# ME 24-688 – Week 1
Unsteady External Fluid Flow Analysis

1 Basics of Fluid Flow Analysis

1.1 Turbulence

By default, when turbulence is included in a fluid flow analysis, the Smagorinsky's Subgrid Scale (SGS) model of the Large Eddy Simulation (LES) model is used. In addition to this method, the standard two-equation k-epsilon turbulence model is available. This model includes a wall function and will consider surface roughness. This model is currently available for 3-D steady or unsteady fluid-flow analyses using either the Segregated or Mixed GLS formulation. In addition, when the Mixed GLS formulation is being used, the "**Use implicit time integration**" option must also be active. This option is active by default and is accessed via the steady or unsteady "**Fluid Flow Formulation Options**" dialog. To reach this dialog, click the "**>>**" button to the right of the "**Formulation**" field within the "**Solution**" tab.

The k-epsilon model can be specified in the same manner as the previous method. In the "**Load Curves**" tab of the "**Analysis Parameters**" dialog, type a "**1**" in the "**Turbulence**" column of the table for the time steps in which you want turbulence to be included as shown in row 2.

|	Multiplier	Steps	Max Iterations	Turbulence	Upwind	Residual Type	Residual Tol	Increm
	1	1	15	0	1	1	0.0001	0
✎	1	100	15	1	1	1	0.0001	0
	1	50	15	0	1	1	0.0001	0

Custom Load-Stepping Settings

In the "**Turbulence**" tab, select the "**k-epsilon**" option in the "**Turbulence model selection**" drop-down box. The initial conditions need to be specified for the k-epsilon turbulence model in the "**Initial Conditions Based On**" section. The "**Characteristic Inlet Length Scale**" and "**Characteristic Inlet Velocity Scale**" set the initial conditions for turbulence kinetic energy and dissipation rate in solving the k-epsilon equations. These values can be found by checking the diameter or the hydraulic diameter of the inlet section, and the specified velocity value at the inlet. If pressure is specified at the inlet surface, estimation could be found either by the knowledge of the user about the application or by solving the problem using the LES turbulence model for a couple of steps.

The default "**Roughness Height**" and "**Roughness Constant**" are assigned to any fixed wall that does not have a part or surface roughness assigned to it. The values in the "**Model Constants**" section are variables used in the k-epsilon equation. These values are determined from experimental data. The defaults are the values used by the standard k-epsilon model. Refer to the Help documentation for information on how each variable is used in the equation.

1.2 Surface Prescribed Turbulence Conditions

Turbulence can be added to the inlet or outlet surfaces of a model in the display area. If you select a surface, right-click and select the "**Surface Prescribed Turbulence Conditions…**" command. The dialog shown below will appear.
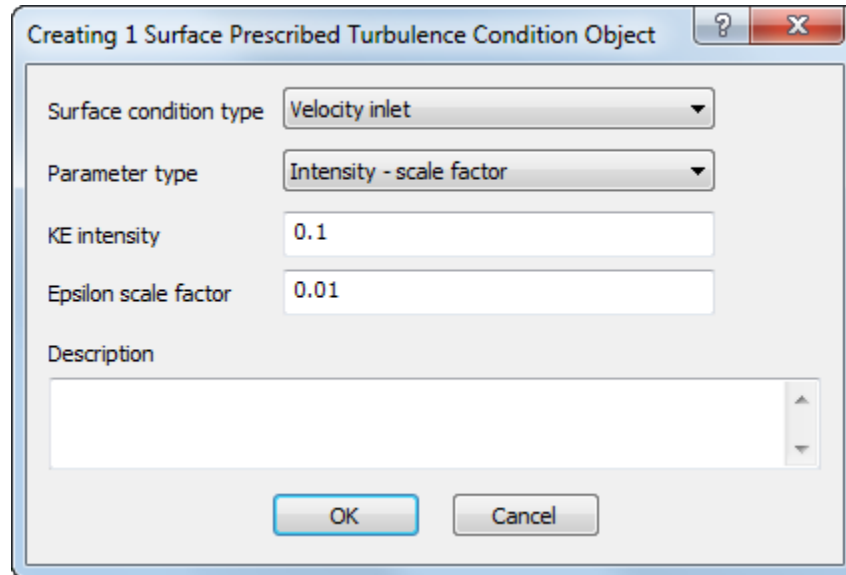
First you must specify whether the turbulence at the surface is driven by a pressure or a velocity in the "**Surface condition type**" drop-down box. The "Velocity inlet" is the preferred option and is required for the inflow boundaries when any velocity, pressure, or inlet/outlet conditions are specified. The "P**ressure outlet**" condition is optional and is only applicable when back flow (that is, flow back into the model) occurs at any nodes along a pressure outlet.

Next, you must choose how you want to specify the conditions in the "**Parameter type**" drop-down box. If the "**Intensity – scale factor**" option is selected, the inlet conditions are specified with the "**KE intensity**" [the turbulent kinetic energy intensity (I)] and "**Epsilon scale factor**" [the turbulent kinetic dissipation scale factor (SF)] fields. The "**KE intensity**" is used to determine the inlet turbulent kinetic energy (k) by $k = 1.5 * (I * U_{inlet})^2$. The "Epsilon scale factor" is used to determine the inlet turbulent kinetic energy dissipation rate (Epsilon) by, $Epsilon = C_{mu} * (K^{1.5})/(S_F * L)$ where L is the characteristic length scale at the inlet, and C_{mu} is the turbulent model constant with a value of 0.09.

If the "**KE and epsilon**" option is selected from the "**Parameter type**" drop-down box, the inlet conditions are specified directly with the turbulence kinetic energy (k) and the turbulence kinetic energy dissipation rate (Epsilon). The values for "**Turb. kinetic energy**" and "**Turb. dissipation rate**" can be obtained from either upstream flow conditions, experiments, or estimation based on the main flow conditions at the inlet. For example, K and epsilon might be estimated by, $K = C_{bc} * (U_{inlet})^2$ and $Epsilon = C_{mu} * (K^{1.5})/L_O$ where Cbc is a constant between 0.003 and 0.01, and L_O is a default mixing length value at the inlet with a value between the minimum admissible and maximum admissible eddies.

1.3 Wall Roughness

The k-epsilon model supports wall roughness. The wall roughness can be applied on two levels in the user interface. The settings can be defined in the "**Default Wall Roughness Settings**" section of the "**Turbulence**" tab of the "**Analysis Parameters**" dialog. Secondly, if a surface is selected, you can right-click and select the "**Add: Surface Prescribed Wall Roughness…**" command. Each surface prescribed roughness that the user defines will override the default value for the selected surface(s).

When any of these methods is used you will need to specify the following values.

