15.2.5. Step-wise methodology of FLUENT usage for a problem

15.2.5.1. Problem-1: Flow through a circular pipe

Problem Statement:

Consider a fluid (air) flowing through a circular pipe at an inlet velocity 1 m/s. The inlet temperature is 300 K and the pipe wall outlet temperature is 600 K. The length and diameter of the pipe are 50 mm and 1 mm respectively.

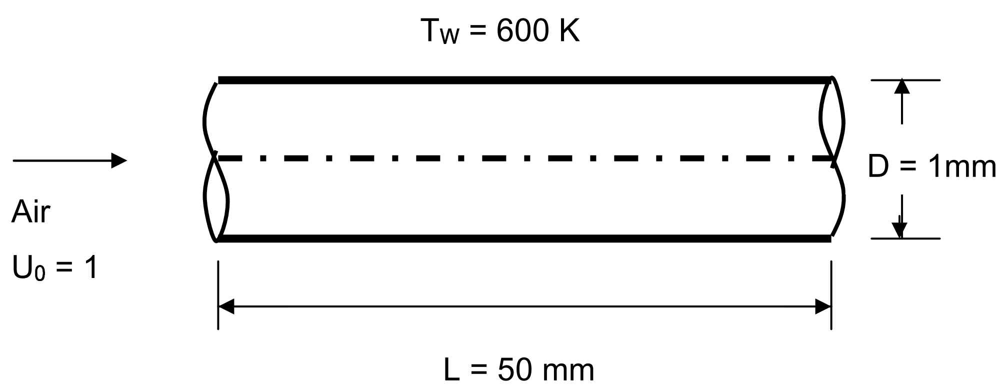


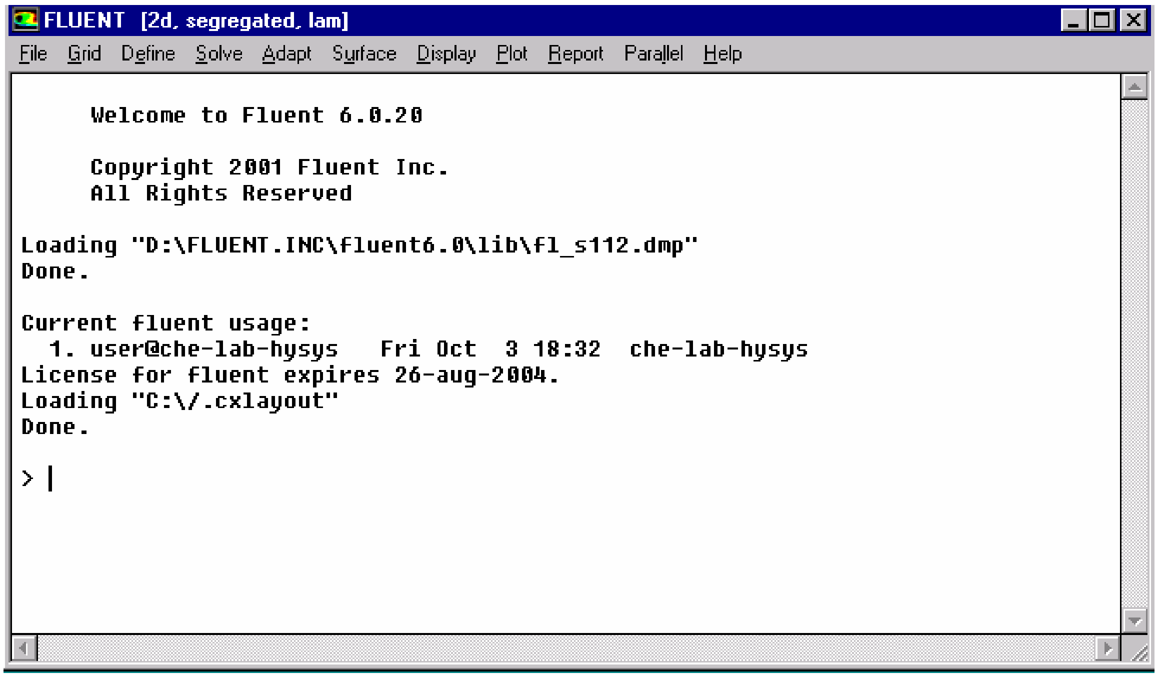
Fig. 15.6. Flow through a circular pipe.

Find the temperature, pressure and velocity contours with the following assumptions:

1. The flow is steady and incompressible.
2. The fluid properties viscosity and density are constant.

Step 1: Starting FLUENT on a Windows System

Click on the Start button, select the Programs menu, select the Fluent.Inc menu, and then select the FLUENT 6 program item. The opening screen should look something like this.



Step 2: Choosing the Solver Formulation

FLUENT provides three different solver formulations:

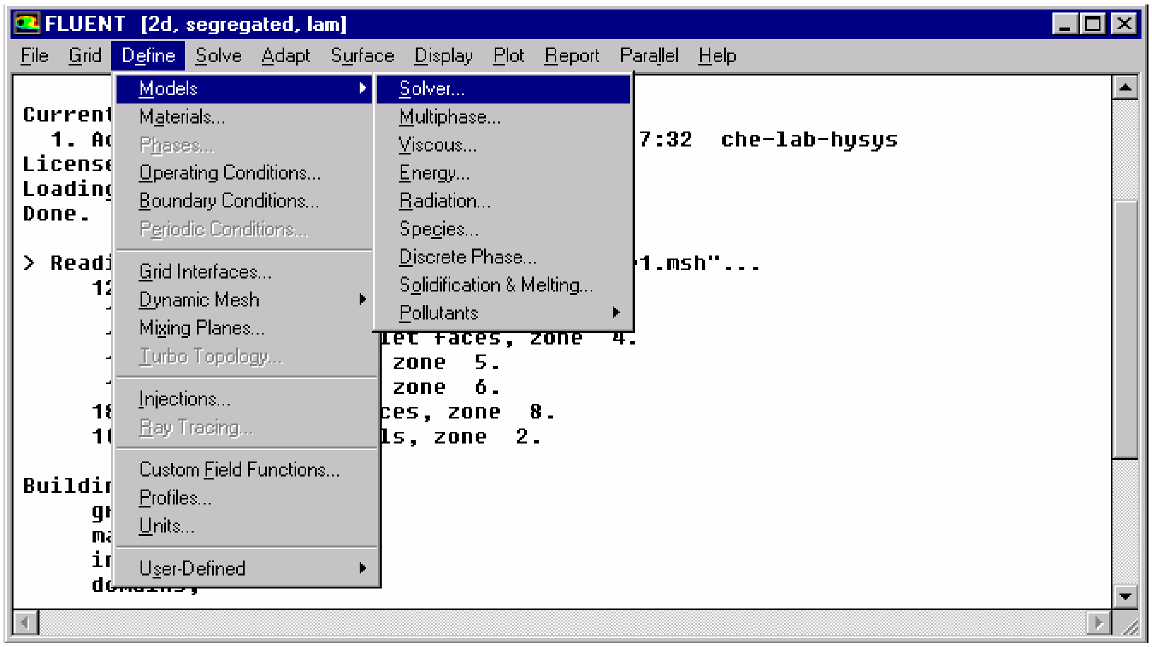
- Segregated
- Coupled implicit
- Coupled explicit

All three solver formulations will provide accurate results for a broad range of flows, but in some cases one formulation may perform better than the others. The segregated and coupled approaches differ in the way that the continuity, momentum, and energy and species equations are solved: the segregated solver solves these equations sequentially, while the coupled solver solves them simultaneously

➢ User Inputs for Solver Selection

To choose one of the three solver formulations, you will use the **Solver** panel (Figure 1).

