Ciespace Engineering Empowered. Collaborative, Open, Agile.

INTERNAL FLOW TUTORIAL

Sample Project 1 Accompanying Tutorial

Abstract

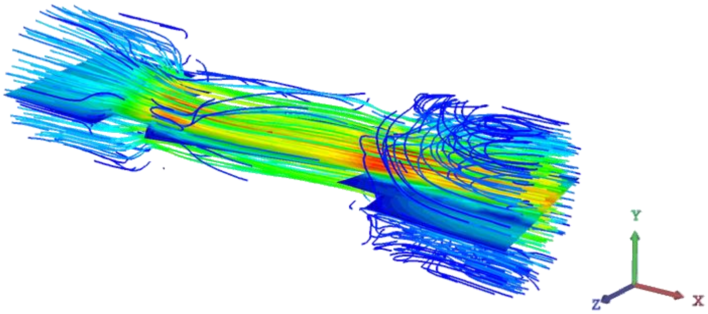
This tutorial will cover the basic workflow of meshing, solving, and post-processing an internal flow use case in Ciespace CFD.

V1.0 9/19/2013

Introduction

This tutorial demonstrates how to set up and run a basic internal flow problem.

You will start by adding a pre formulated sample project to your projects list.



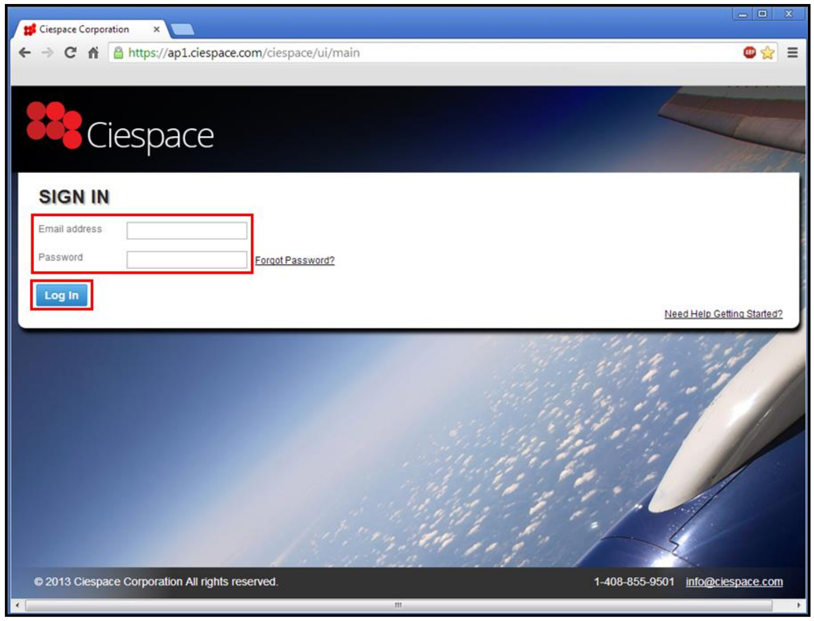
Then you will duplicate the workflow by right clicking on the existing import node and adding the required nodes. You will mesh the geometry and set up a solver for a steady flow problem.

Finally, the results will be post processed.

Steps in the Tutorial appear in Black font. Selections in the UI are outlined in red boxes . Additional information or hints are in Blue.

Step 1: Logging in and adding a sample project

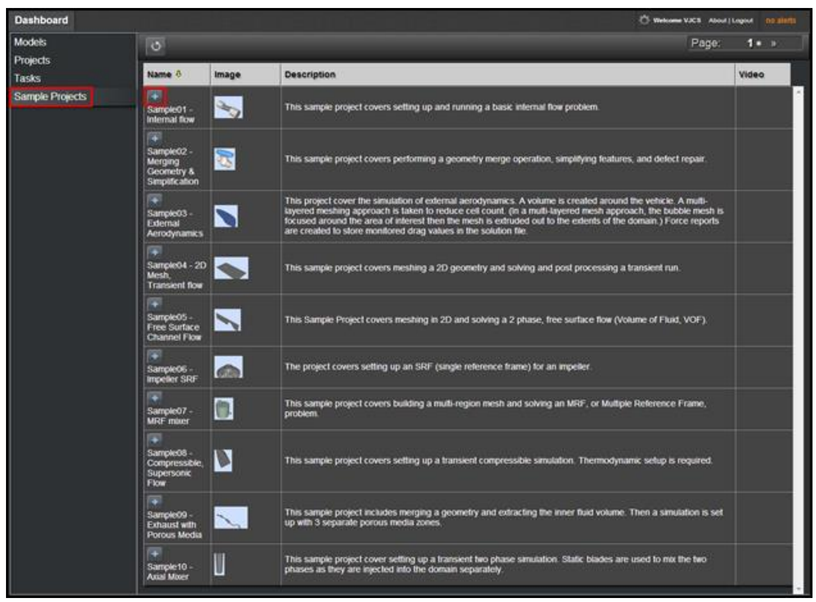
1. Log into Ciespace with the provided "Email address" and "Password" for your assigned server .



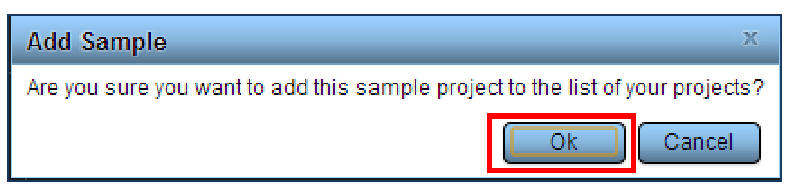
2. In the Dashboard, click on “Sample Projects”.