Roughness Height
This is the height of the surface irregularities (typically designated as Δ) for uniform sand-grain roughness, or a mean height value for non-uniform sand-grain roughness. For other commercial pipes, the "**equivalent sand-grain roughness**" height can be used, and the data can be found in most fluid and piping handbooks. The default value of 0.0 indicates that the surface is treated as a smooth wall.

Roughness Constant
This is the non-dimensional constant which is used to consider the effect of the non-uniform nature of surface roughness for real world pipes. For uniform sandgrain surfaces, it takes a value of 0.5 as the default. A higher value between 0.5 and 1.0 might be chosen to consider the effect of non-uniformity of the wall roughness.

Returning to the Internal Fluid model, we will now apply a load curve.

2 Results Evaluation and Presentation

2.1 Result Types

The following sections outline the different CFD results types available in Autodesk Simulation 2012.

Reaction Forces

The buttons within the "**Reaction Forces**" pull-down menu in the "**Force and Stress**" panel of the "**Results Contours**" tab will only be available if the "Calculate" option was selected in the "Options for calculating reactions" drop-down box in the "**Output**" tab of the "**Analysis Parameters**" dialog. The commands will display the reaction forces in the fluid model. Non-zero reaction forces will exist at the boundaries where a prescribed velocity has been applied. Either the magnitude; the individual X, Y and Z components; or a vector plot can be displayed.

Velocity

The velocity results can be displayed using the "**Velocity**" pull-down menu in the "**Velocity and Flow**" panel of the "**Results Contours**" tab. The "**Magnitude**" option is the default result type and will automatically be displayed on the model as the analysis proceeds. In addition, the X, Y or Z magnitude of the velocity can be displayed. The "**Vector Plot**" command is often useful for a fluid flow analysis because it allows the user to easily see the direction of the flow

Pressure

The nodal-based smoothed pressure results can be displayed using the "**Pressure**" command menu in the "**Force and Stress**" panel of the "**Results Contours**" tab. The least-squares method of smoothing the pressure results will be used. This method will provide accurate results for internal nodes. For high Reynolds number flows, this method may not provide accurate results at boundaries where prescribed velocities have been applied.

Vorticity

The vorticity is a measure of the rotation of a fluid element as it moves through a flow field. Vorticity results are only available for 2-D elements. The nodal-based smoothed vorticity results can be displayed using the "**Vorticity**" command, in the "**Velocity and Flow**" panel of the "**Results Contours**" tab. The least-squares method of smoothing the vorticity results will be used. This method will provide accurate results for internal nodes. For high Reynolds number flows, this method may not provide accurate results at boundaries where prescribed velocities have been applied.

Vorticity Precision

The precision results based on the vorticity can be displayed using the "**Vorticity Precision**" command, in the "**Velocity and Flow**" panel of the "**Results Contours**" tab. Vorticity precision is only available for 2-D elements. Precision is a method of highlighting stepped changes in results between adjacent elements. In an ideal model, the vorticity would change smoothly between adjacent elements. However, in the process of discretizing the model with elements, there will be a change in the results. The vorticity precision measures the local gradient in vorticity scaled by a global maximum vorticity. This will allow the user to see areas that may require a finer mesh.

Vorticity precision is calculated as:

$P_n = Max(|\omega_e - \omega_n|)/|\omega_{max}|$, e = neighboring elements of node n, if $|\omega_{max}| \neq 0$
$P_n = 0$,   if $|\omega_{max}| = 0$
where:

ω_n = the vorticity obtained by the smoothing process at node n
ω_e = the interpolated vorticity at the center of element e
ω_{max} = a global maximum of ω_n

Flow Rate

The volumetric flow rate results (in units of length3/time) can be displayed using the "**Flow Rate Through Face**" command within the "**Velocity and Flow**" panel of the "Results Contours" tab. This is an element-based result, calculated at the centroids of the elements and projected to the element faces. To inquire on the flow rate for selected element faces, smoothing of the results must be disabled. The smoothing of the results can be enabled and disabled using the "Smooth Results" command within the "Settings" panel of the "Results Contours" tab. Click this command to toggle smoothing on or off.

You may also inquire on the nodal flow rates when result smoothing is enabled, in which case the nodal results are approximated from the original element-based results. However, the greatest accuracy is obtained when using the element face flow rate results.

A positive value represents flow into the element and a negative value represents flow out of the element. Nodal results will be zero at interior nodes, since the sum of flow into and out of the node must be zero. However, nonzero results will be shown for individual element faces within the interior fluid flow path. Both face-based and nodal results will be zero for boundaries at which the velocity is constrained in all three global directions. The exception to this is for nodes along the edge of a surface that has one of the following conditions applied to it—a prescribed inlet/outlet, a nonzero prescribed velocity, or a fan surface. In these cases, the nodal flow rate approximation takes into account the flow through the adjacent elements. Even though the node itself is constrained to zero velocity, there will be a nonzero flow result due to the flow through the applicable portion of the adjacent elements.

Stress

The buttons within the "**Stress**" pull-down menu in the "**Force and Stress**" panel of the "**Results Contours**" tab will only be available if the "**Output stress data**" checkbox is activated in the "**Output**" tab of the "**Analysis Parameters**" dialog. The commands will display the stresses within the fluid due to the viscosity. The "**XX**," "**YY**," and "**ZZ**" commands will display the normal stress in the respective directions. The "**XY**," "**YZ**," and "**ZX**" commands will display the shear stress in the respective directions.

2.2 Presentation Options

There are several different methods for presenting the results to gain better insights into the models performance. The different methods are outlined below.

3-D Visualization of 2-D Elements

Access the "**Tools**" tab and select the "**Application Options**" command. Next, click on the "**Results**" tab. Within this dialog there is a button called "**3-D Element Visualization…**"
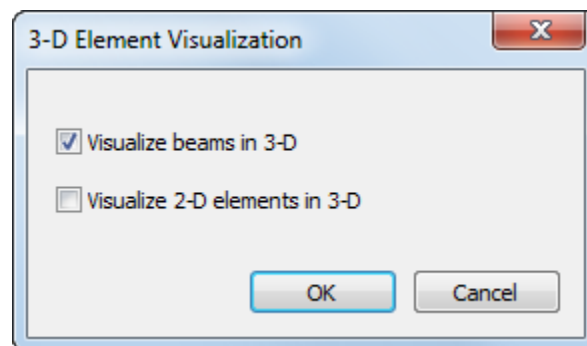
The "**Visualize 2-D elements in 3-D**" option must be activated prior to loading a model into the Results environment for the 3-D rendering capability to be available. Assuming that this has been done, right-click on a part or parts within the model tree and enable the "3-D Visualization" option to view a 2-D model as if it were a solid model. This capability is applicable to both planar and axisymmetric 2-D models and is available only within the Results environment.

