FLUENT - Turbulent Flow Past a Sphere - Step 3

UNDER CONSTRUCTION

Author: Daniel Kantor and Andrew Einstein, Cornell University

Problem Specification
1. Create Geometry in GAMBIT
2. Mesh Geometry in GAMBIT
3. Specify Boundary Types in GAMBIT
4. Set Up Problem in FLUENT
5. Solve!
6. Analyze Results
7. Refine Mesh

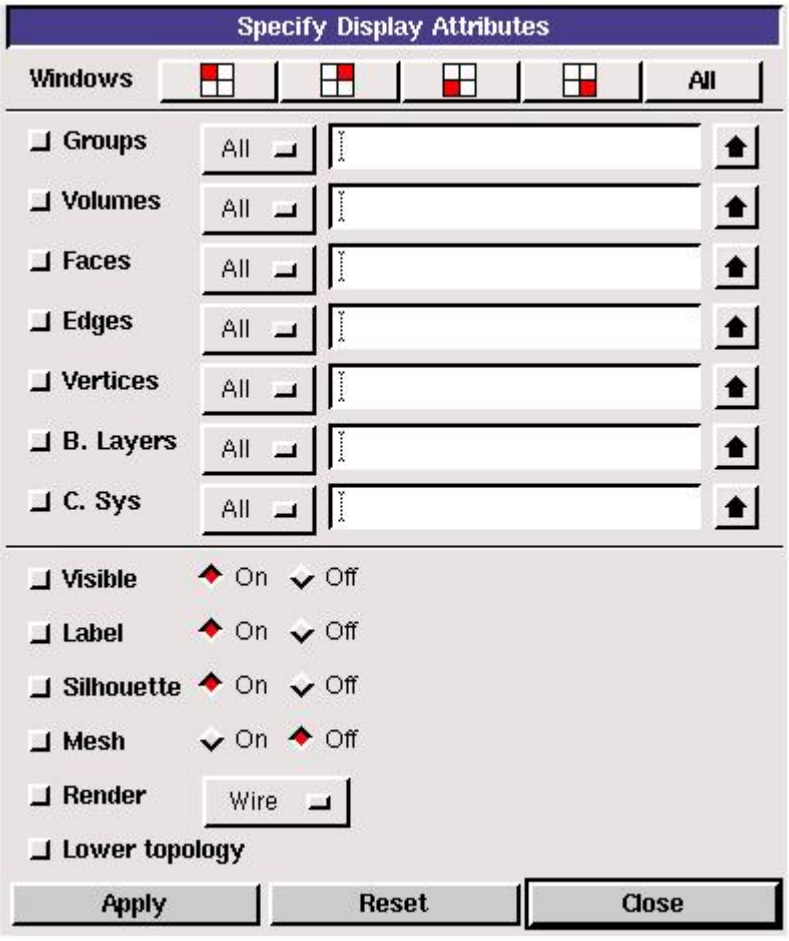
Problem 1

Step 3. Specify Boundary Types in Gambit

Now that the model has been meshed, we need to group the relevant faces to define boundary types. To make it easier to locate the important faces, click on the Specify Model Display Attributes button at the bottom of the screen.

Global Control Toolpad > Specify Model Display Attributes 

Check the box next to "Mesh" and select "Off" then click "Apply." This will hide the mesh from view, allowing you to see the wireframe model again.



