FLUENT/GAMBIT Fluids Analysis

ME 4005 & 6005

Below are step-by-step instructions on how to create a mesh in Gambit, how to save and export the mesh, how to import the mesh from within Fluent and finally, how to solve the problem using the solvers and models contained in Fluent. You will then be given a problem that you are to investigate and report on.

GAMBIT

Take care to insure that you are in the correct directory. Fire up gambit from the command prompt by typing gambit *filename.*

The first thing that you should do is to specify which solver you need from the Solver menu. Choose 'Fluent 5/6'. This will determine what type of menu popup throughout your session.

Generate a grid. There are two ways of generating a mesh. Gambit calls them 'top-down' or 'bottom-up' in the user manuals. These instructions are bottom-up. You will create vertices upon which the edges will be built upon. Connecting edges will create a face. Connecting faces will create a volume (3D). Once the face or volume is created, a mesh can be generated on it. For this example, we will stick to 2D, node -> edge -> face -> mesh. Remember to save and save often.

> **Vertex**: There are four buttons under the word OPERATIONS in the top right corner of Gambit. They are, from left to right, the *geometry*, *mesh*, *zones* and *tools* command. At this time, click on the geometry button. Note: most of the buttons in Gambit toggle off and on. The blank space under the buttons on the right hand side is now showing more buttons and windows. Directly under OPERATIONS is GEOMETRY with 5 buttons: *vertex, edge, face, volume,* and *groups.* Click on the *vertex* button.
>
> By this time, you will have noticed that as you move the mouse over the function buttons a window near the bottom of Gambit tells you what that button does. Use this function to familiarize yourself with the various buttons in Gambit.
>
> Once you have clicked on the *vertex* button more buttons appear below. Click on the button directly below the vertex button called Create Vertex. A floating window called Create Real Vertex appears below. Here you may enter the coordinates of the vertices in your problem. Don't worry about local coordinates at this time. Enter your coordinates in the global area.
>
> As you enter in the vertices, they will show up as white X's in the view area. If you cannot see them they may be outside of your viewing area. To remedy this,

click on the Fit to Window button, the top left big button in the GRAPHICS/WINDOWS CONTROL area (near bottom right).

If at any time you wish to undo the command you just did, look for the button that has the arrow that is 'spinning' from right to left. The Undo command can undo more than one command, just keep clicking.

For more complicated geometry, such as an airfoil, the vertex data can be imported. Go to File -> Import -> Vertex Data. Enter the path to the file or us the browser. The data file Gambit can read has to have the file extension.dat. The format of the data in the file must be tab or space delimited. Airfoil data can be downloaded to your account from the following web site:
http://amber.aae.uiuc.edu/~m-selig/ads/coord_database.html

Most of the data downloaded from the internet will typically need to be modified. There should be no text in addition to the data and a column of zeros for the z-axis will need to be added.

Edges: Once the vertices are created, you want to create edges connecting them. Under GEOMETRY, click on the edge button (second from left). When the EDGE buttons pop up, right click on the first button on the left. A drop down list will appear giving different options for the edge type. When one of these options is selected a floating window will be displayed. To create smooth curved edges use the NURBS option. There are two methods for the NURBS, interpolate and approximate. The approximate method with a tolerance of zero will give a smooth curve. To select the vertices for the NURBS line left click the up arrow on the right side of the yellow vertices box. Select the vertices with the mouse and click on the ---> button. Once the vertices are selected, the final one will turn red and the others will turn pink. If the vertices are the ones you want to connect with an edge then click Apply in the floating window. An edge will appear in yellow. Use this procedure to create an edge for the top and bottom of the airfoil and the control volume.

Face: Under GEOMETRY, click on the *face* button (third from left). When the FACE buttons pop up click on the first button on the left: Create Face. A floating window called Create Face From Wireframe will appear. Selecting an edge is the same as selecting a vertex. Hold the shift key down and left click on the edge. The edge will turn red. Select a second edge: the first will turn pink and the second will turn red. Select all edges comprising the face and click Apply in the window. A face will be created, its color is light blue. To create a single face from two faces use the Boolean Operations Subtract option.