3. In the Sample Projects tab, click on the blue “+" button next to "Sample01-Internal Flow" project to add it to the project list.

   Tutorials and videos are avaliable for some of the sample projects by selecting the “Tutorial” link in the Tutorials column.

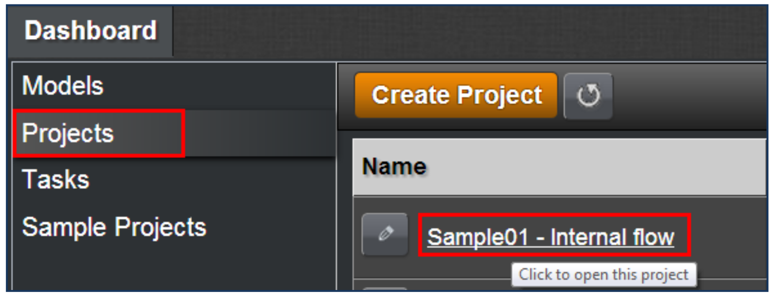


4. Click "Ok" button in the "Add Sample" pop-up window.



5. In the Dashboard, click on “Projects”.

6. In the Projects tab, click on the link of the newly created project to open it.



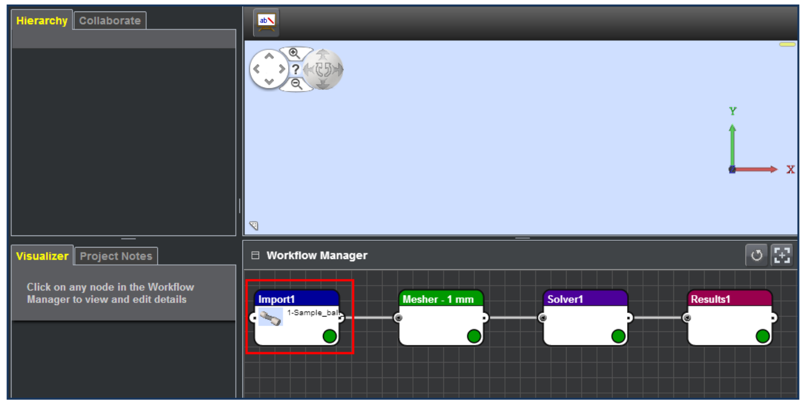
7. Double Click on the "Import1" node in the Work Flow manager to view the surface model.

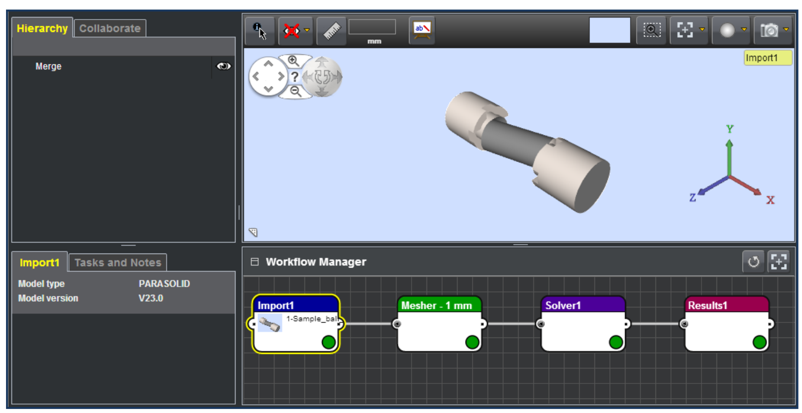
   Double clicking on a node in the workflow will display the output of that node in the graphics window.

   A single click will show the settings of that node in the”vizualizer” on the lower left hand portion of the screen.

   The values in a previously executed node can be changed and re-executed.

   You can also clone a node in an existing workflow – this will copy the workflow steps and settings from where you made the clone. Changes can then be made and the nodes re-executed.



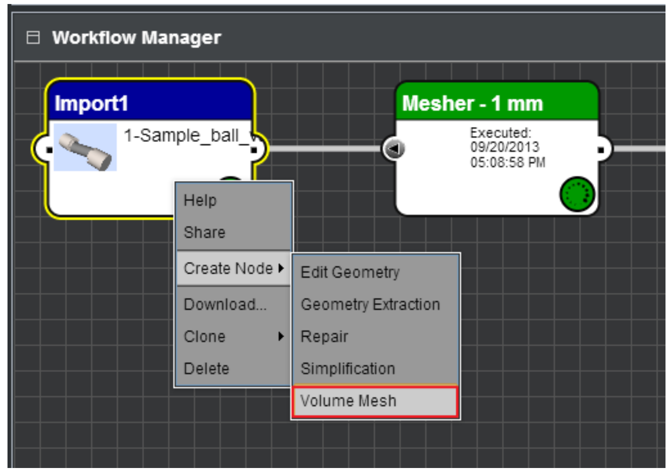


Step 2: Creating the volume mesh

<table>
<tr>
<td>

1. Right click on the “Import1” node an select “Create Node” -> “Volume Mesh”.
2. Click on the newly created volume mesh node to view the options in the vizualizer.

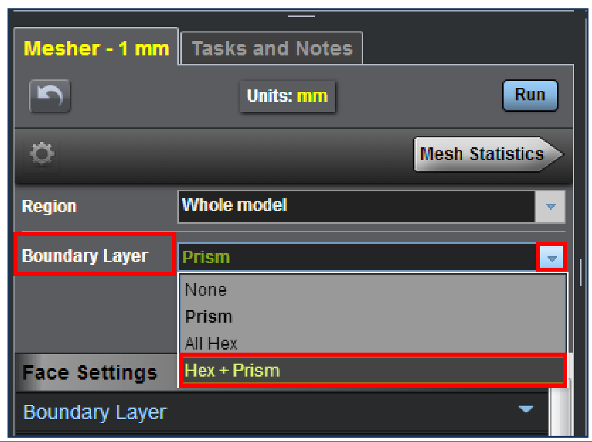
</td>
<td>



</td>
</tr>
<tr>
<td>

3. Under “Boundary Layer” select the type “Hex + Prism” from the drop-down list. This automatically sets "Mesh type" to "Non-conformal" and “Element/Cell Type” to “Hexdom (Hex + Tet)”.