Particle Paths

For an unsteady fluid flow analysis, it is possible to track the movement of a massless particle in the fluid flow model using the "**Add Particle Paths**" command located in the "**Flow Visualization**" panel within the "**Results Inquire**" tab. The "**Particle Paths**" dialog will appear. This dialog can also be accessed by selecting nodes in the display area, right-clicking, and selecting the "Add Particle Paths…" command. If the dialog is accessed in this manner, the "**Track Nodal Selection (Interactive Mode)**" checkbox will be activated and the part of the dialog below the checkbox will be grayed out.

Streamlines

For both steady and unsteady fluid flow analyses, it is possible to show the flow through a node during a fluid flow analysis using the "**Add Streamlines**" command located in the "**Flow Visualization**" panel within the "**Results Inquire**" tab. The "**Streamlines**" dialog will appear. This dialog can also be accessed by selecting nodes in the display area, right-clicking and selecting the "**Add Streamlines…**" command. If the dialog is accessed in this manner, the "**Track Nodal Selection (Interactive Mode)**" checkbox will be activated and the part of the dialog below the checkbox will be grayed out.

Add Streamlines

3 Project 2 - 3-D Flow around a Building

3.1 Line Up

Concepts that will be illustrated during this project:

- Boundary layer meshing.
- Applying surface prescribed velocities.
- Applying surface inlet/outlets.
- Displaying particle paths and streamlines.

Objective

Run an unsteady fluid flow analysis to determine the velocity profile as air flows over and around a building at 3 mph (52.8 in/s) and at 30 mph (528 in/s). We expect transitional flow for the lower speed and turbulent flow for the higher speed so we will enable the turbulence option for all calculation steps except for the initial condition (speed = 0 in/s).

Geometry

The file, *Building.step* – contains the air volume shown below. The volume corresponding to the building has already been removed. Create a boundary layer mesh for the model shown below. Use 100% mesh size and deactivate the boundary layer on the inlet, outlet, top and sides.
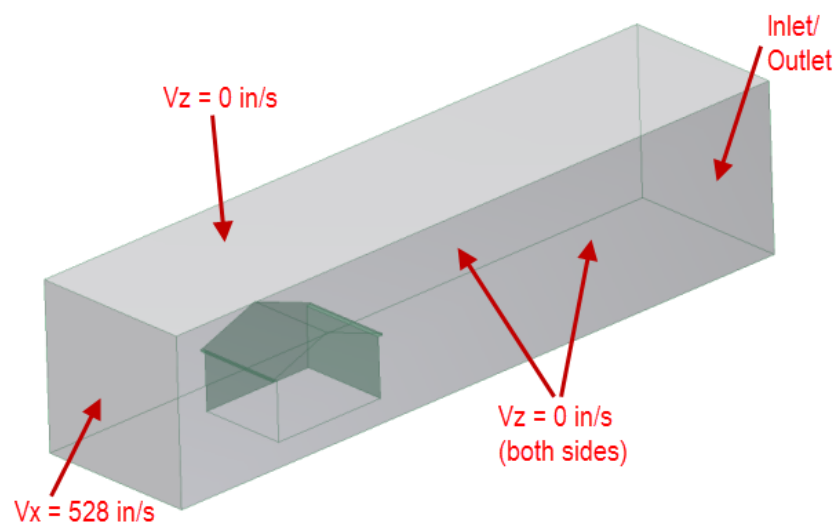
Loading
- Apply a velocity of 528 inches/second in the X direction to the surface on the left end of the model.
- Apply a velocity of 0 inches/second in the Y direction to the front and back surfaces of the model.
- Apply a velocity of 0 inches/second in the Z direction to the top surface of the model.

Constraints
Apply an inlet/outlet to the right end of the model.

Element
3-D

Material
Air

Load Curve

Time	Multiplier	Steps	Turbulence
0	0	1	0
2	0.1	20	1
42	0.1	10	1
50	1	20	1
90	1	10	1

Results
Animate the results to view the changing velocity profile. Observe the appearance of streamlines and particle paths as the results are animated. Compare the maximum velocity magnitude result at the end of the simulation to the table below.