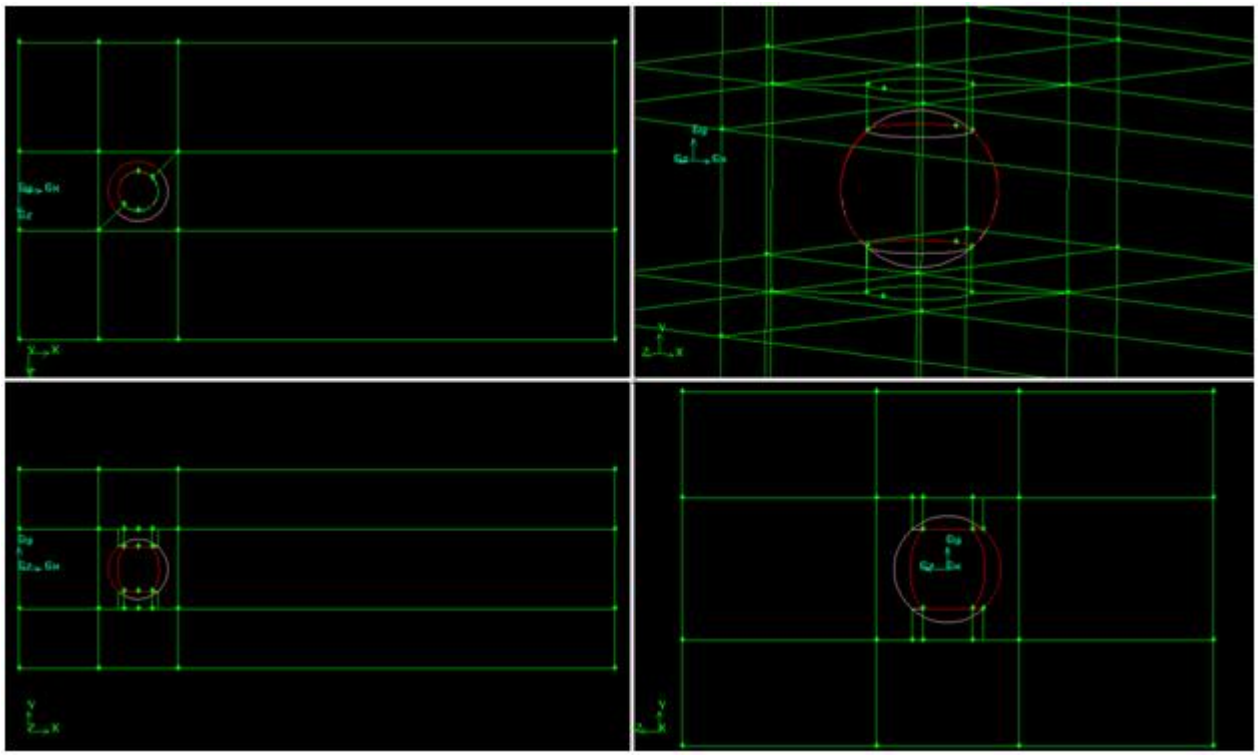
Open the Zones command window and go to Specify Boundary Types.

