

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.

Introduction	2
Step 1: Logging in and adding a sample project	3
Step 2: Creating the volume mesh	5
Step 3: Setting up and running the simpleFoam solver	8
Step 4: Post processing the results	12

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 .	Image: comparison Image: comparison </th
2.	In the Dashboard, click on "Sample Projects".	Dashboard C Beauer V/CS Aber/ Loyer Of Bills Models Poge: 1 + > Pogeds
3.	In the Sample Projects tab, click on the blue "+" button next to "SampleO1- 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.	Tasks Interaction Interaction <thinteraction< th=""> <</thinteraction<>
4.	Click "Ok" button in the "Add Sample" pop-up window.	Add Sample × Are you sure you want to add this sample project to the list of your projects? Ok Cancel

5.	In the Dashboard, click on "Projects". In the Projects tab, click on the link of the newly created project to open it.	Dashboard Models Create Project Projects Image: Create Project Tasks Name Sample Projects Sample01 - Internal flow Click to open this project Click to open this project
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 	Hierarchy Collaborate Image: Collaborate Image: Collaborate Visualizer Project Notes Click on any node in the Workflow Image: Click on any node in the Workflow Image: to view and odit details Image: Click on any node in the Workflow
	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.	Hierarchy Collaborate Hurge Import1 Tasks and Notes Model yees PARASOLD Model yees V23.0

Step 2: Creating the volume mesh



	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.	Itterarchy Collaborate
8.	Click on "Edit group" (yellow pencil) button to confirm the selection.	Mesher - 1 mm Tasks and Notes Units num Units num Empire Fregion Whole model
9.	Enter 1.2 for "Growth function parameter".	Boundary Layer (Inc. + Pham) * Keth Type Dior.Canforms * mondom (Her. + Tet) Face Settings Face Settings
10.	Enter '4 for "Number of layers".	Bondard Laver Wals Growth function type Growth function ppameter
11.	Enter 0.3 for "First layer thickness".	Number of layers Z Final layer fluctmess Z
12.	Expand the "Element Size" panel.	Meranty Consorrate
13.	Under "Size on face" click on the "Edit Group" button.	Machar - 1 mm Taxics and Theorem
14.	Use the zone select tool to select all the faces of the fluid volume and confirm the selection.	Image: Control of the State Sta
15.	Enter 1 as the size.	Undern stor Expedite Note Research (Micros)
16.	Press Run.	

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

Mesher - 1 mm Executing - Total job completion 18% Module Bubble packing module for surface mesh generation progress: 44%

- 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.



Step 3: Setting up and running the simpleFoam solver

 Right click on the new volume mesh node and select "Create Node" -> "Solver". 	Uverkflow Mana Import1	ager Die_ball		Mesh Volun	er Help E Share Copy Settin Create No Download Clone Delete	ngs de • Solver Mesh Tool
2. Expand the "Problem Setup" panel.	F	Problem Se	etup			^
3. By default "simpleFoam" solver is selected.		Flow	Incompress Laminar	ible	Compressibl Turbulent	•
Ciespace will select the correct OpenFOAM		Heat transfer	IsoThermal		Energy	
solver based on the physics specified. If in a		Material	One phase	Two pl	hases Multip	ole
problem you don't see a solver listed then that		State	Steady		Transient	
set of physics is incompatible with OpenFOAM		Rotation	None	Single	Multip	ble
(for example there is no steady state two phase		Porous	No		Yes	
(Standard) solver; but there is a two phase		Moving mesh	None			*
transient solver).		Pressure- Velocity	SIMPLE			×.
		Solver	simpleFoam			-
A Expand the "Doundary Conditions" papel						
4. Expand the Boundary Conditions panel.		Boundary	Conditions			^
5. Select "Pipe Flow" as the problem type from the		Problem	Custom		_	Verify
dropdown list.		Group	Custom Wind Tunnel		S	elect patch type 🤟
			Pipe Flow			
		nitial Cor	External Aerody Turbo Machine	namics		-
		Function	ranso maciline	.,		