Maximum Velocity Magnitude (in/s) at 90 seconds
~ 944

3.2 Instructions

1. Open *Building.STEP in Autodesk Simulation 2012.*

2. Choose the option to **Use STEP file units** if it is not already selected and click the **OK** button. The original STEP file length unit is inches.
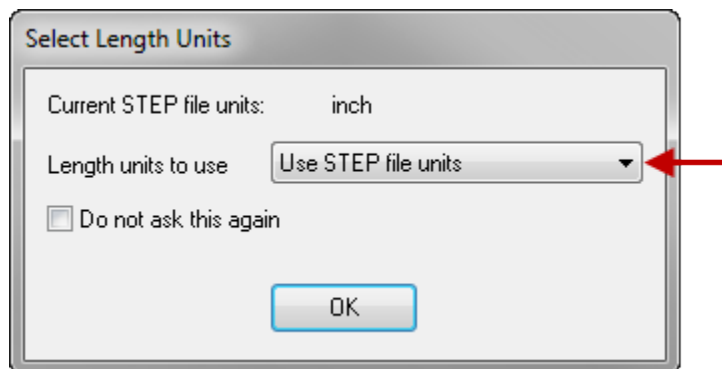
3. A dialog will appear asking you to choose the analysis type for this model. Click the arrow to the right of the analysis type field, select the **Fluid Flow** pull-out menu, and choose the **Unsteady Fluid Flow** type. Press the **OK** button.
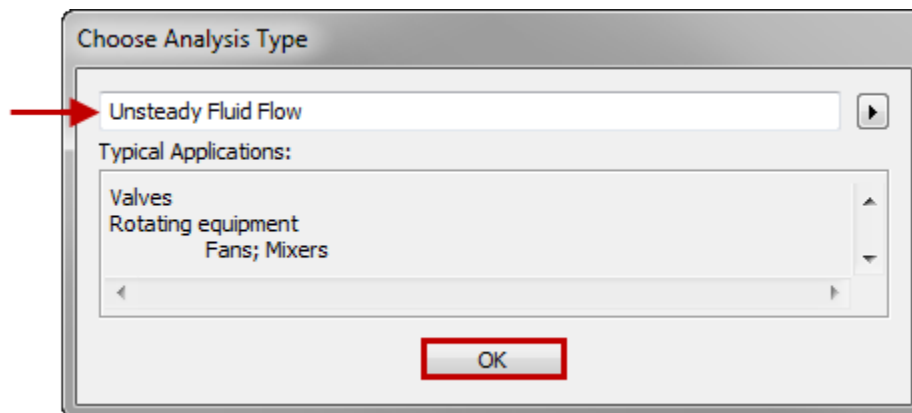
4. To visually verify the orientation of the house-shaped void in the model, we will temporarily turn off the shading by displaying only edges. Click **View tab | Appearance panel | Visual Style pull-down | Edges** to display only the edges of the model.
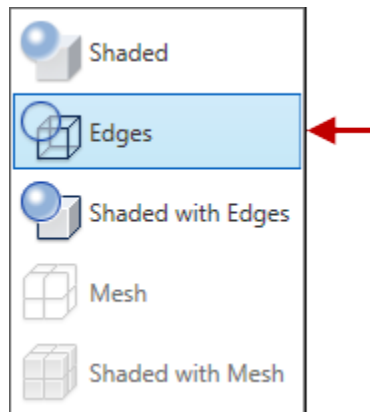
Note the outline of the house within the interior of the model.
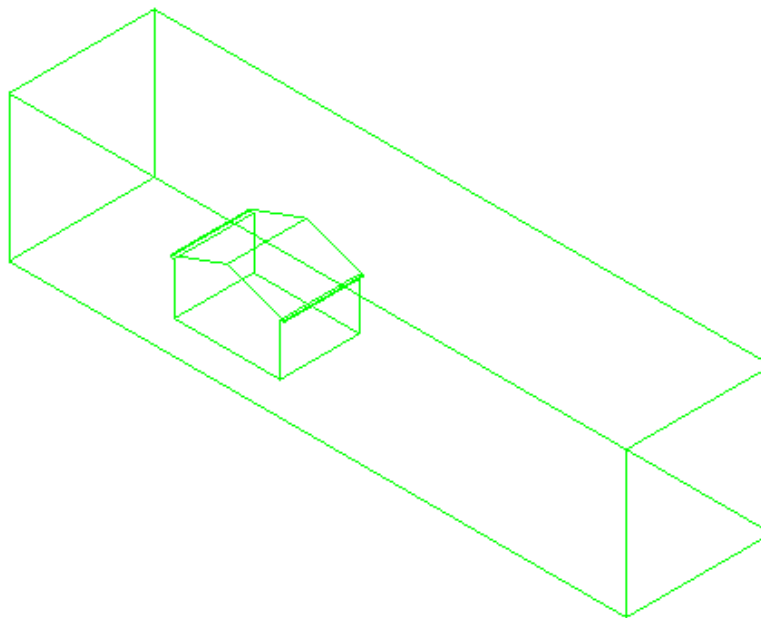
5. Switch the visual style back to **Shaded with Edges.**

6. The mesh settings now need to be changed. Click **Mesh tab | Mesh panel | 3D Mesh Settings** to open the **Model Mesh Settings** dialog. Click the **Options** button.

7. Select the **Solid** icon on the left side and select **Tetrahedra and wedges (boundary layer)** Solid Mesh Type. Click **OK** twice to complete the mesh settings.
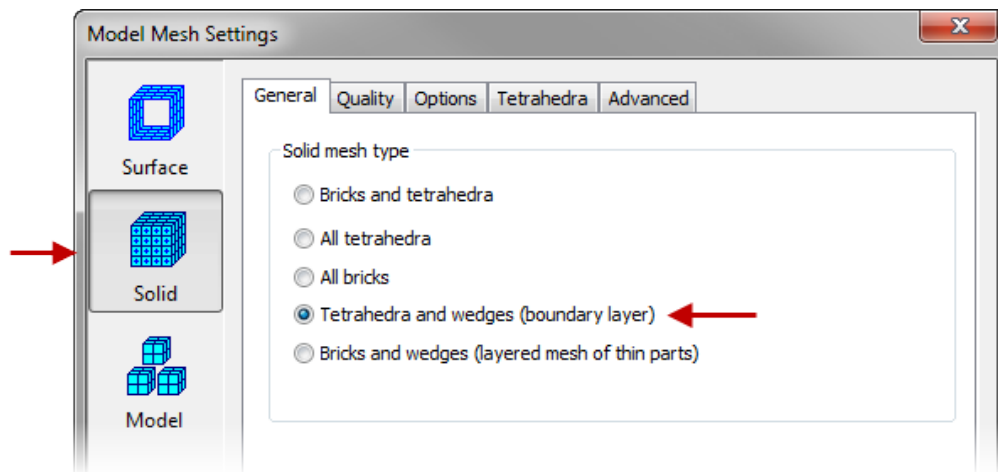
8. Exclude surfaces from the Boundary Layer. Holding down the **<Ctrl>** key, select the **Front**, **Back**, **Left**, **Right** and **Top** faces of the model as shown below. Right-Click in the display area and select **CAD Mesh Options | Exclude from Boundary Layer** option.
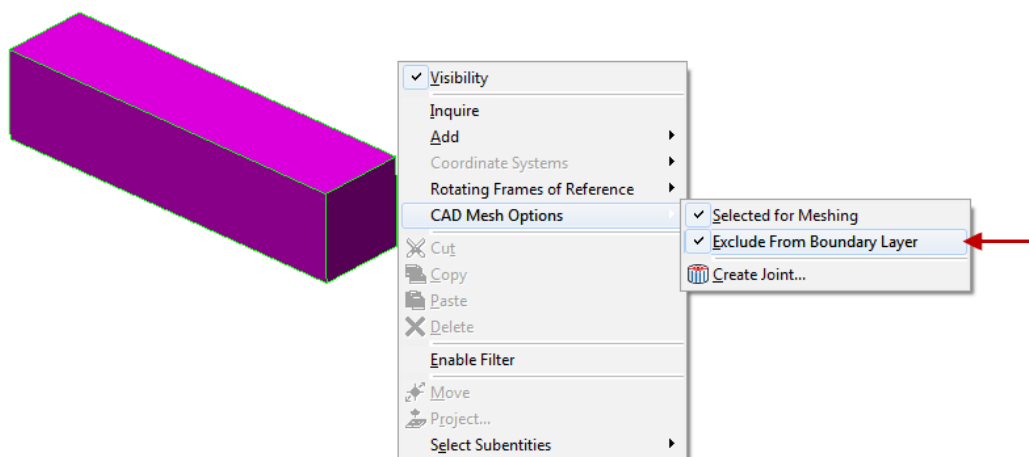
9. Mesh the model by clicking **Mesh tab | Mesh panel | Generate 3D Mesh**. Click **No** when prompted to view the mesh results.

