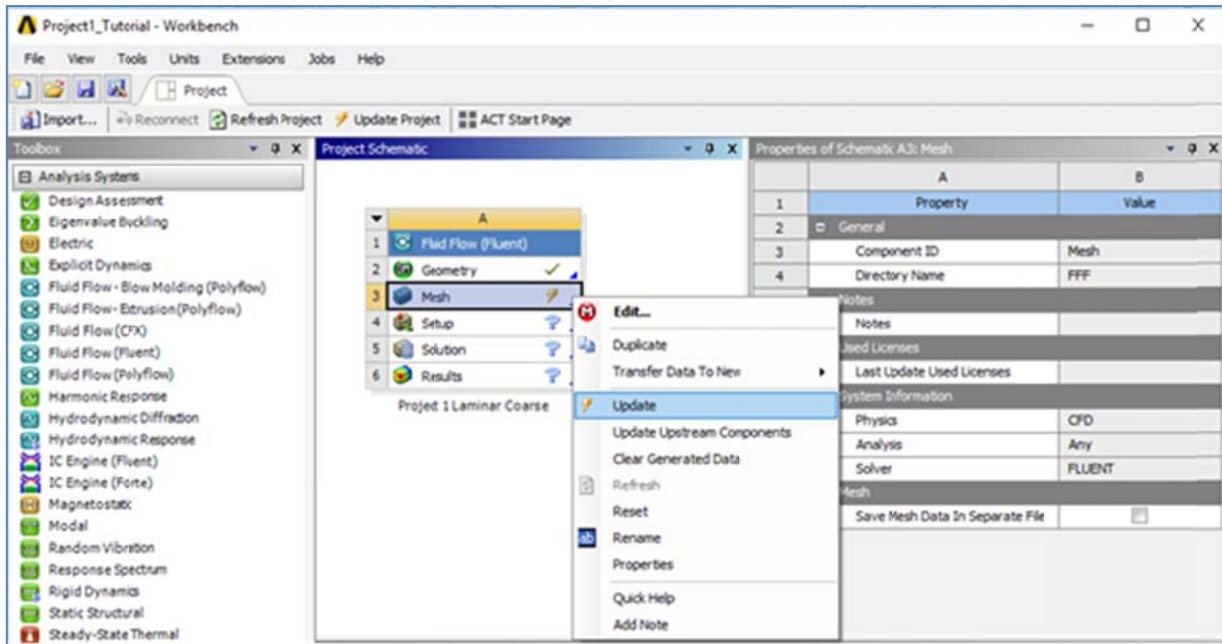


Figure 4.1

### 4.2 Starting Fluent

1. To open **Fluent**, *double-click Setup* in the **Project Box**. A dialog will pop up with settings. Select the following options and click **OK**.

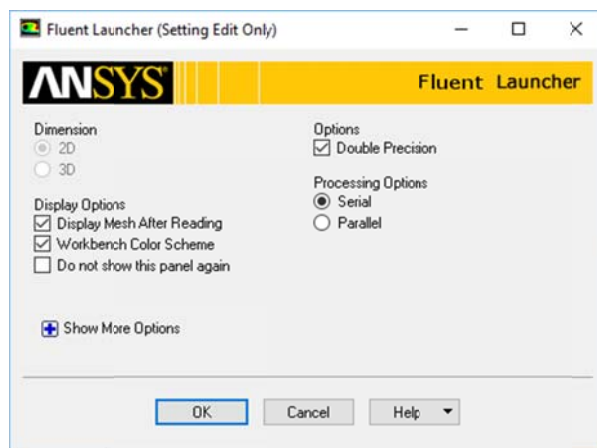


Figure 4.2

### 4.3 Fluent Layout

1. **Fluent** is segmented into several menus.
  - a. **Top Ribbon** – Adjusting properties
  - b. **Tree** – Each step of setting up the simulation
  - c. **Task Page** – The setup of variables and properties in each step of the Tree
  - d. **Main Window** – View the domain, simulation progress, or results
  - e. **Console** – The command line showing progress or errors.
2. Operating Fluent will mainly consist of running down the **Tree** and editing variables in each respective **Task Page**.