</td>
<td>



</td>
</tr>
<tr>
<td>

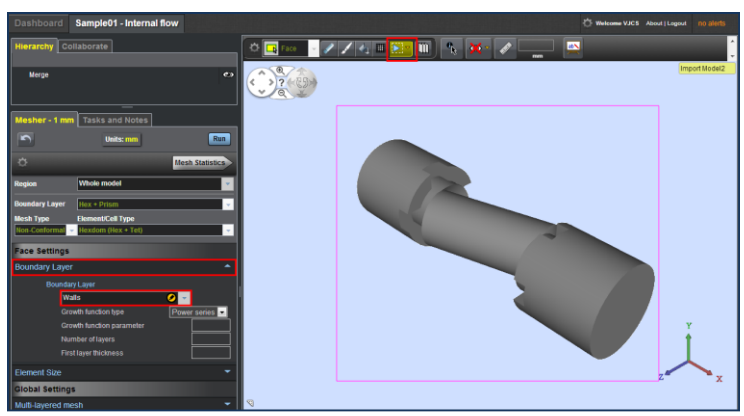
4. Expand the Boundary Layer settings panel.

5. Click on “New Faces” enter the name “Walls” and hit enter. (This will automattically put you in selection mode for this face group).

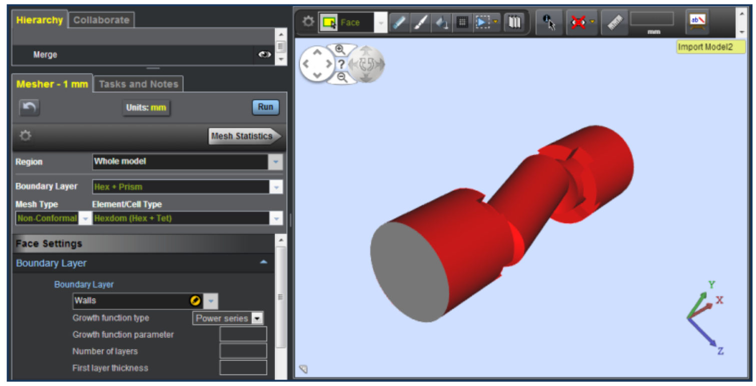
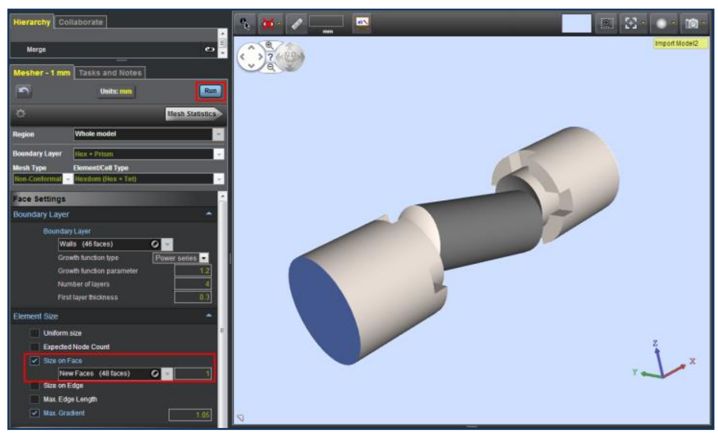
   You can also manually invoke selection by picking the small pencil icon next to the group name. While the pencil is yellow selection is “active”. Picking the pencil again will “close” seleciton.

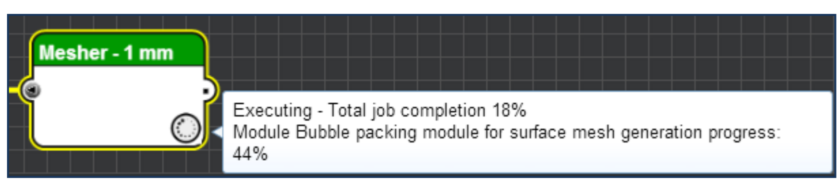
6. Use the Box select tool to select all the faces of the fluid volume.

</td>
<td>



</td>
</tr>
</table>

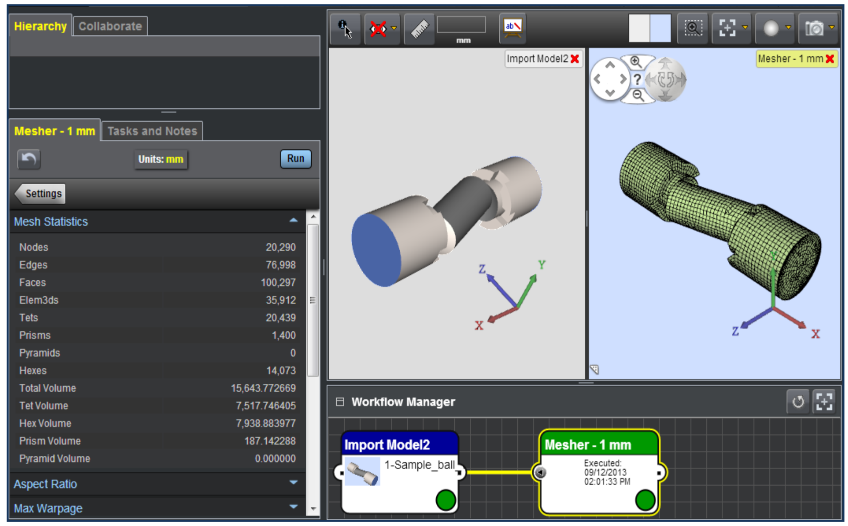
All the selection tools become visible once selection is active in the upper left hand corner of the graphics window. You can also toggle between selection and de-selection by picking the gear icon by the selection toolbar.	
7. Click on the X min and X max faces of the fluid volume to unselect the faces. 8. Click on "Edit group" (yellow pencil) button to confirm the selection. 9. Enter 1.2 for "Growth function parameter". 10. Enter '4 for "Number of layers". 11. Enter 0.3 for "First layer thickness".	
12. Expand the “Element Size” panel. 13. Under “Size on face” click on the "Edit Group" button. 14. Use the zone select tool to select all the faces of the fluid volume and confirm the selection. 15. Enter 1 as the size. 16. Press Run.	