10. Right-Click on the **Material** node of *Part 1* in the **FEA Editor Browser** and select **Edit Material**.
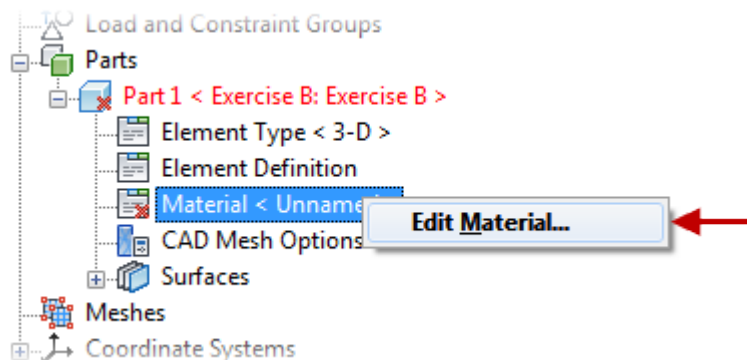
11. Select **Air** under the **Gas** material folder. Click **OK** to complete the material assignment.
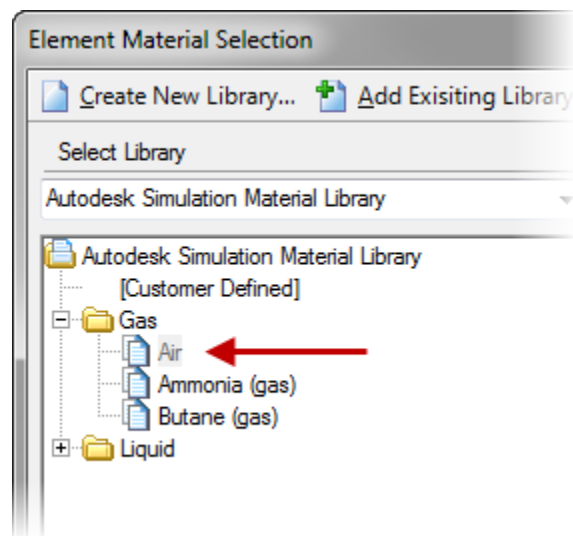
12. Click **Setup tab | Model Setup panel | Parameters** to open the **Analysis Parameters** dialog.