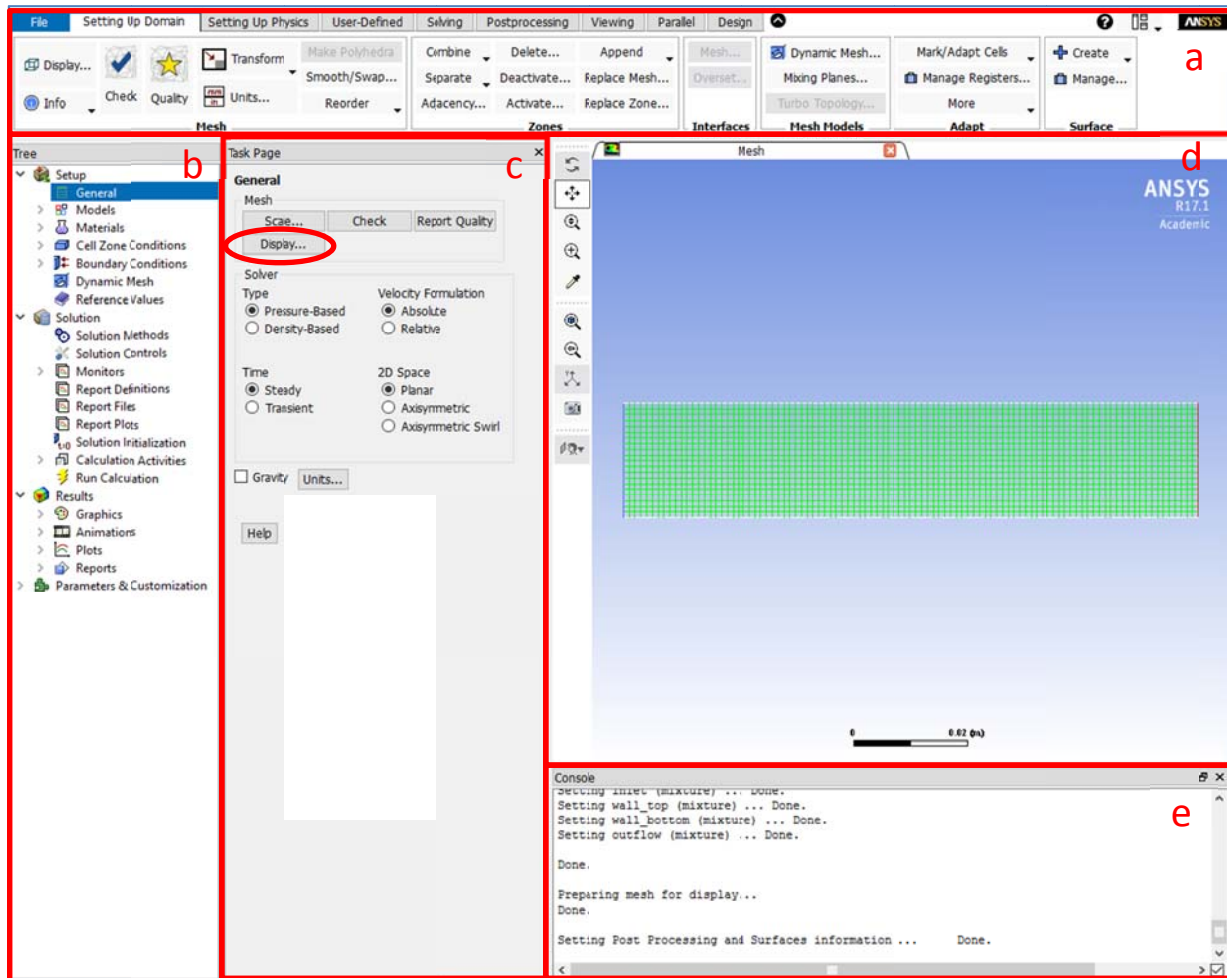


Figure 4.3

## 4.4 Setup – General

Note: Recommended selections for Project 1 are in **Bold and Underlined**

1. Mesh
  - a. Once Fluent opens, your geometry and mesh should automatically display. If it doesn't, you can manually display components by clicking **Display** in the **Task Page**.
  - b. If you need to **scale** the geometry (e.g. you made the geometry in inches and you want the solution in SI units), you can also do it here.
  - c. You may also **Check** the mesh for errors and quality here.
2. Solver
  - a. Type
    - i. **Pressure-Based** for low-speed incompressible to high-speed compressible flows
    - ii. Density-Based reserved for flows with strong interdependence between density, energy, momentum, or species.
  - b. Velocity Formulation
    - i. **Absolute** is where majority of the domain is non-rotational
    - ii. Relative may be useful in cases where the domain rotates
  - c. Time
    - i. **Steady** solves for a steady-state (developed) solution
    - ii. Transient is used if the solution is a function in time
  - d. 2D space
    - i. **Planar** is your standard 2D space in Cartesian coordinates
    - ii. Axisymmetric revolves the domain around an axis (e.g. cylindrical coordinates)
    - iii. Axisymmetric swirl revolves the domain around an axis with some tangential velocity
3. Gravity can include buoyancy effects for flows or gravitational settling in particle tracking.