Operation Toolpad > Zones Command Button  **> Specify Boundary Types** 

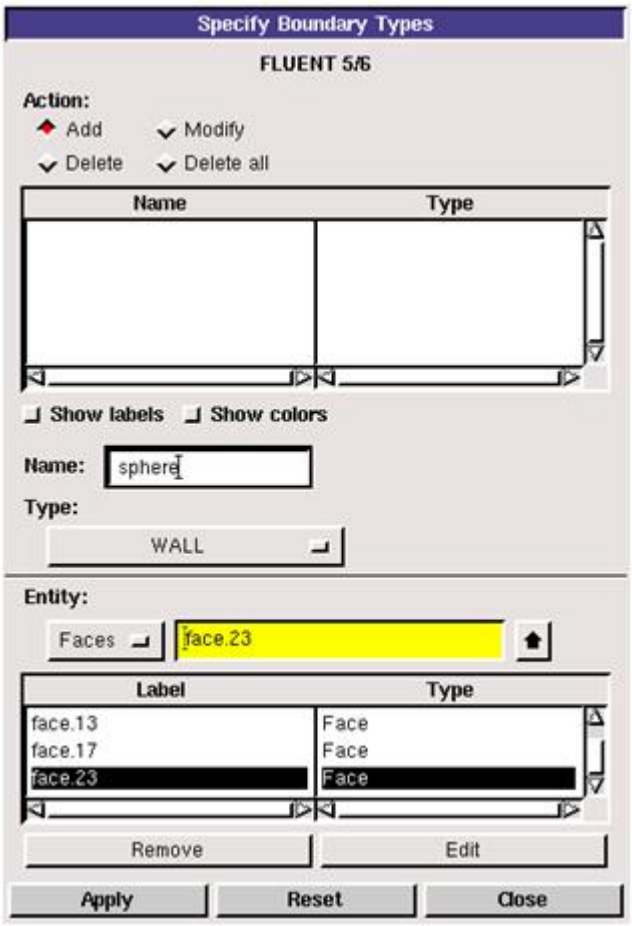
Selecting Wall Types

The only solid surface in the entire model is the sphere, so we'll want to define a single Wall type boundary for the sphere. Hold down the right mouse button to zoom in on the sphere within your model. Hold down 'Shift' and left-click with the mouse to select the four faces that compose the sphere's surface. Selected faces will be highlighted red. It is recommended that you switch to the four-perspective view to make sure you're selecting the right faces.

Global Control Toolpad > Select Preset Configuration 

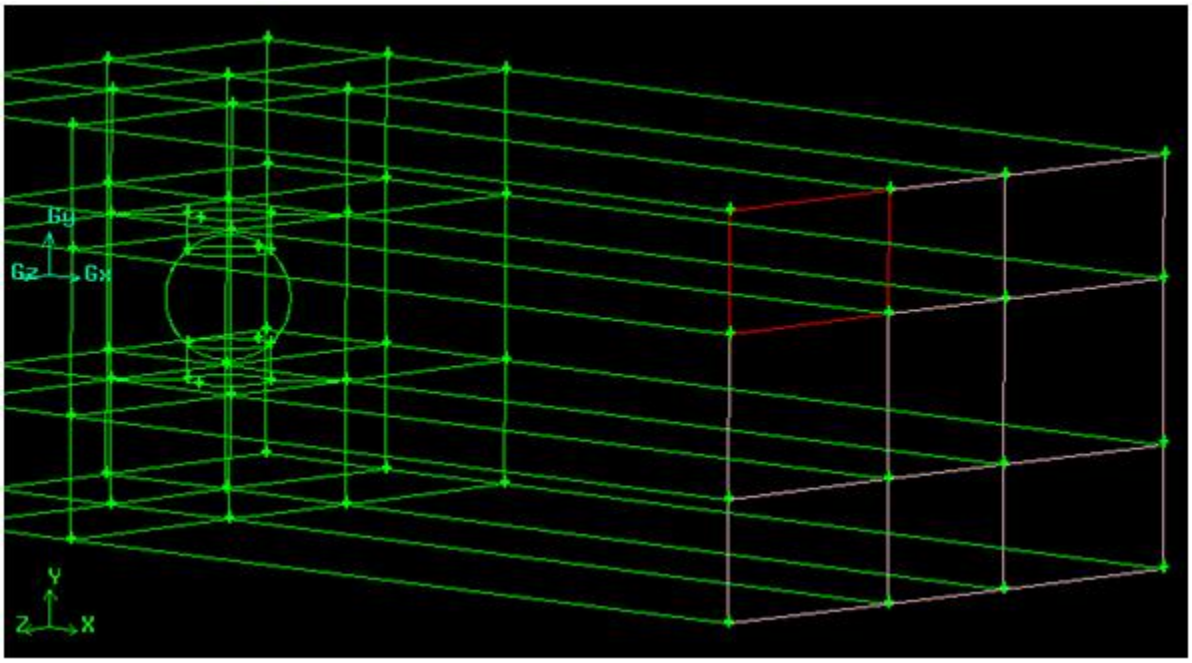


For "Name" enter "sphere" and for "Type" select "WALL." The names you give the boundary types in Gambit will be used to identify them in Fluent. Click "Apply."