13. Ensure the **Load Curves** tab is active and complete the following items.

- Click the **Add Row** button four (4) times.
- Populate the **Time(s)**, **Multiplier**, **Steps**, and **Turbulence** columns as shown below.
- Click the **OK** button to complete.
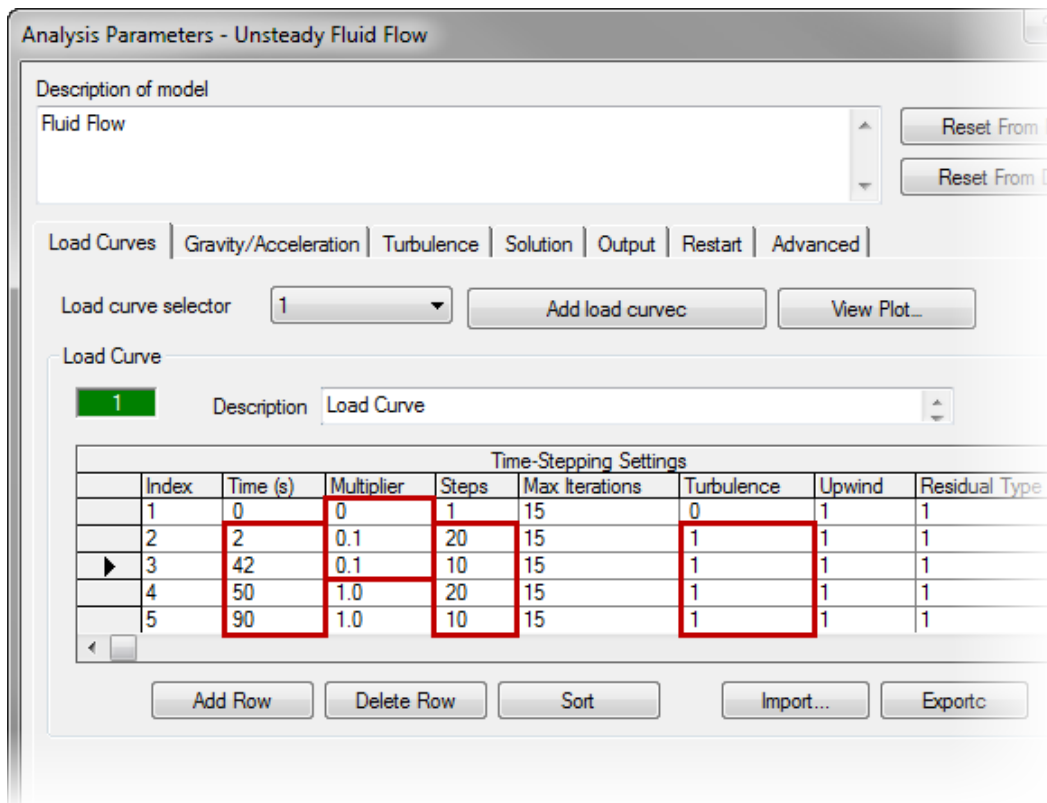
14. Select the end face as shown below.
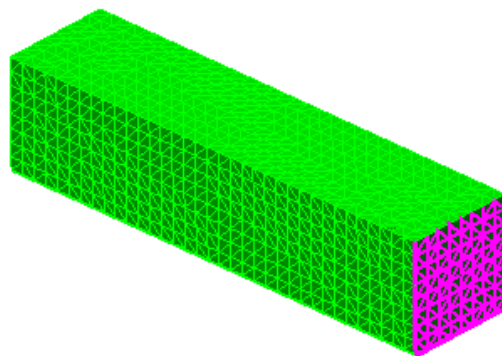
15. Click on the **Setup tab | Fluid Flow Loads panel | Inlet Outlet** to set the selected face as an Inlet/Outlet.
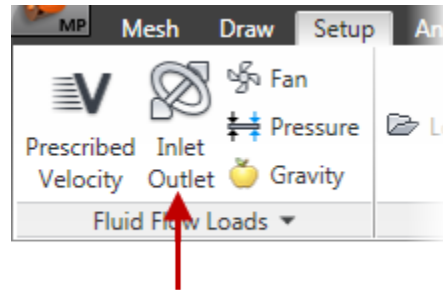
16. Select the left side end face as shown below.
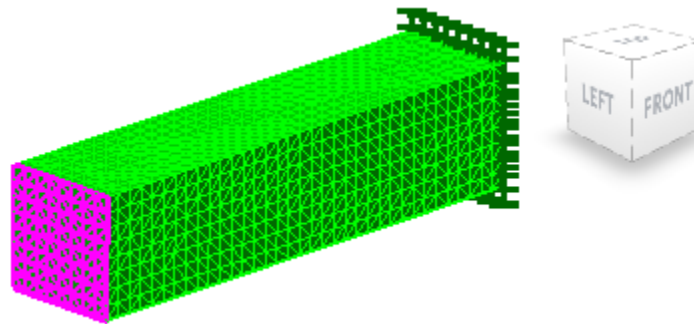
17. Click on the **Setup tab | Fluid Flow Loads panel | Prescribed Velocity** to open the following dialog. Check the **X Magnitude** checkbox and enter 528 as the magnitude. Click **OK** to complete.
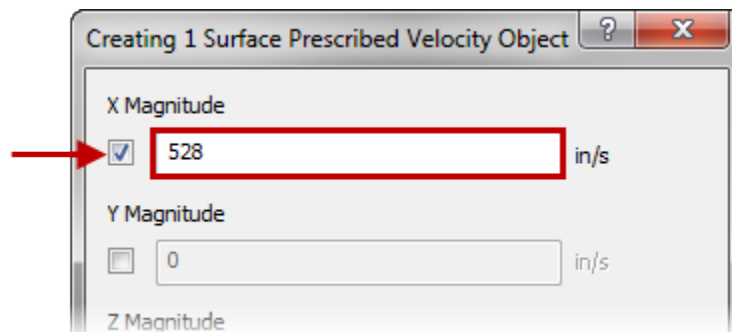
18. Select the front face as shown below.
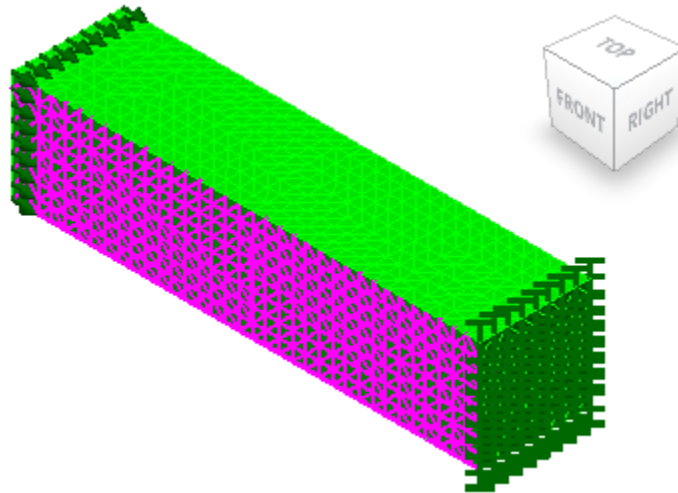
19. Click on the **Setup tab | Fluid Flow Loads panel | Prescribed Velocity** to open the following dialog. Check the **Y Magnitude** checkbox and enter 0 as the magnitude. Click **OK** to complete.
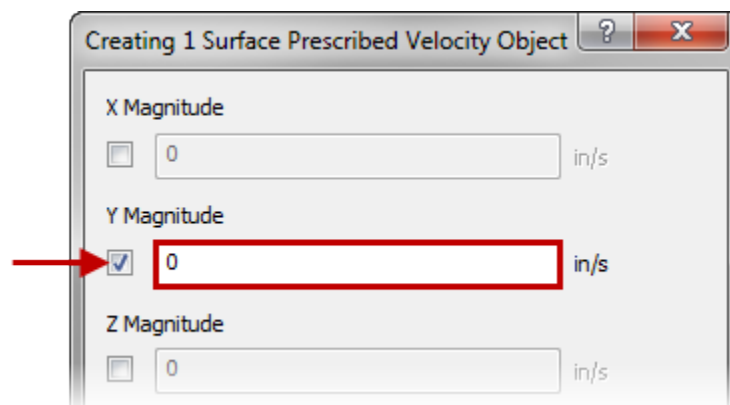
20. Select the top face as shown below.
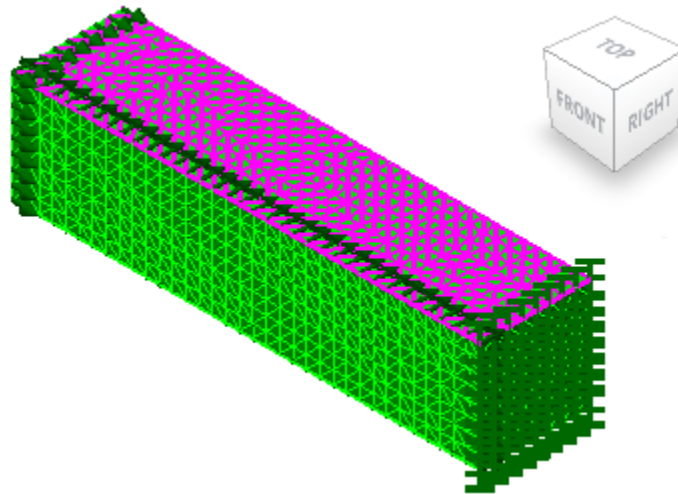
21. Click on the **Setup tab | Fluid Flow Loads panel | Prescribed Velocity** to open the following dialog. Check the **Z Magnitude** checkbox and enter 0 as the magnitude. Click **OK** to complete.
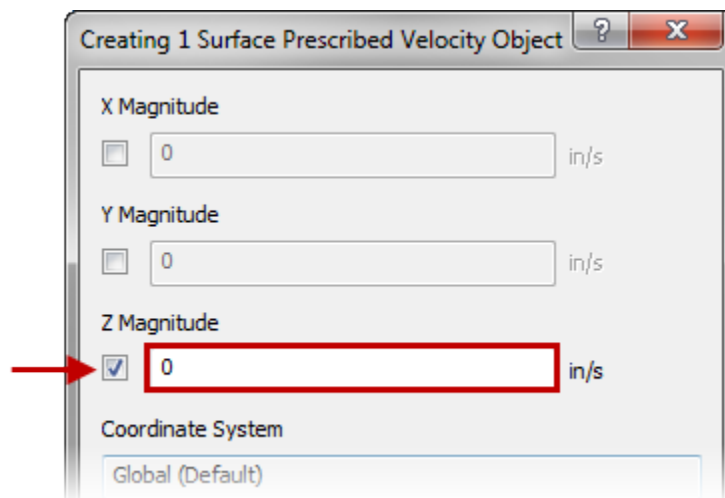
22. Select the back face as shown below.
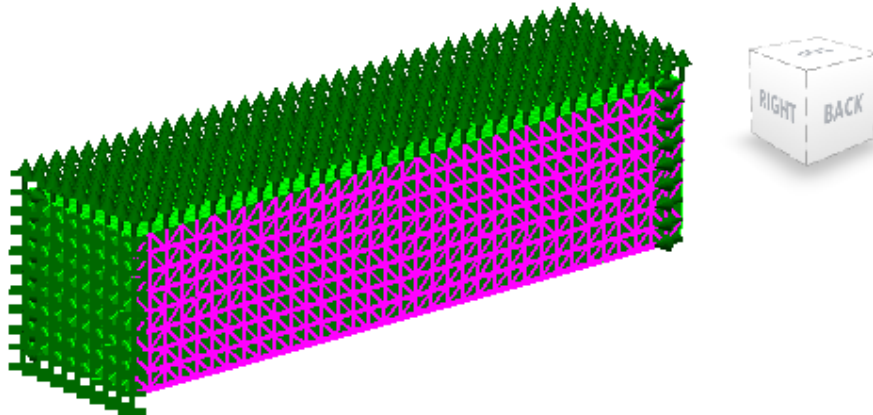
23. Click on the **Setup tab | Fluid Flow Loads panel | Prescribed Velocity** to open the following dialog. Check the **Y Magnitude** checkbox and enter 0 as the magnitude. Click **OK** to complete.
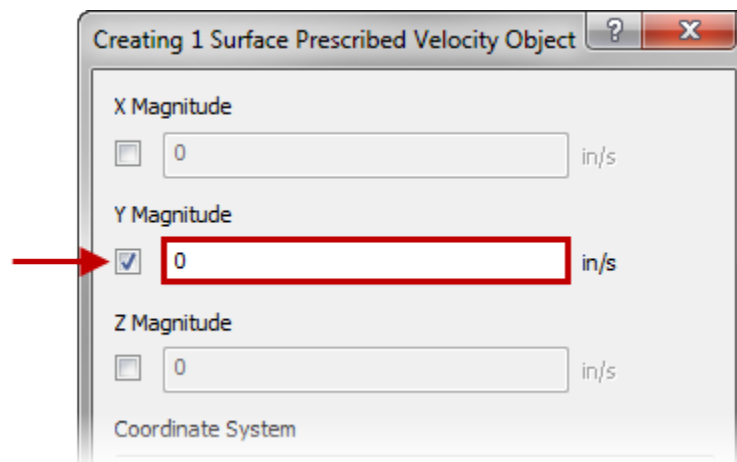
24. The model is ready to be solved. Click **Analysis tab | Analysis panel | Run Simulation**. The analysis will take approximately three (3) minutes to solve depending on computer resources.