Define → **Models** → **Solver...**



To use the segregated solver, retain the default selection of **Segregated** under **Solver**. To use the coupled implicit solver, select **Coupled** under **Solver** and **Implicit** (the default) under **Formulation**. To use the coupled explicit solver, select **Coupled** under **Solver** and **Explicit** under **Formulation**.

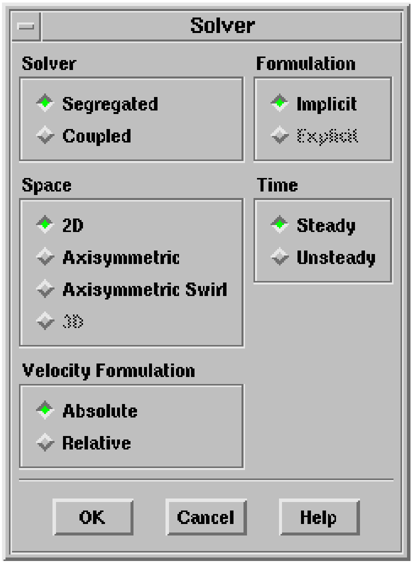


Fig. 15.7. The Solver Panel

For this problem-segregated, 2D, Steady state absolute model solver has been selected.

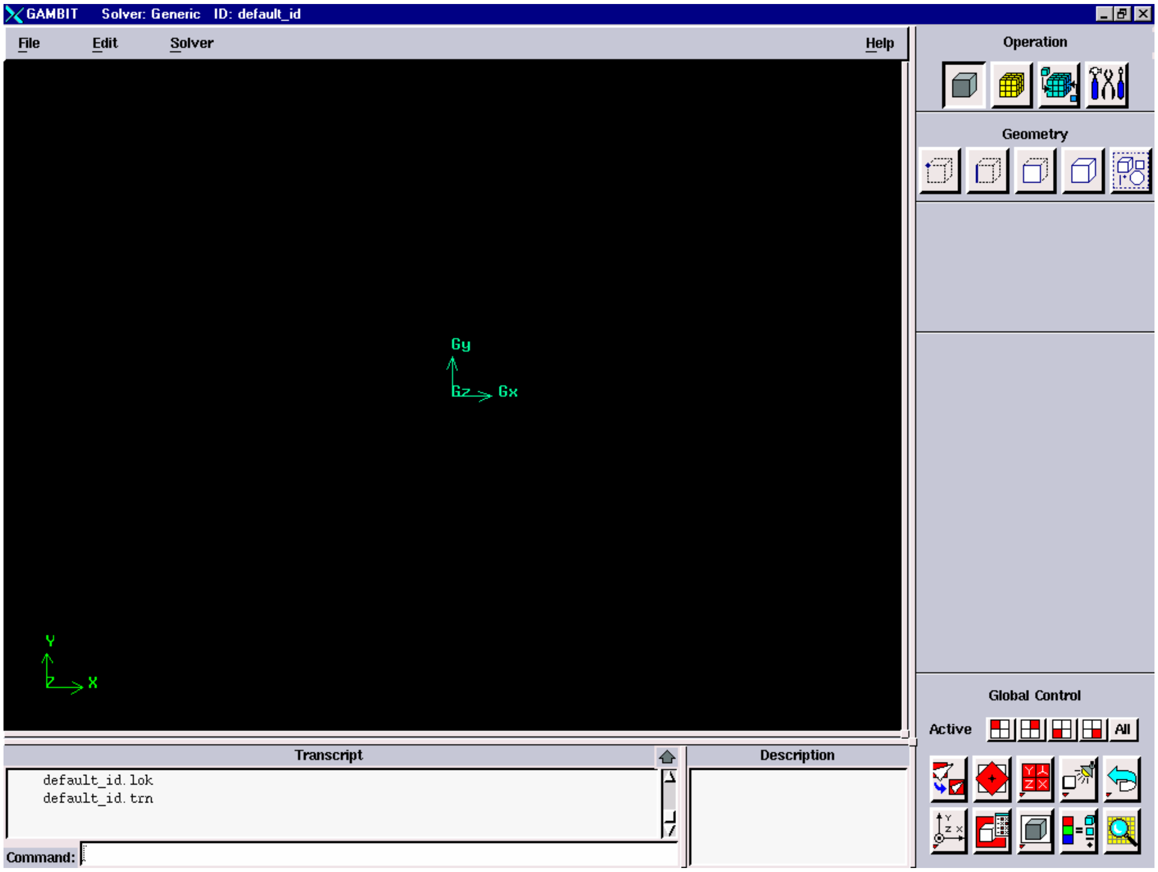
Step 3: Reading Mesh Files

Mesh files, also known as grid files, are created with the **GAMBIT** grid generator. A grid file is--from **FLUENT**'s point of view--simply a subset of a case file. The grid file contains the coordinates of all the nodes, connectivity information that tells how the nodes are connected to one another to form faces and cells.