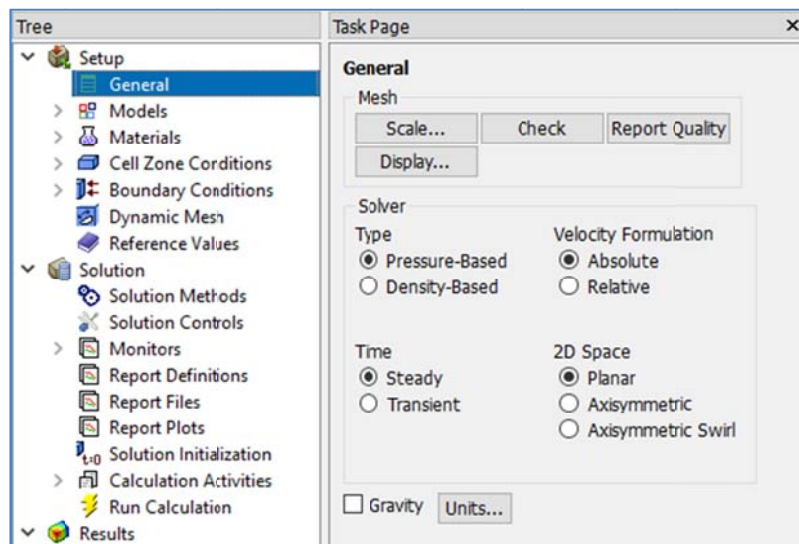


Figure 4.4



## 4.5 Setup – Models

Descriptions:

1. **Energy tracking** is used for heat transfer solutions and things that are a function of temperature (e.g. diffusion, air density, relative humidity)
2. **Viscous Model** determines how you solve/simplify the Navier-Stokes equations. For Laminar flow, simply choose the **Laminar model**. For Turbulent flow, you have several options that approximate a solution. The most common is k-epsilon because of its simplicity.
3. **Species tracking** is when you would simulate a species (e.g. relative humidity) mixing/diffusing
4. **Discrete Phase** is used for particle tracking within the fluid.

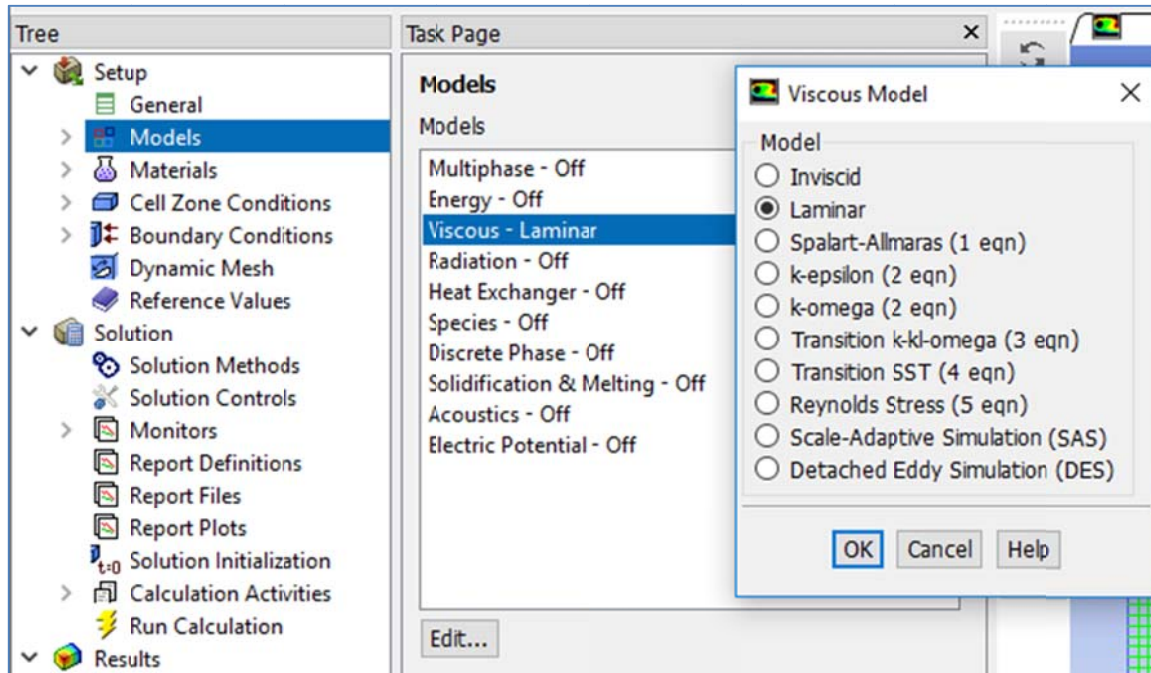


Figure 4.5

## 4.6 Setup – Materials

If you want to change environmental conditions, such as the working fluid (e.g. air, water, helium) or material surface roughness (affects skin friction), you can do it in this tab

Default properties are fine.