25. Improve the viewing of the results click **View tab | Appearance panel | Loads and Constraints** to turn off the display of the loads and constraints on the model.

26. Click on the on the **Results Contours tab | Load Case Options panel | First** button to display the initial condition.
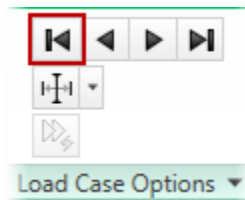
27. Click on the on the **Results Contours tab | Load Case Options panel | Next** button to display Time Step 1 will be displayed (0.1 s)
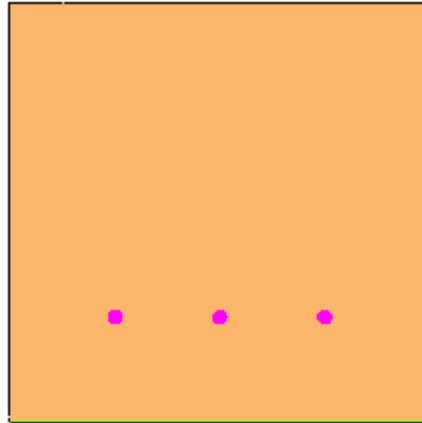

.
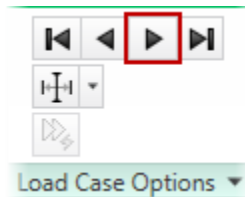
28. Click **Left View** on the **View Cube**

29. Change the selection to **Point** for **Shape** and **Nodes** for **Select**.
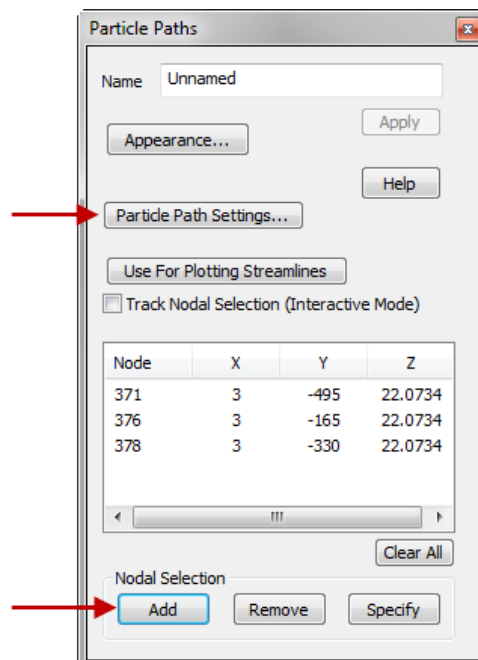
30. Holding the **<Ctrl>** key, select three (3) nodes shown below

31. Click on the **Results Inquire tab | Flow Visualization panel | Add Particle Paths** button to open the **Particle Paths** dialog. Complete the following items.
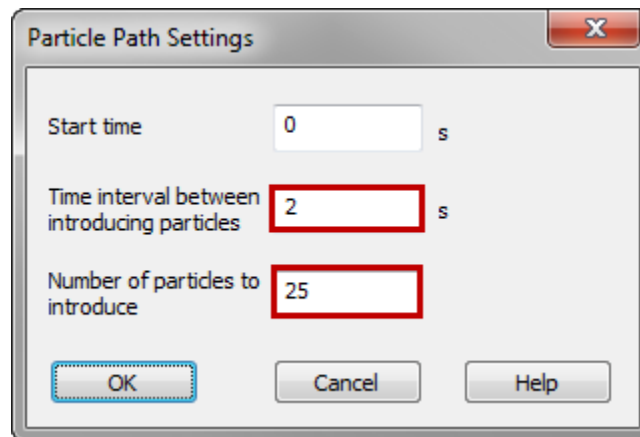
- Select the **Add** command under the **Nodal Selection** field.
- Press the **Particle Path Settings** button.

32. Complete the following items in the Particle Path Settings dialog.

- Type **2** in the **Time interval between introducing particles** field.
- Type **25** in the **Number of particles to introduce** field.
- Press the **OK** button.