- **GAMBIT Grid Files**

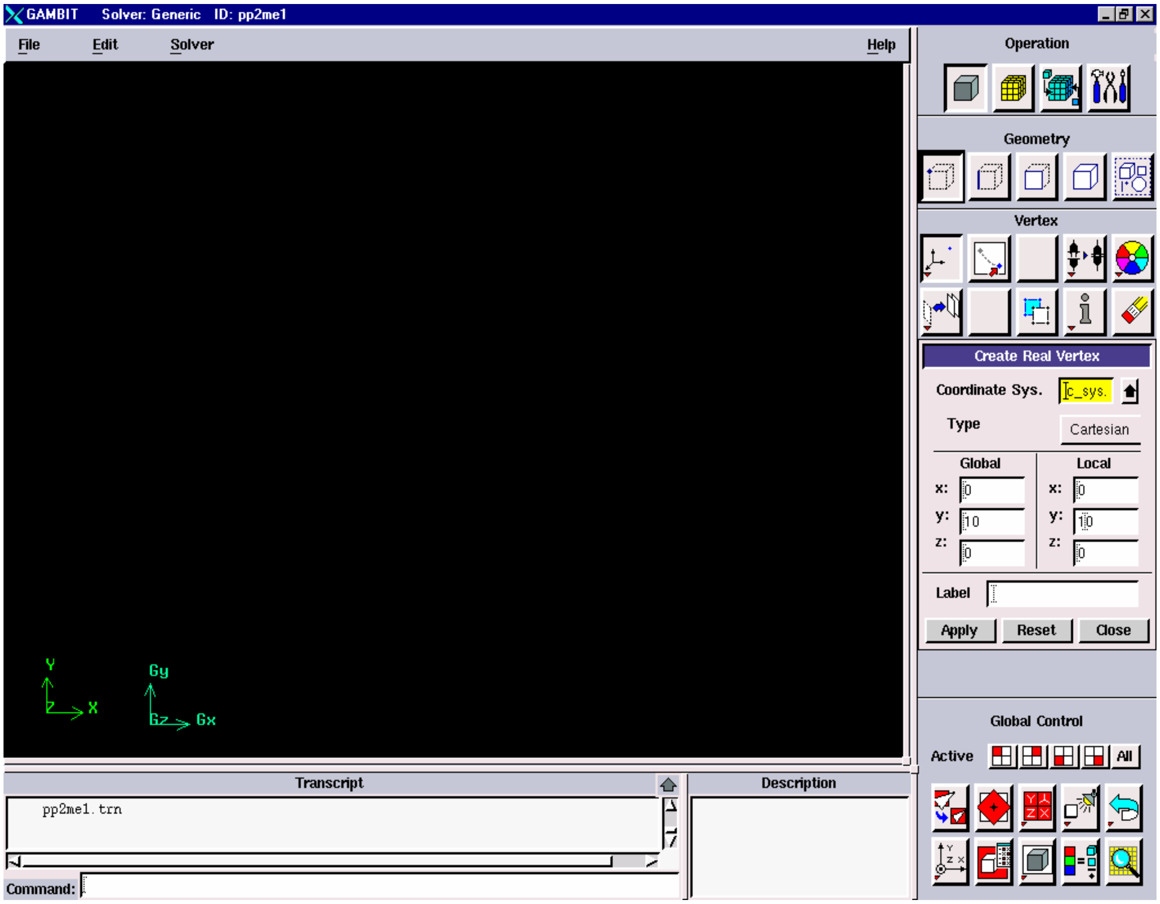
GAMBIT is used to create 2D and 3D structured/unstructured/hybrid grids. GAMBIT is a software package designed to help analysts and designers build and mesh models for computational fluid dynamics (CFD) and other scientific applications. GAMBIT receives user input by means of its graphical user interface (GUI). The GAMBIT GUI makes the basic steps of building, meshing, and assigning zone types to a model. To create any of these meshes for **FLUENT**, follow the following procedures

Click on the exe file of the GAMBIT. The opening screen should look like this.



➤ *Creating the Geometry*

When you click the Geometry command button on the Operation toolpad, GAMBIT opens the Geometry subpad. The Geometry subpad contains command buttons that allow you to create, move, copy, modify, summarize, and delete vertices, edges, faces, and volumes. The Geometry subpad also contains a command button that allows you to perform operations involving groups of topological entities.The symbols associated with each of the Geometry subpad command sets are shown as below.



GAMBIT modeling forms require you to specify a location in space relative to a specified coordinate system. For example, the **Create Real Vertex** form shown in Fig. 2 requires you to specify the three coordinates describing the point at which the vertex is to be created.

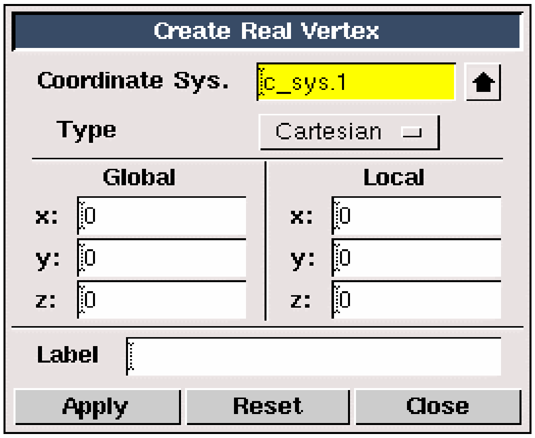
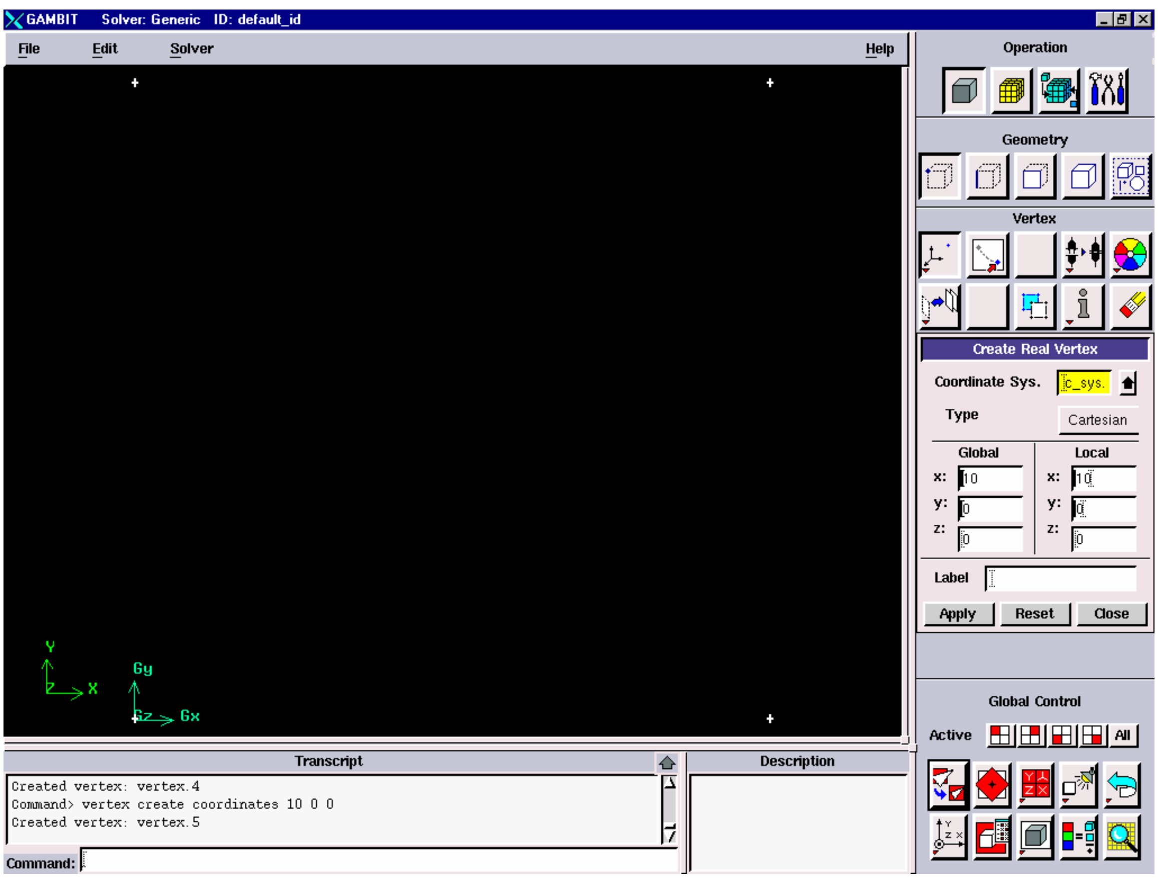