17. Monitor the progress of the volume mesh by hovering the mouse pointer over the spinning icon in the “Mesher - 1mm” node.	

18. Once the mesh has finished it will automatically load in the graphics area.

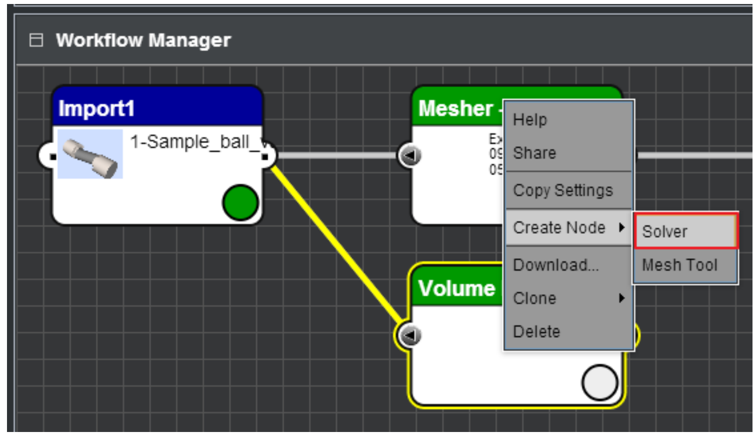
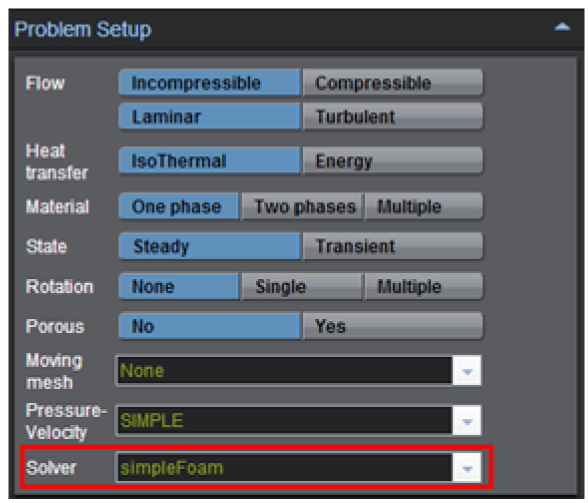
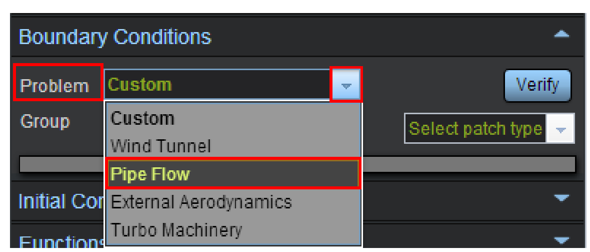
19. Press “Mesh Statistics” to see details on element size and quality metrics.

    You can download the result (output) from any node by selecting the “Download” menu in the right mouse click menu when on the node. You can also export a mesh to different formats by right picking on the mesh node and selecting the export menu.



V1.0 9/19/2013

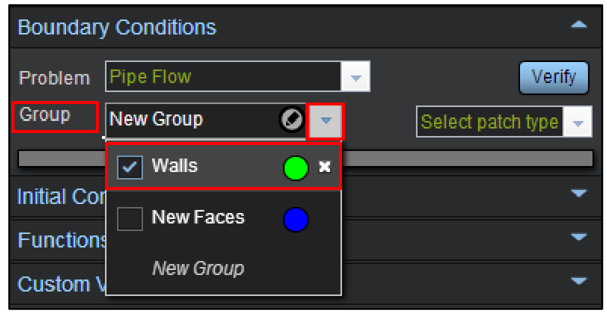
Step 3: Setting up and running the simpleFoam solver

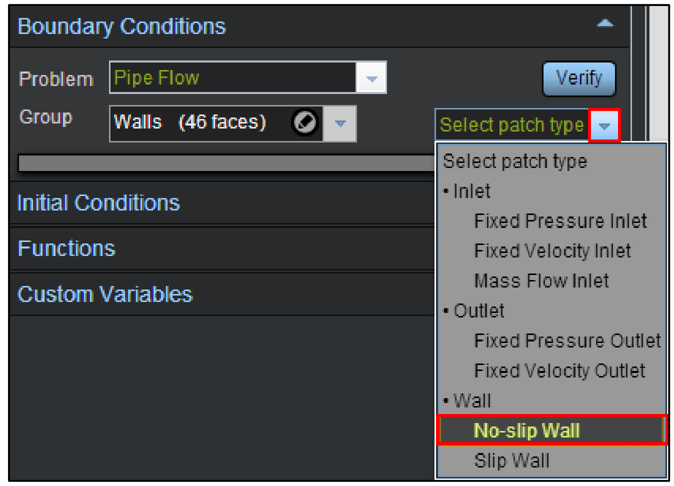
1. Right click on the new volume mesh node and select “Create Node” -> “Solver”.	
2. Expand the “Problem Setup” panel. 3. By default "simpleFoam" solver is selected. Ciespace will select the correct OpenFOAM solver based on the physics specified. If in a problem you don’t see a solver listed then that set of physics is incompatible with OpenFOAM (for example there is no steady state two phase (Standard) solver; but there is a two phase transient solver).	
4. Expand the “Boundary Conditions” panel. 5. Select "Pipe Flow" as the problem type from the dropdown list.	