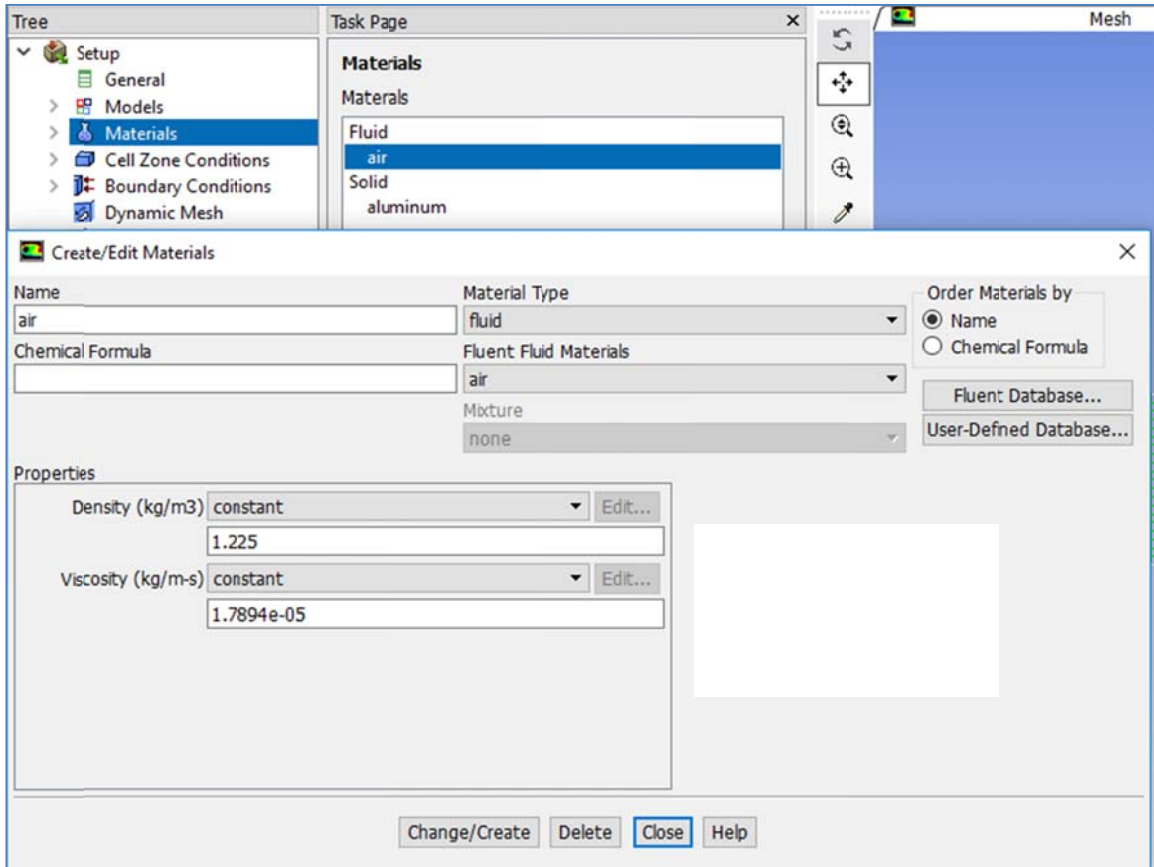


Figure 4.6

Note that Cell Zone Conditions will be skipped over (defaults are fine).

## 4.7 Setup – Boundary Conditions

This is where we'll define the boundary conditions for the simulation. Note that the edges you labeled during meshing correspond to the boundaries found in fluent. Select a zone and click Edit...

1. **Inlet** – Set your **Velocity Magnitude** to  $0.1\text{ m/s}$
2. **Outflow** – default weighting of  $1$ . If you have multiple outflows, you can scale the proportion of the mass flow that exits each outflow (e.g.  $0.2$  outflow 1,  $0.8$  outflow 2)
3. **Wall** – Both default *stationary wall* with *no slip*. For other cases, you may want a moving wall.