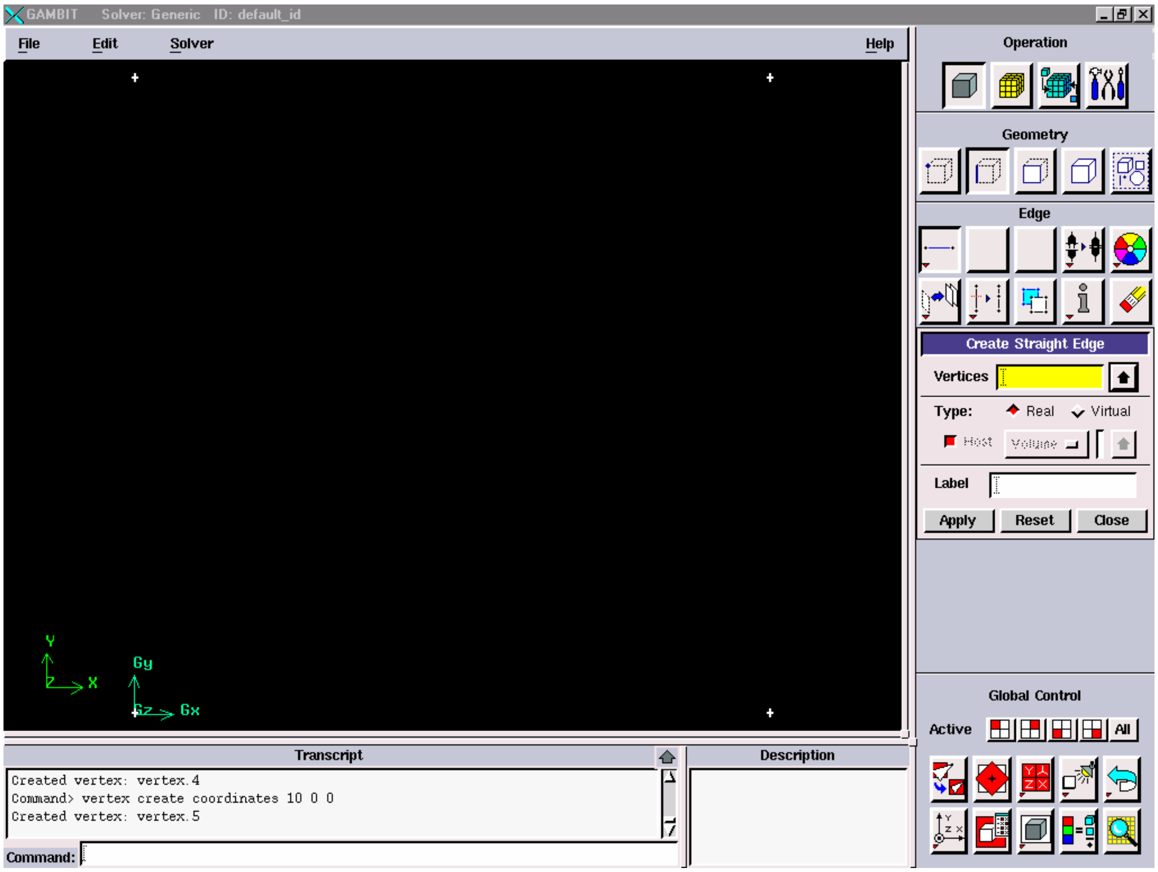
Fig. 15.8. Creating Real Vertex

By specifying the co-ordinates in create real vertex form the four vertices look as below.

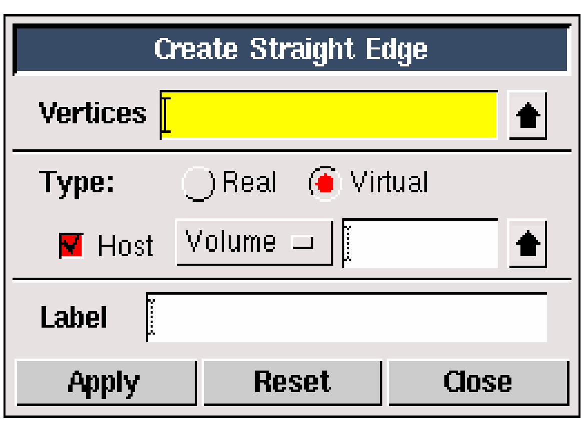


To create Edges

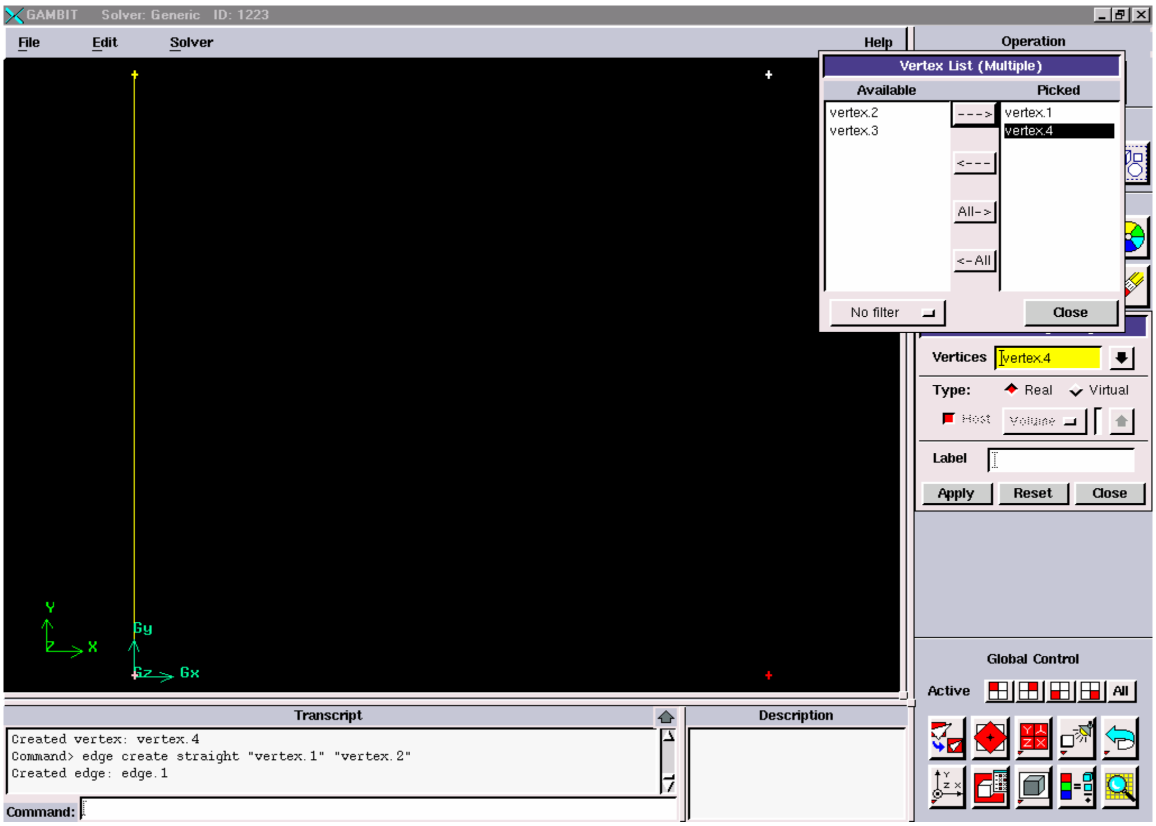
The following commands are available on the **Geometry/Edge** subpad.