6. For "Group" click on the dropdown list and click the check box next to “Walls” and click on “Walls”. Set the patch type to “No-slip Wall”.

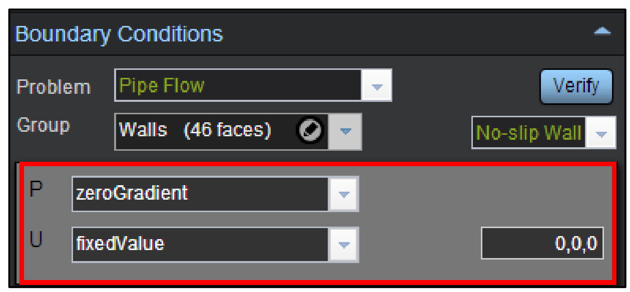
   In Ciespace we have grouped the valid patch types (wall, inlets, etc…) by problem type.

   You can also set problem type to “Custom” where all of the patch types are available and you can manually enter more advanced types such as groovyBC.





7. Leave the default BC settings as it is.



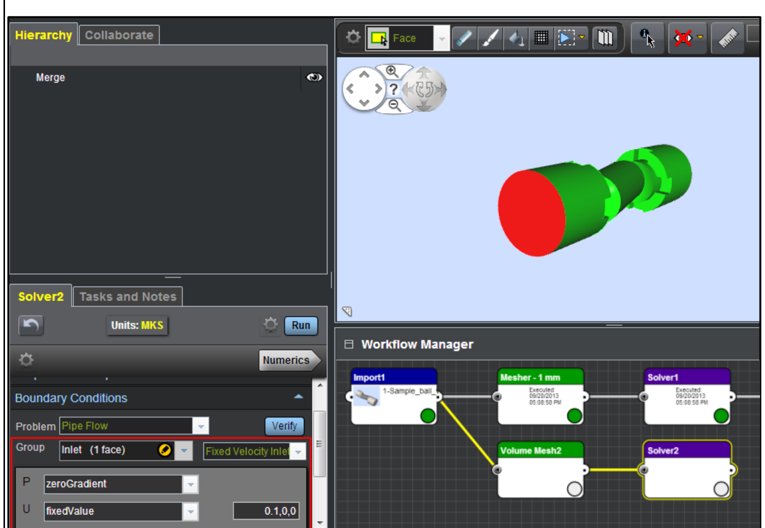
8. Select “New Group”, type Inlet, and press enter.

9. Select inlet face (negative X on the model), and confirm the selection by picking the yellow pencil icon.

10. Set the patch type to "Fixed Velocity Inlet".

11. Set the U value for BC type "fixedValue" to 0.1,0,0.

    Note: Units in the solve node are MKS; so we are entering .1 m/s.

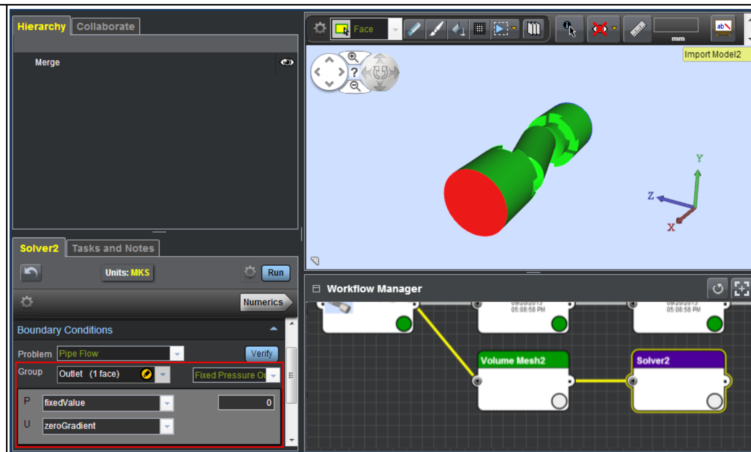


12. Select "New Group", type Outlet, and press enter.

13. Select the outlet face (positive X on the model), and confirm the selection.

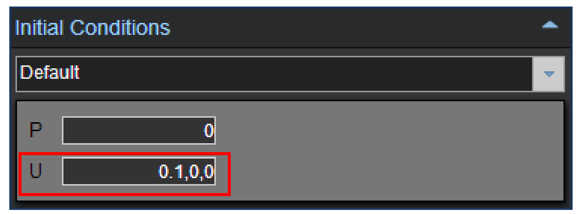
14. Set the patch type to "Fixed Pressure Outlet".

15. Leave the default BC type as it is.



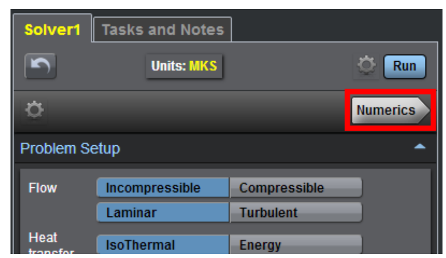
16. Expand the "Initial Conditions" panel.