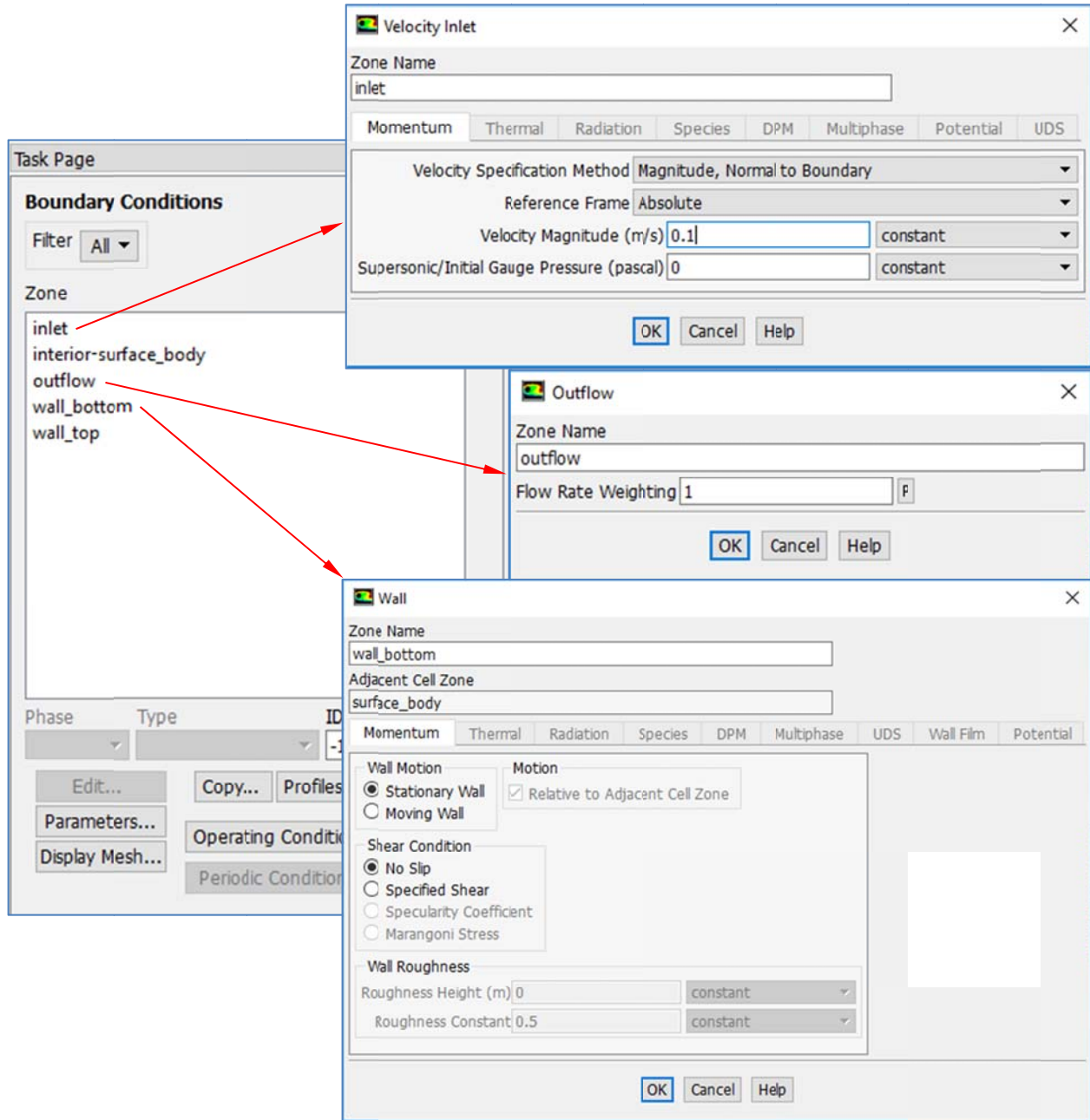


Figure 4.7

## 4.8 Solution – Solution Methods

1. Scheme
  - a. **SIMPLE** – (Semi-Implicit Method for Pressure-Linked Equations) Default, robust.
  - b. SIMPLEC – (SIMPLE-Consistent) Allows faster convergence for simple problems (e.g., laminar flows with no physical models employed)
  - c. PISO – (Pressure-Implicit with Splitting of Operators) For unsteady flow problems or mesh with high skewness
2. Gradient
  - a. **Green-Gauss Cell Based** – Default method, solution may have false diffusion/smearing
  - b. Green-Gauss Node Based – Minimizes false diffusion, recommended for tri/tet mesh
  - c. Least Squares Cell Based – For polyhedral meshes, same accuracy, properties as node
3. Pressure
  - a. **Standard** – The default scheme. Reduced accuracy for flows with large surface-normal pressure gradients near boundaries (but should not be used when steep pressure changes are present in the flow – PRESTO! scheme should be used instead)
  - b. PRESTO! – For highly swirling flows, steep pressure gradients, or curved domains
  - c. Linear – When other options result in convergence difficulties or unphysical behavior
  - d. Second-Order – Use for compressible flows
  - e. Body Force Weighted – When forces are large e.g. high Ra convection, swirling flows
4. Momentum
  - a. First Order Upwind – Easiest convergence, only first order accurate, and very diffusive i.e. dissipates large gradients and can suppress small/sharp features
  - b. **Second Order Upwind** – 2<sup>nd</sup> order accuracy, essential when flow is not aligned with mesh grid direction e.g. during tri/tet mesh