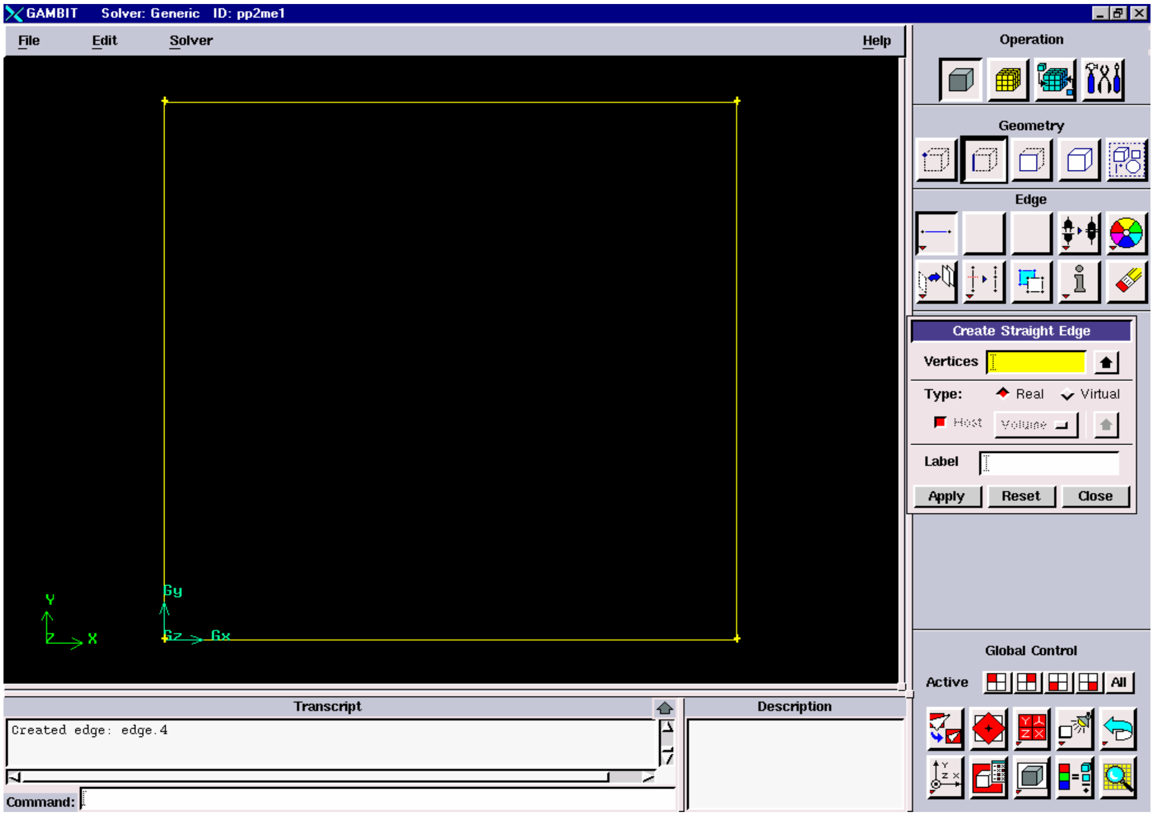
To open the **Create Straight Edge** form (see below), click the **Create Straight Edge** to open the command button on the **Geometry/Edge** subpad



To create an edge by means of straight edge command, you must specify two vertices that comprise the end points of the edge. The edge starts from first (start) vertex to the second (end). For example a straight-line edge is created between vertex1 and vertex 4 as shown below.

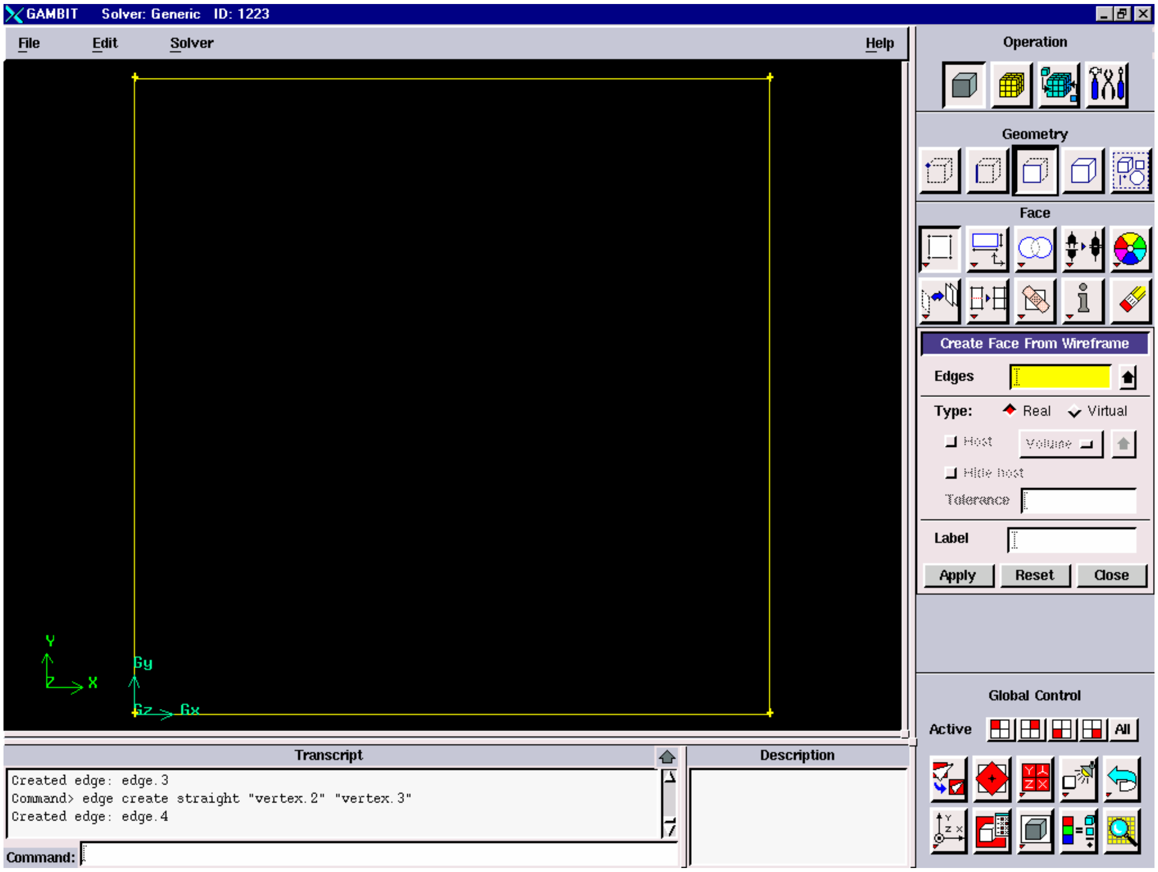


Similarly all edges are created.