17. Set the U value to 0.1,0,0.



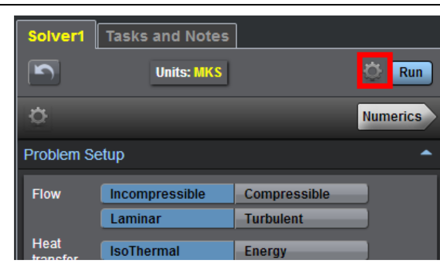
18. Press the "Numerics" arrow to enter the Numerics tab.

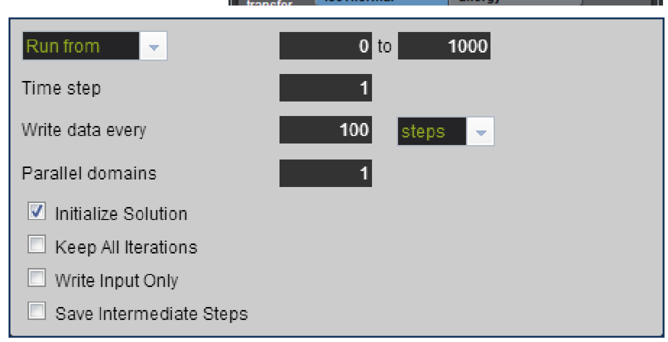
If you want you can tweak solver numeric here – we have chosen best defaults for the problem types for you. We won't make any changes in this tutorial.



19. Click the "gear" icon next to the "Run" button.

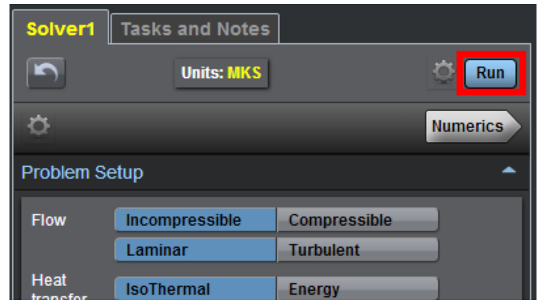
20. Leave the "Run Parameter" default settings.





21. Click “Run”.

    The job is now kicked off; you will see the status icon in the node (the green circle when up to date) spinning and as with the mesh you can hover to get a status.

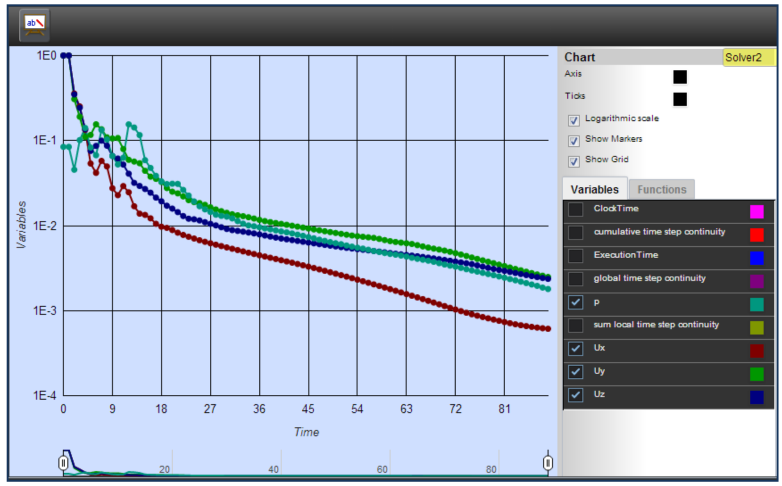


22. When the model is running you can double click on the Solver node to monitor Residuals.

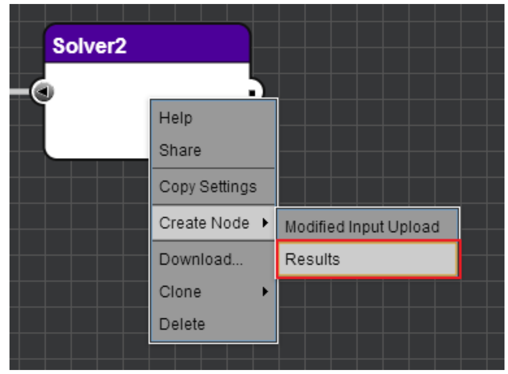
    The residuals plot may take a moment to populate initially and then will update every 10 seconds.

23. To monitor residual of a single variable, unpick other residual box.

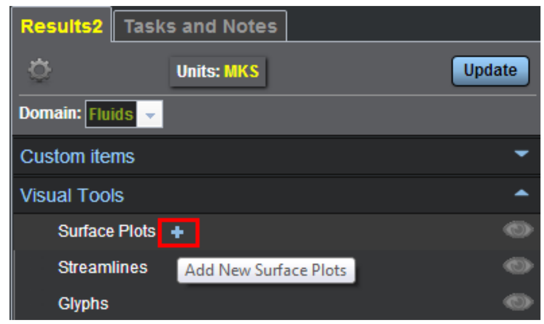
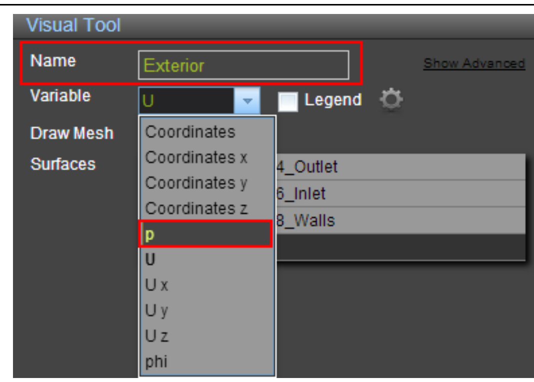
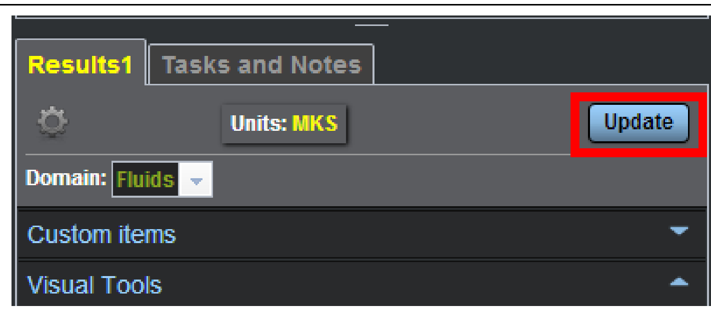
24. Hover your mouse button over a point to see the value.