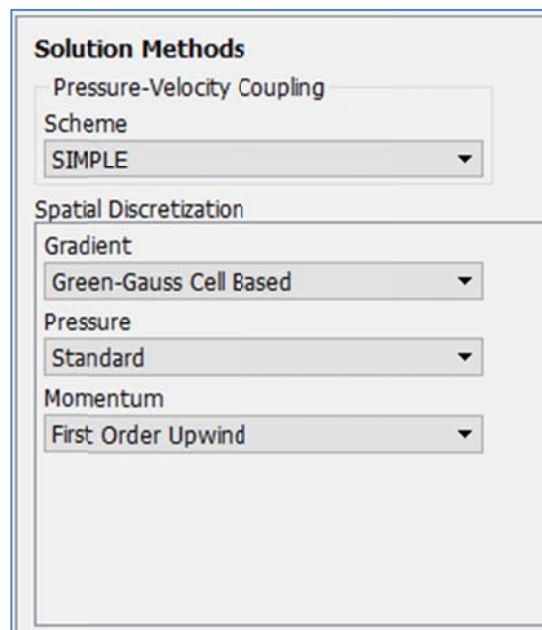


Figure 4.8

#### 4.9 Solution – Monitors

1. Select **Residuals** and *click Edit...*
2. Under **Residual Values**, *check the box* that says **Normalize**. This scales the last iteration of Residuals by the maximum Residuals and can be used as a better gauge for convergence.
3. Under **Convergence Criterion**, select **None**. The Convergence Criterion tells Fluent when the solution has converged to a value that you determine to be accurate enough and stops calculating the solution early. By turning this feature off, Fluent will continue calculating the solution until it reaches the number of iterations you set.

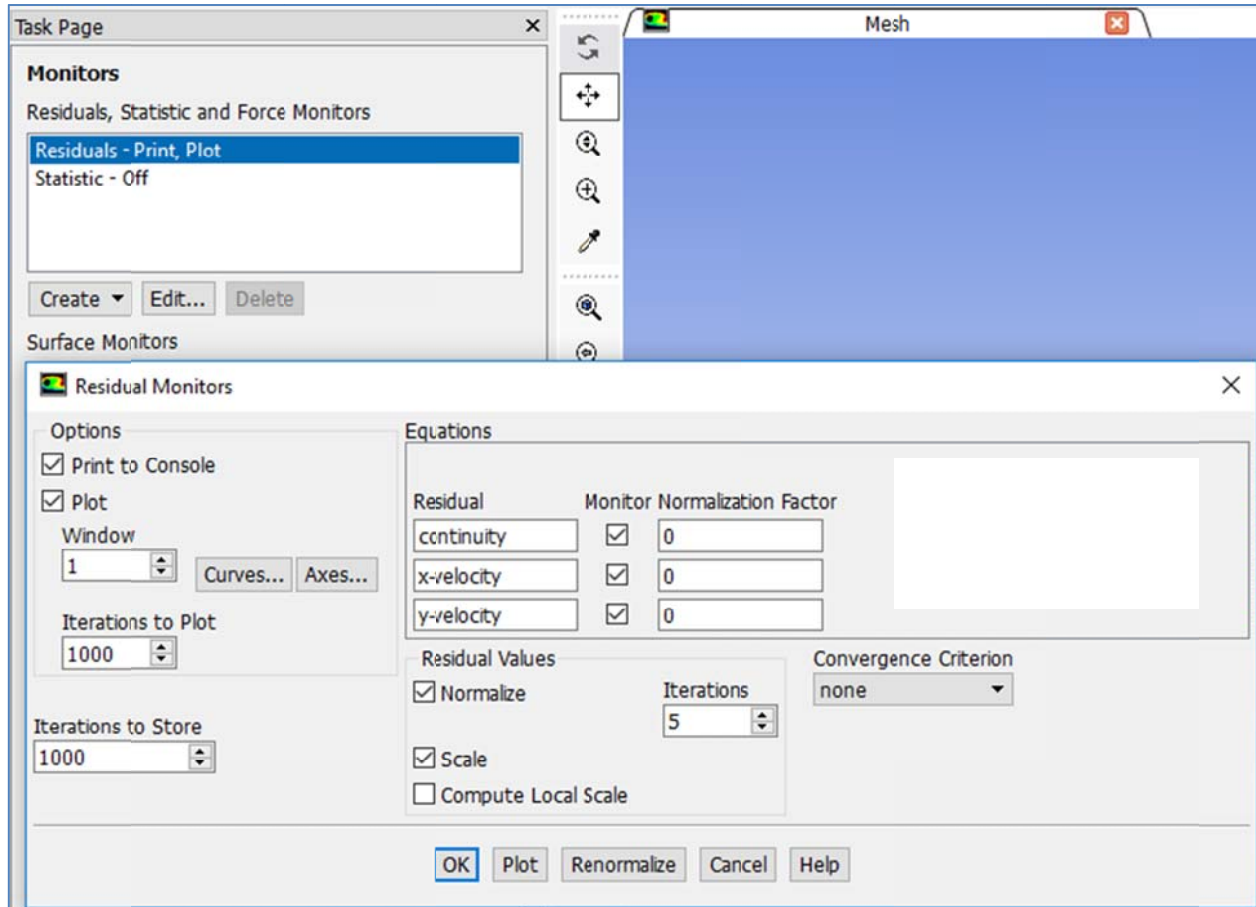


Figure 4.9

#### 4.10 Solution – Solution Initialization

Select your initialization technique, the boundary to initialize with, and then *click Initialize*