- **To create Face**

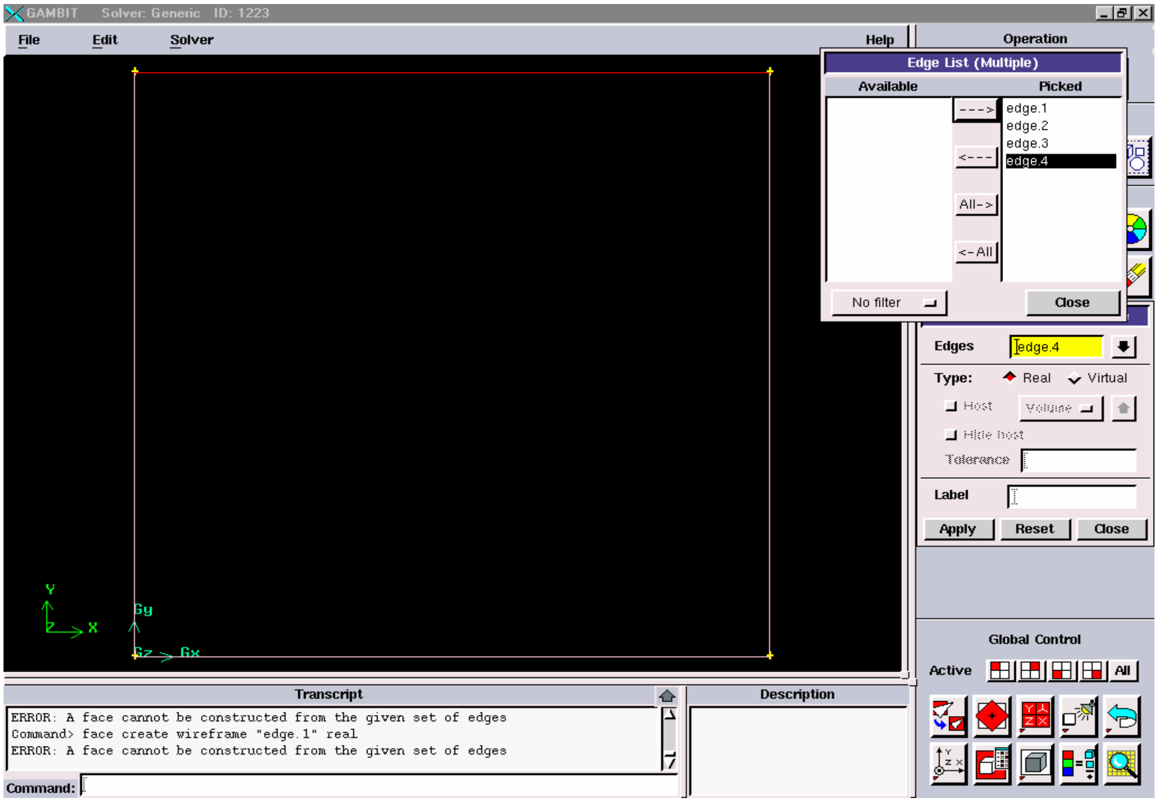
Create Face From Wireframe command allows you to create a face from three or more existing edges. The following commands are available on the **Geometry/Face** subpad.



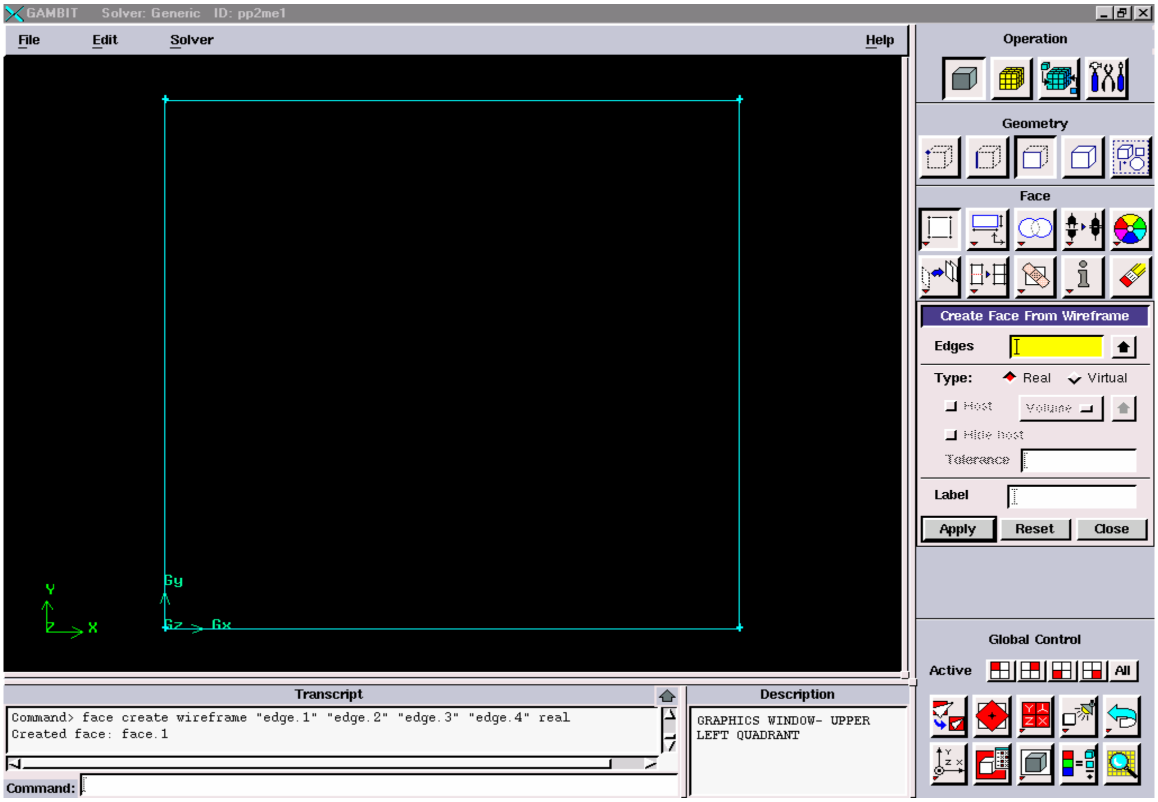
To create a face by means of the **Create Face From Wireframe** option, you must specify the following parameters:

- The edges that define the wire frame
- The face type—real or virtual.

The face is created as shown.