Mesh: A mesh can now be created on the face. Under the OPERATION button, click on Mesh Command button. Where the word GEOMETRY used to be, the word MESH will appear with five buttons: *boundary-layer*, *edge, face*, *volume* and *group*. You want to mesh the face that you have just created, so click on *face*. Click on the top left button in the FACE menu area, the button is called:

Mesh Faces. This will cause the Mesh Faces floating window to pop up. Let everything stay at its default, select the face and click Apply. Gambit may hesitate while it's thinking and then you will see the mesh in yellow. You can play around with mesh spacing but keep the elements and type at Gambits default setting.

Boundary Conditions: You can set or change the boundary conditions in Fluent but you can also do it in Gambit, in fact, it's a little bit easier. Up in the OPERATIONS menu; click on the Zones button. Under the word ZONES two buttons will appear: Specify Boundary Types and Specify Continuum. Click on the Specify Boundary Types button. A floating window called Specify Boundary Types will appear. Make sure that at the top of this window the solver name 'Fluent 5/6' appears, if not go to the solver menu and choose 'Fluent 5/6'. You must have this correct as different solvers specify BC's differently.

Change the Entity pop down menu to edges. Select the edge that will be the velocity inlet and under the Type pop down menu choose Velocity_Inlet. It is recommended that you label the different edges. This will help you keep track of them in the Fluent output reports. The labels must be one word, i.e. no spaces or tabs. To finish creating the BC click *Apply*. Now select the edge that will be the outlet and choose Outflow. The top and bottom edges of the airfoil and control volume are Walls. There is a list at the top of this window that should reflect the two BC's that you have created.

Save and Export: The file that you have been saving to throughout the session is a Gambit file and is different from a mesh file. To create the mesh file for Fluent to import click on File -> Export -> Mesh.   The next pop up window will have file type (UNS/RAMPANT/FLUENT 5) and file name. Type in the name as you please but keep the .msh filename extension.   If the geometry is 2D, then check the box “Export 2d Mesh”.

You may now quit Gambit.

FLUENT

Once Fluent is loaded, type 2d at the command prompt. The instructions below should be followed roughly in the order that they are written. A choice in one menu may alter choices in another menu. Note that in the title bar of the Fluent window, there are the descriptors: [2d, segregated, lam]. These will help you keep track of what models/solvers you are using.

Read in the grid. There are two files that contain data that Fluent needs to solve problems. The first is the Case file, which stores all of the information on boundary conditions, what solvers and models were used, etc. The second is the Data file, which stores the solution. Even though the mesh is not a true case file, read in the mesh using the File -> Read -> Case command. You will have to navigate to your working directory. Fluent will tell you what it is doing as it reads in the mesh. Be sure to follow this dialog and spot any errors.

Check, Display and Scale the Grid. Fluent assumes that the grid units are in meters (SI units). If you created your mesh in Gambit in anything other than meters you will need to scale the grid. Go to Grid -> Scale. In the pop up window, change the pull down menu "Grid Was Created In" to whatever units you used in Gambit (ft). Click on Scale, the Xmax and Ymax fields should now reflect the proper values. Close the window.

The proper values should also be shown when you check the grid. Go to Grid -> Check. A list of statistics will appear in the activity window.

To display the grid, go to Display -> Grid. A display window will show the grid. The inlet will be blue, the outlet red, walls white and the mesh green. This will give a visual check on the boundary conditions. There are two ways to fix incorrectly specified BC's. One way is to fire up Gambit, redo the BC's, export the correct mesh and re-import the mesh into Fluent. Another quicker way is to fix them within Fluent. This will be explained in the Boundary Conditions section.

Definition of Properties: Once the grid is correct, you can define the properties of your problem. The three crucial categories under Define are Models, Materials and Boundary Conditions. It is a good habit to specify these in order; for example, changes in the Models menu will change the menus in Boundary Conditions.

Models -> Solver: You will not change anything here, however, take a look around and familiarize yourself with the various options available for solvers.