1. **Standard Initialization** – This sets all mesh cells to a single starting value. Typically, you select values that are the closest to what you think the final solution will look like to help the solution converge faster. **Compute from Inlet** takes inlet conditions (from the boundary conditions tab you set earlier) and applies it to the whole mesh as a starting guess. The calculations then refine the flow to a converged steady-state solution
2. Hybrid Initialization – Provides a quick approximation of the flow field by solving Laplace’s equation to estimate a flow field and pressure field.

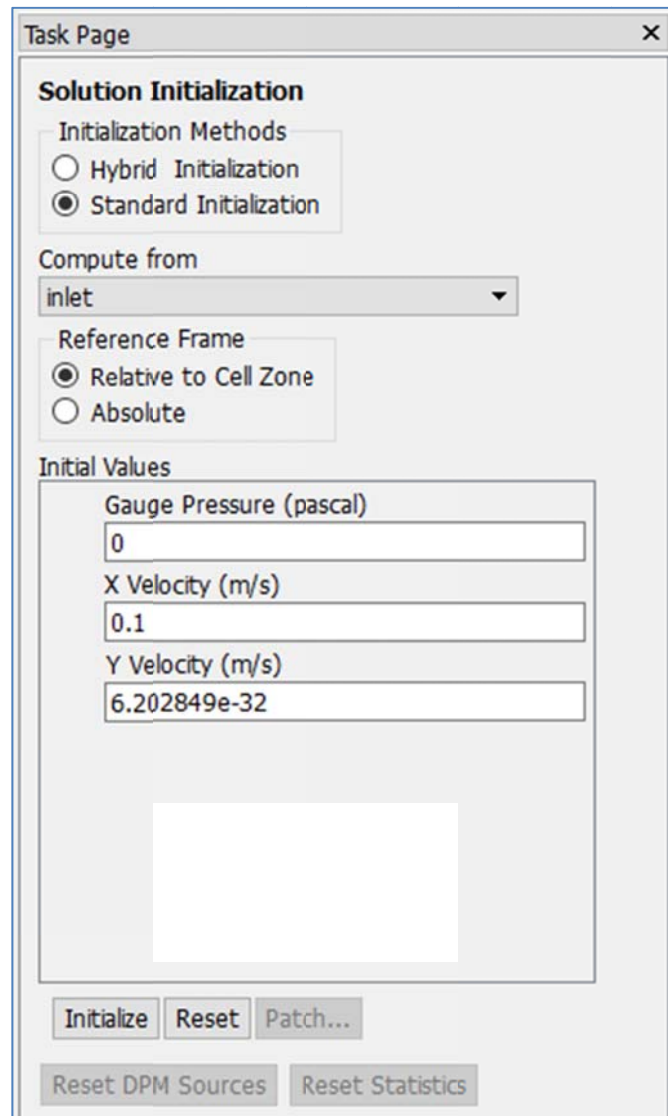


Figure 4.10

#### 4.11 Solution – Run Calculation

1. **SAVE YOUR WORK BEFORE PROCEEDING** (sometimes Fluent crashes). File, **Save Project**
2. **Check Case** to make sure everything is good to go (your mesh, boundary conditions, etc.)
  - a. Note: Since I selected First order upwind earlier, my Fluent recommended using a higher order Momentum Solver (second order upwind) for improved accuracy. This may be a good suggestion, so I went back to solution methods and changed it. It also suggested Least Squares for Gradient. You can make that change too.
3. Set the **Number of Iterations** to **1000**. You can always stop the calculation early. Sometimes I'll stop at around 200 and check that the flows/magnitudes make some sense before spending all that time calculating to find out I made a mistake setting the boundary conditions.
4. **Click Calculate**