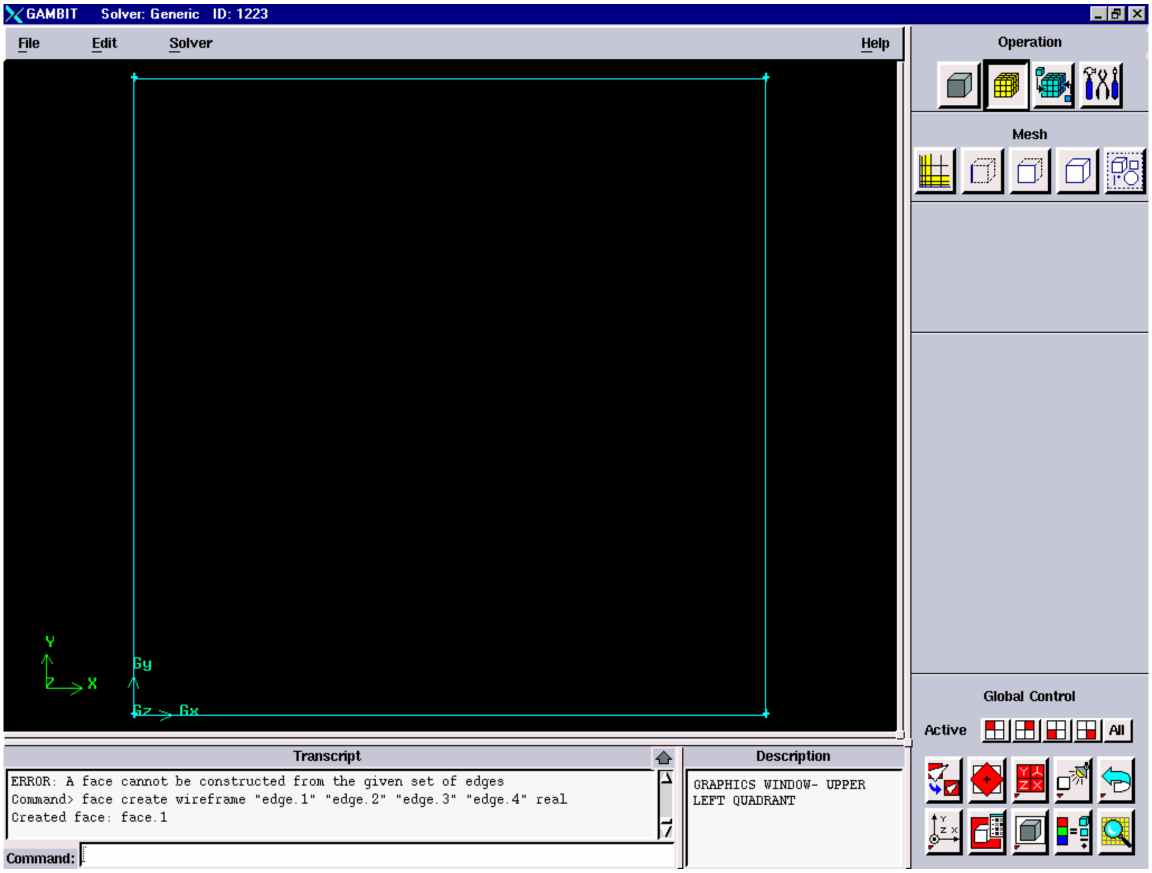


The final face window looks like this.



➢ Meshing The Model

When you click the **Mesh** command button on the **Operation** toolpad, GAMBIT opens the **Mesh** subpad. The **Mesh** subpad contains command buttons that allow you to perform mesh operations involving boundary layers, edges, faces, volumes, and groups. The symbols associated with each of the **Mesh** subpad command sets are as follows.



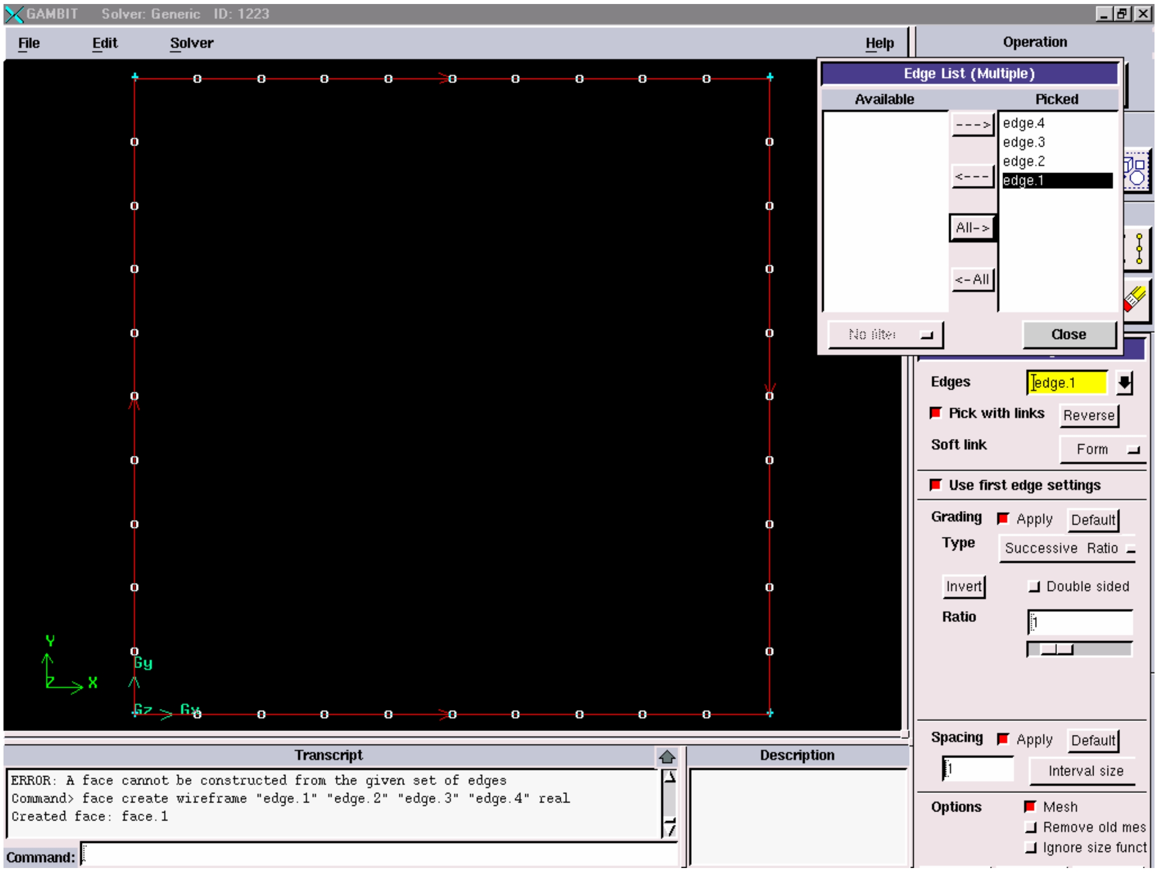
Mesh Edges

The **Mesh Edges** command allows you to grade or mesh any or all edges in the model. When you *grade* an edge, GAMBIT applies the mesh node spacing specifications but does not create mesh nodes on the edge. When you *mesh* an edge, GAMBIT creates mesh nodes according to the specifications.