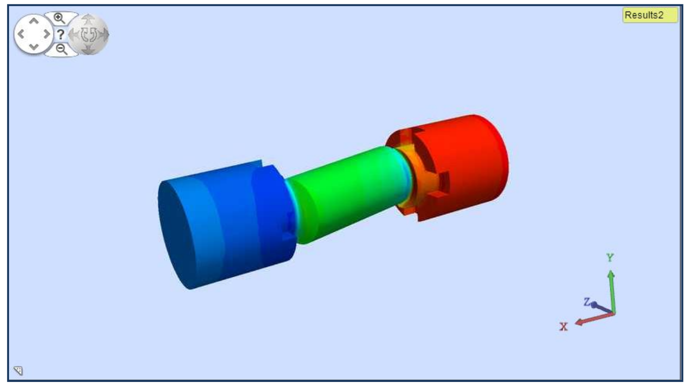
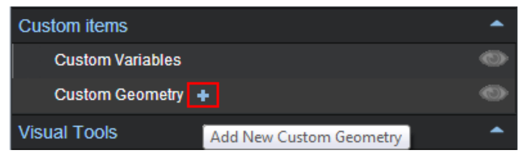
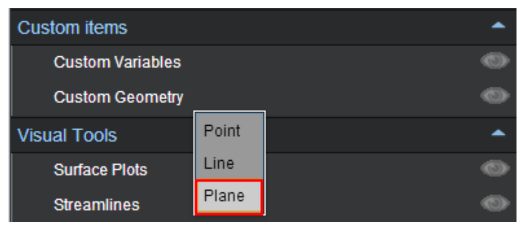
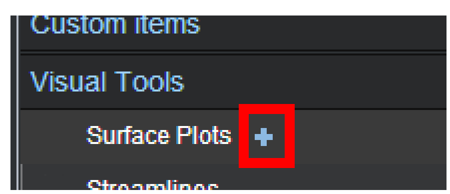


Step 4: Post processing the results

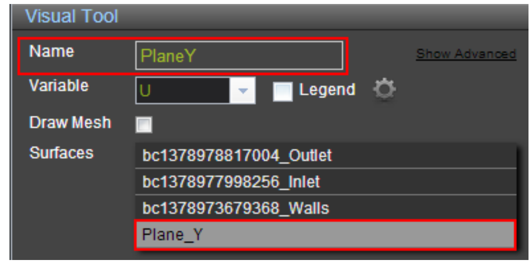
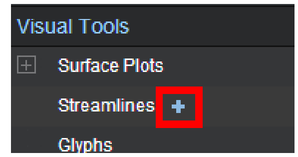
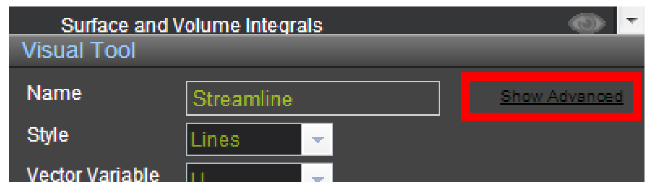
<table>
<tr>
<td>1. After the solver has completed, right click on the “Solver2” node and select “Create Node” -> “Results”.</td>
<td>
</td>
</tr>
<tr>
<td>2. Expand the "Visual Tools" panel.

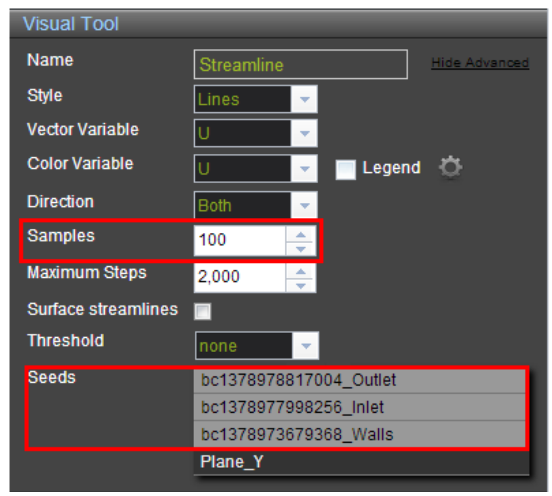
3. Click the “+” sign next to “Surface Plots”.