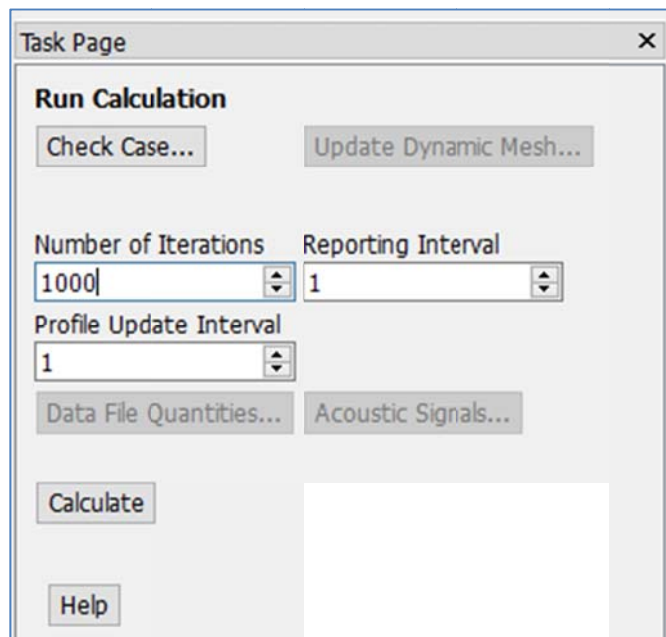


Figure 4.11

## 4.12 Calculating Solution

Once the program calculates, you'll see a plot of the residuals (errors) for each of the equation parameters (in this case, continuity, x-velocity, y-velocity).

Note that my solution converged after approximately *400 iterations* to the minimum error.

Also note that the Console prints the residuals corresponding to each iteration.

Once you have a solution calculated, save your work again.

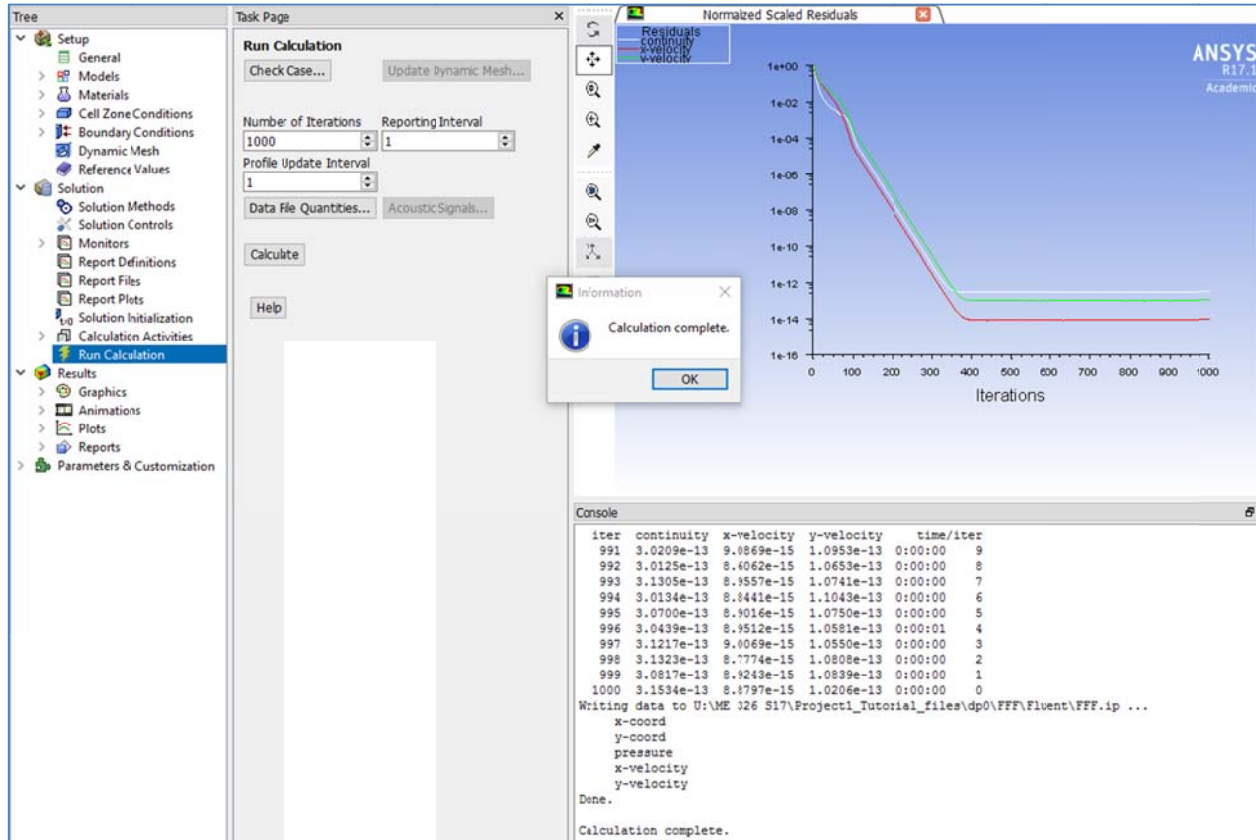


Figure 4.12