33. Press **<Esc>** to dismiss the **Particle Paths** dialog.

    NOTE: Displaying the particle paths has automatically turned on transparency for the model, so that the particles within the interior will be clearly visible.
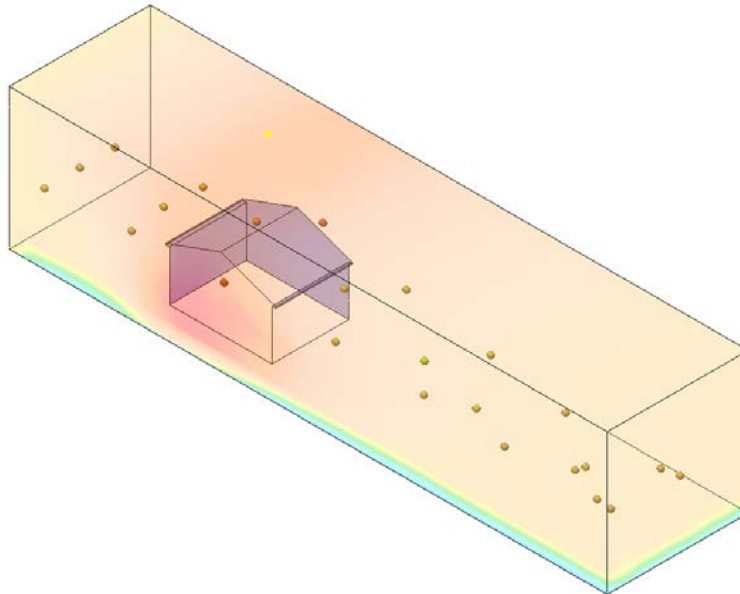
34. Click **Home View** on the **View Cube.**

35. Click on the on the **Results Contours tab | Captures panel | Start** Button to animate the particles path throughout the analysis.
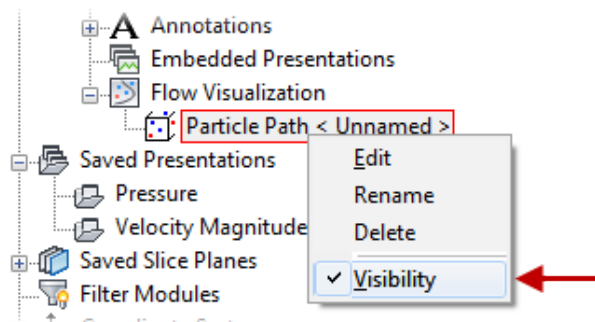
36. The results should look similar to the image below.

NOTE: As the animation proceeds, the particles will flow through the model. The animation will likely be slow at first as all the particle paths are calculated. Keep the animation running and subsequent repetitions should display much more quickly. When you are finished with the particle tracking, use the **Results Contours tab | Captures panel | Stop** command.

37. Turn the visibility of the particle path off complete the following actions:

- Right-click on the **Particle Path** heading listed under the Presentation **1**. Particle Path is a subheading under **Flow Visualization**.
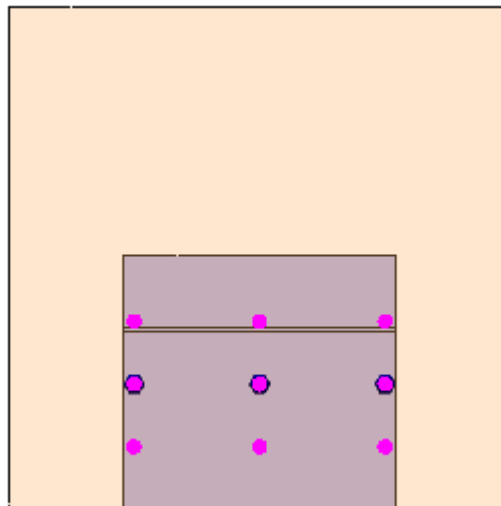- Click on the **Visibility** command to toggle off visibility of the particles.

38. Click **Left View** on the **View Cube.**

39. Ensure the selection is **Point** for **Shape** and **Nodes** for **Select**.

40. Holding the **<Ctrl>** key, select Nine (9) nodes. The selected nodes should look similar to the image below.

41. Click on the **Results Inquire tab | Flow Visualization panel | Add Streamlines** button to open the **Streamlines** dialog.
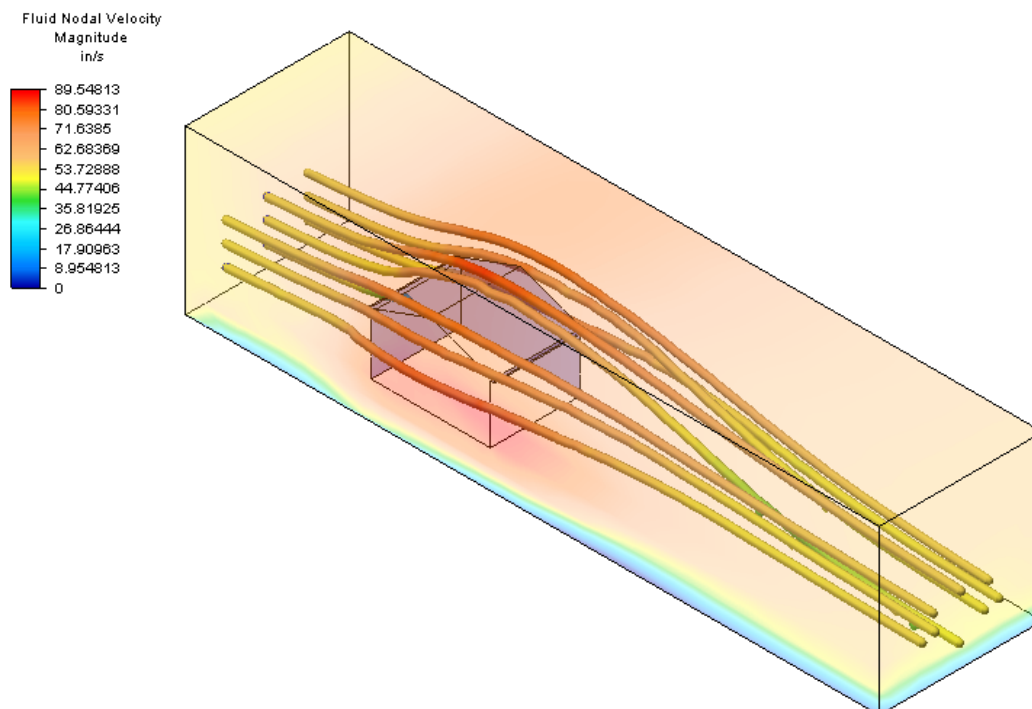
42. Click the **Add** button for the **Nodal Selection**. Press **<Esc>** to dismiss the **Streamlines** dialog

43. Click **Home View** on the **View Cube**.

44. Click on the on the **Results Contours tab | Captures panel | Start** button to animate the particles path throughout the analysis.

NOTE: As the animation proceeds, the streamlines will update as the flow pattern through the model changes. When you are finished viewing the streamlines, use the **Results Contours tab | Captures panel | Stop** command.