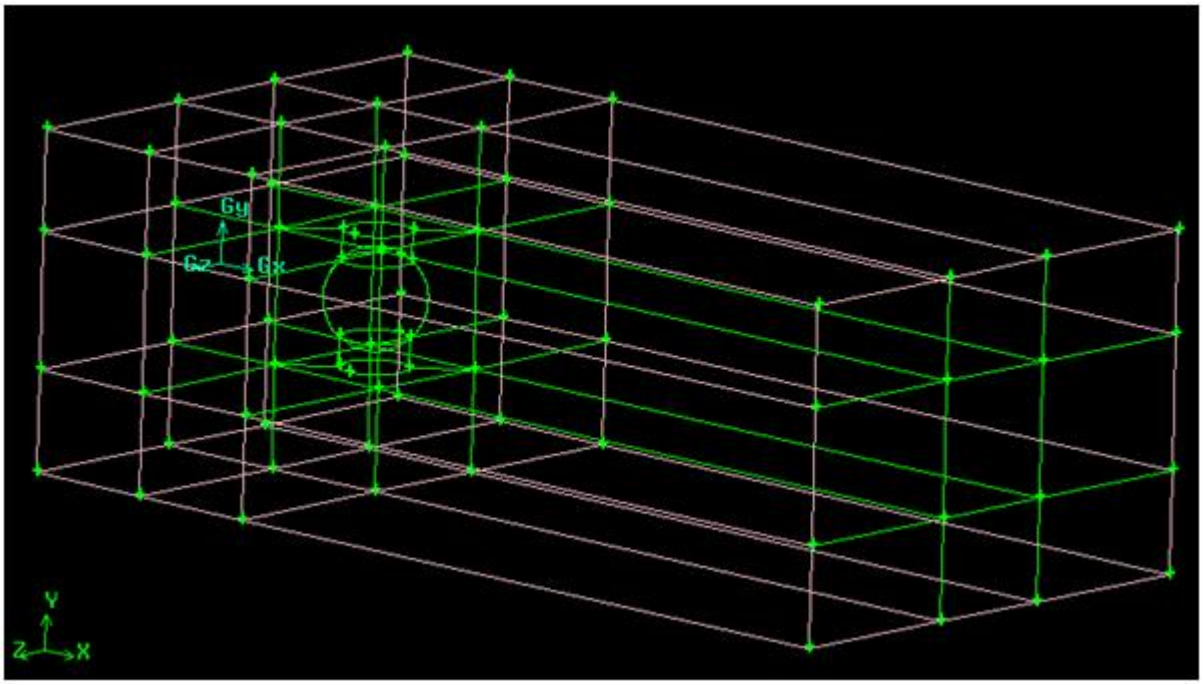


Selecting Flow Inlets/Outlets

Next the Inlets and Outlets need to be defined, establishing the flow direction through the model. The outer edges of the flow domain, which were copied and used to segment the flow domain and (as a result) were segmented themselves, will serve as the inlets and outlets. We'll address the outlets first. The flow domain side in the yz-plane and located furthest from the sphere is our outlet. There are nine faces that need to be selected. Before you start using the mouse to select faces, it's advisable that you open the list of available faces on the Specify Boundary Types panel. If you click on an edge but Gambit highlights the wrong face, just keep clicking on the same edge until the face you want is selected. Then go to the "Picked" column of the available faces list and simply remove the extra faces before moving on (the most recent selection will always be at the bottom of these lists, so if you have two extra faces selected just remove the two above the last entry in the "Picked" column).



For "Name" enter "outlet" and for "Type" select "Outflow" then click "Apply." The other five sides of the flow domain are all inlets, meaning that there are forty-five individual faces that all need to be selected together. The flow domain's corner edges (think of the original big rectangle) can be used to easily select the majority of the necessary faces.



For "Name" enter "inlets" and for "Type" select "VELOCITY_INLET" then click "Apply."

Save and Export

Your mesh is now complete. Save your work.

Main Menu > File > Save

and export the mesh as "SingleSphere.msh".

Main Menu > File > Export > Mesh...

Pay attention to the command history window while the mesh is being exported. If you missed any key faces when setting the boundary types, Gambit will report it and identify the face in question. You'll have to go back to the Specify Boundary Types command window, change "Action" to "Modify" and make whatever additions/deletions are necessary.

Go to Step 4. Set Up Problem in FLUENT

Go to all FLUENT Learning Modules