 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" 	Boundary Conditions
where all of the patch types are available and you can manually enter more advanced types such as groovyBC.	Boundary Conditions Problem Pipe Flow Group Walls (46 faces) Select patch type Initial Conditions Functions Custom Variables Outlet Fixed Pressure Inlet Fixed Pressure Outlet Fixed Pressure Outlet Fixed Velocity Outlet • Wall No-slip Wall Slip Wall
7. Leave the default BC settings as it is.	Boundary Conditions Problem Pipe Flow Group Walls (46 faces) Verify No-slip Wall P zeroGradient U fixedValue 0,0,0
8. Select "New Group", type Inlet, and press enter.	Hierarchy Collaborate
 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.	Solver2 Tasks and Notes Tasks and Notes Workflow Manager Workflow Manager Solver1 Solver
Note: Units in the solve node are MKS; so we are entering .1 m/s.	Problem Proc Flow Group Intel (1 face) Find Velocity Intel Find Ve

12. Select "New Group", type Outlet, and press	Herarchy Collaborate
enter.	
13. Select the outlet face (positive X on the model), and confirm the selection.	
14. Set the patch type to "Fixed Pressure Outlet".	Solver2 Tasks and Notes Unite: MKS Run O Numerics Boundary Conditions Numerics
15. Leave the default BC type as it is.	Protein interior inte
16. Expand the "Initial Conditions" panel.	Initial Conditions
17. Set the U value to 0.1,0,0.	Default
18. Press the "Numerics" arrow to enter the	Solvert Tasks and Notas
Numerics tab.	Units: MKS CRun
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.	Incompressible Compressible Flow Incompressible Compressible Compressible Laminar Turbulent Heat IsoThermal Energy
19. Click the "gear" icon next to the "Run" button.	Solver1 Tasks and Notes
20. Leave the "Run Parameter" default settings.	Problem Setup Flow Incompressible Compressible Laminar Turbulent Heat IsoThermal Energy
	Run from 0 to 1000 Time step 1
	Write data every 100 steps -
	✓ Initialize Solution
	Keep All Iterations Write Input Only
	Save Intermediate Steps

V1.0 9/19/2013

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.	Solver1 Tasks and Notes Solver1 Tasks and Notes Units: MKS Run Numerics Problem Setup Flow Incompressible Compressible Laminar Turbulent Heat IsoThermal Energy
 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. 	1E0 Chart Solver2 1E-1 Chart Solver2 1E-1 Clogarithmic scale Clogarithmic scale Show Markers Show Markers Show Ord Variables Functions Clogarithmic scale Show Ord Variables Functions Clogarithmic scale Stable Functions Clogarithmic scale Solver2 Stable Functions Clogarithmic scale Solver2 Stable Functions Clogarithmic scale Solver2 Solver3 Solver3
23. To monitor residual of a single variable, unpick other residual box.24. Hover your mouse button over a point to see the value.	1E-3 1E-4 0 9 18 27 36 45 54 63 72 81 <i>Time</i> 0 20 40 60 80 0

Step 4: Post processing the results

1.	After the solver has completed, right click on the "Solver2" node and select "Create Node" -> "Results".	Solver2 Help Share Copy Settings Create Node Modified Input Upload Download Clone Delete
2.	Expand the "Visual Tools" panel.	Results2 Tasks and Notes
3.	Click the "+" sign next to "Surface Plots".	Units: MKS Update Domain: Fluids
4.	Change the name "Surface" to "Exterior".	Custom items Visual Tools
		Surface Plots + O
5.	Change the Variable to "p".	Visual Tool Name Exterior Show Advanced Variable U Coordinates Coordinates x Coordinates x Coordinates z P U U X U y U x U y U z phi
6.	Click on "Update" button to view exterior surface pressure result.	Results1 Tasks and Notes Units: MKS Update Domain: Fluids Custom items Visual Tools

	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).	Results2
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).	Visual Tools
	Now let's create some custom geometry to view results on – for instance a slide through the model.	Custom items Custom Variables Custom Geometry + Visual Tools Add New Custom Geometry
8. 9.	Expand the "Custom items" panel. Click the "+" sign next to "Custom Geometry" and select "Plane".	Custom items Custom Variables Custom Geometry Visual Tools Point Surface Plots Streamlines 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. 	Custom Item Name Plane_Y Point 0 0 Normal
11	Add another Surface Plot, renaming it to "PlaneY". Pick the "+" sign next to surface plots.	Custom items Visual Tools Surface Plots





V1.0 9/19/2013

Congratulations,

You have successfully completed the tutorial!

V1.0 9/19/2013