Models -> Energy: Turn the energy equation on or off (default).

Models -> Viscous: There are various assumptions used when numerically solving the governing equations. The first is to assume that the flow is inviscid. The second is to assume laminar flow (default), no turbulence. And the third is to turn on a turbulence model, for example, the k-epsilon turbulence model. How will each of these assumptions affect your solution? Which turbulence model is correct for your flow configuration? Do you have to worry about wall effects?

Materials: The default fluid is air. If you'd like to change the material then click on the Database button, choose the material that you need and click on Copy. This copies the material properties from the database into Fluent. Once this is done, make sure that the material that you want to use is in the Name field. Close the Materials window.

Boundary Conditions: The condition for the velocity inlet is the only BC that needs to be set. Change the velocity magnitude (m/s) and direction to the desired values. To set the conditions at the inlet, click on the word velocity_ inlet in the Zone area. The Type area should adjust to reflect "velocity-inlet". If you wanted to change this inlet to something else, in the Type list click on the boundary condition needed. A window will pop up to make sure that this change is what you want, click yes or no. If the specification is correct, click on the Set button.

Enter the velocity and click OK in the pop up window. Close the Boundary Conditions window.

*It is a good time to save all of this informatio*n. Go to File -> Write -> Case. *Navigate to whatever directory and save. You can change the name but keep the .cas filename extension. When you type in a name the .cas extension will automatically be added. You will see this verified in the activity window.*

Solution Parameters: The problem is set up and we now need to look at solution parameters. You should keep an eye on the iterations as they (hopefully) converge to the solution. To do this, go to Solve -> Monitors -> Residual. You will see that the Print box is already checked. This means that the residuals will print to the activity area. If you'd like the residuals to be plotted to the screen, click on the Plot box. While you're here, take a note of the convergence criterion. Is this okay for your particular problem? Once you are done, click OK.

Fluent uses iterative techniques to solve for the steady state solution. Iterative methods need a first guess. To initialize the solution domain, go to Solve -> Initialize -> Initialize. In this window you can put in the initial values of the velocity field. Compute From allows you to initialize the flow field to the values specified on various boundaries. Go ahead and Compute From velocity_inlet.#. Click on Init to initialize the flow field, the Apply button is used at another time. Once the field is initialized, close the window.

Solve: You can now solve the problem. Go to Solve -> Iterate. A window appears that has a field where you can specify how many iterations to perform. It is always good to choose a small number at first to see if the solution is going wild (i.e. incorrect BC's) or seems to settle down. Enter a small number, say 20 and click on Iterate. You'll see the residuals printed in the activity area and the plot of residuals displayed in the plot window. If everything is okay, then put in a higher number, say 100, and click Iterate. Do this until Fluent says "Solution Converged".

Display the Solution: Is your solution correct? You'll want to view vector, contour and XY plots of your data. Under the Display menu, you'll see Contours, Velocity Vectors, Path Lines and Particle Tracks among others. These will plot the various quantities for you. If you'd like to see an XY plot of, for example, the temperature along a wall, go to Plot -> XYPlot.

Hardcopy: Fluent does not have the capability of printing the plots that it generates. It does however, let you save these plots as postscript, tiff, EPS, PICT, etc., files. To do this, display the plot that you want to save in the display window. Then go to File -> Hardcopy. A window will appear which have all of the formats listed. For your report choose TIFF. Choose color or gray scale. DO NOT change the resolution field. I know it says "0" but it will work out ok. Click on Save and navigate to your working directory.

Saving Case and Data files: You've done a lot by now so it would be good to save both the case and data files. Go to File -> Write -> Case & Data. Navigate to where you want to be and click okay. Since you have already saved the case file, it will ask you if you want to overwrite. If you don't, click cancel and save the data file separately using File -> Write -> Data.

You may now quit Fluent and all of the information is safely stored in the case and data files. When you want to review or change this simulation just fire up Fluent and read in the case and data files, File -> Read -> Case & Data. You can change inlet conditions and recalculate a different solution, or try new models and save them as different case and data files.