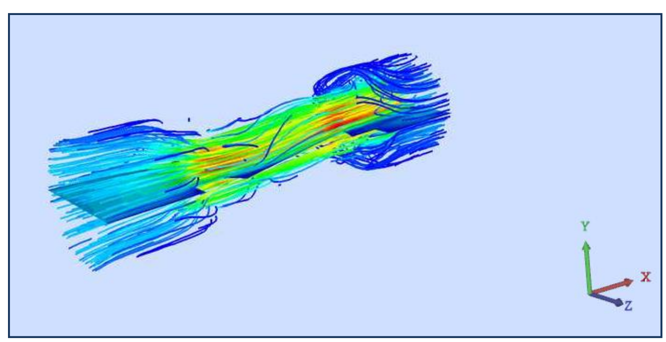
4. Change the name "Surface" to "Exterior".</td>
<td>
</td>
</tr>
<tr>
<td>5. Change the Variable to "p".</td>
<td>
</td>
</tr>
<tr>
<td>6. Click on "Update" button to view exterior surface pressure result.</td>
<td>
</td>
</tr>
</table>

You should now see the pressure plot on the exterior faces. Had we left the quantity at U the plot would be all red (0) because the velocity at the walls is 0 (no-slip wall boundary condition).	
7. Click on the eye of "Exterior" surface to turn it off. (It will grey out, which means it will not be visible the next time you update the Results).	
Now let's create some custom geometry to view results on – for instance a slide through the model. 8. Expand the "Custom items" panel. 9. Click the "+" sign next to "Custom Geometry" and select "Plane".	
10. Change the "plane" name to "Plane_Y" and enter the values as shown in "Custom item". Note: Units in results are in MKS; when entering dimensions for custom geometry creation use m (unlike the mm units used in the mesher node). You can toggle on the display of the custom geometry using the "eyeball" next to the item.	
11. Add another Surface Plot, renaming it to "PlaneY". Pick the "+" sign next to surface plots.	

<table>
<tr><td>12. Select the surface "Plane_Y" from the list.

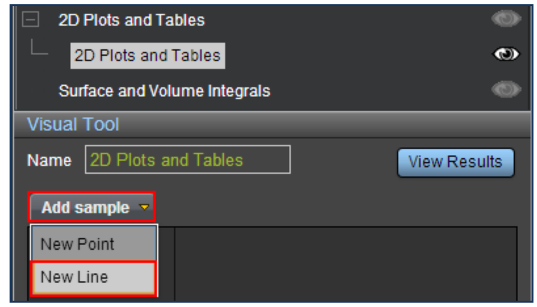
This is where you pick what surfaces will be displayed in your visual tool. Any group used in the solver node will appear here.</td><td>
</td></tr>
<tr><td>13. Click the “+” sign next to “Streamlines”.</td><td>
</td></tr>
<tr><td>14. Click on "Show Advanced".</td><td>
</td></tr>
<tr><td>15. Enter "100" for "Samples".

16. Select the seeds as shown from the list.</td><td>
</td></tr>
<tr><td>17. Click on "Update" button to view velocity and streamline result.

You can toggle the displays on or off using the “eyeball” icons. You can also change the display quality by picking the “Video Settings” icon in the upper left of the graphics window.</td><td>
</td></tr>
</table>

18. Click the “+” sign next to “2D Plots and Tables” in “Custom Geometry”.

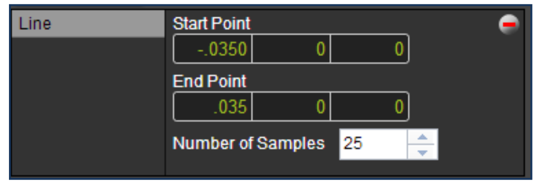
19. Click on "Add sample" and Select the "New Line".



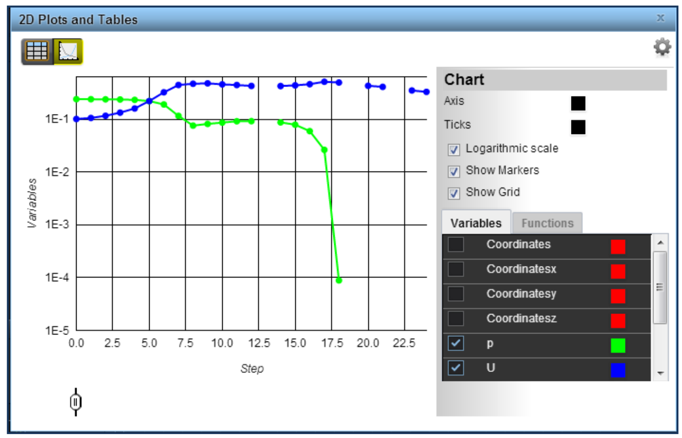
20. Enter the value as shown to the right.

21. Change the "Number of Sample" to 25.

22. Click on "View Results" button.

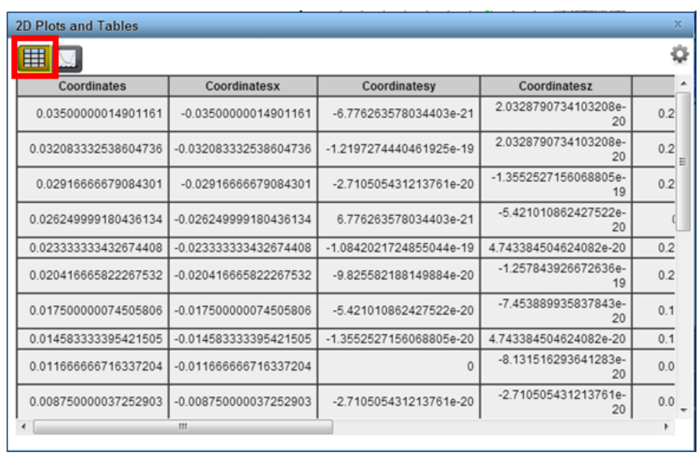


23. Click on "gear" icon to hide the Chart Options".



24. Click on “View Grid" button.

If you want you can copy/paste the table values into another spreadsheet tool like Excel.



Congratulations,

You have successfully completed the tutorial!