To perform a grading or meshing operation, you must specify the following parameters:

- Edge(s) to which the grading specifications apply
- Grading scheme
- Mesh node spacing (number of intervals)
- Edge meshing options.

These operations are shown below



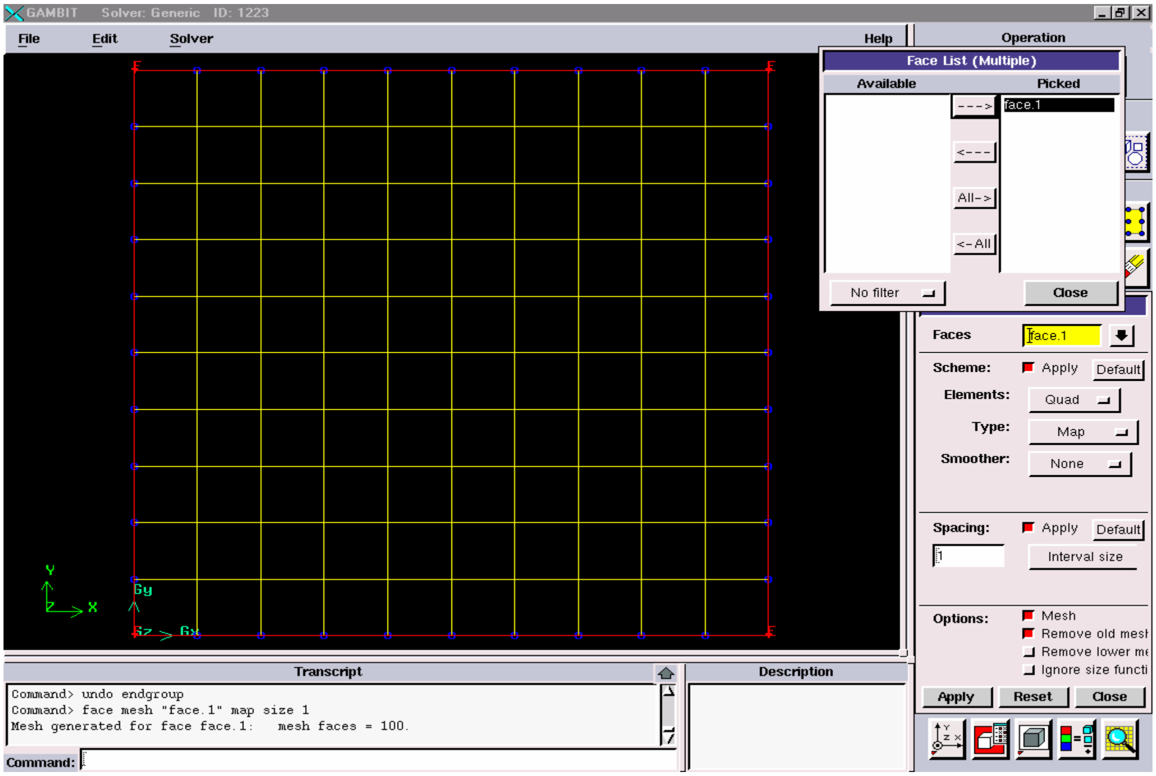
Mesh Faces

The **Mesh Faces** command allows you to create the mesh for one or more faces in the model. When you mesh a face, GAMBIT creates mesh nodes on the face according to the currently specified meshing parameters.

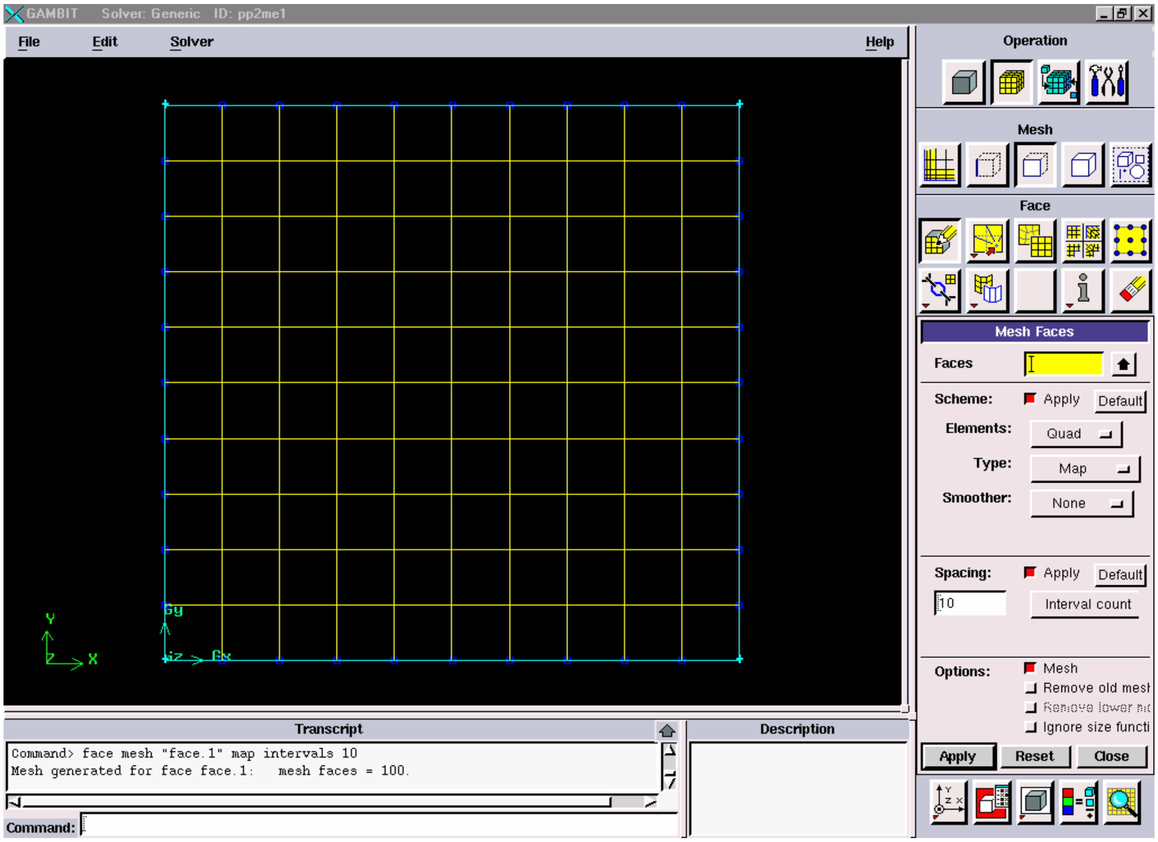
To mesh a face, you must specify the following parameters:

- Face(s) to be meshed
- Meshing scheme
- Mesh node spacing
- Face meshing options

The operations are shown below



After specifying the faces to be meshed and the interval size, the final mesh looks as such.

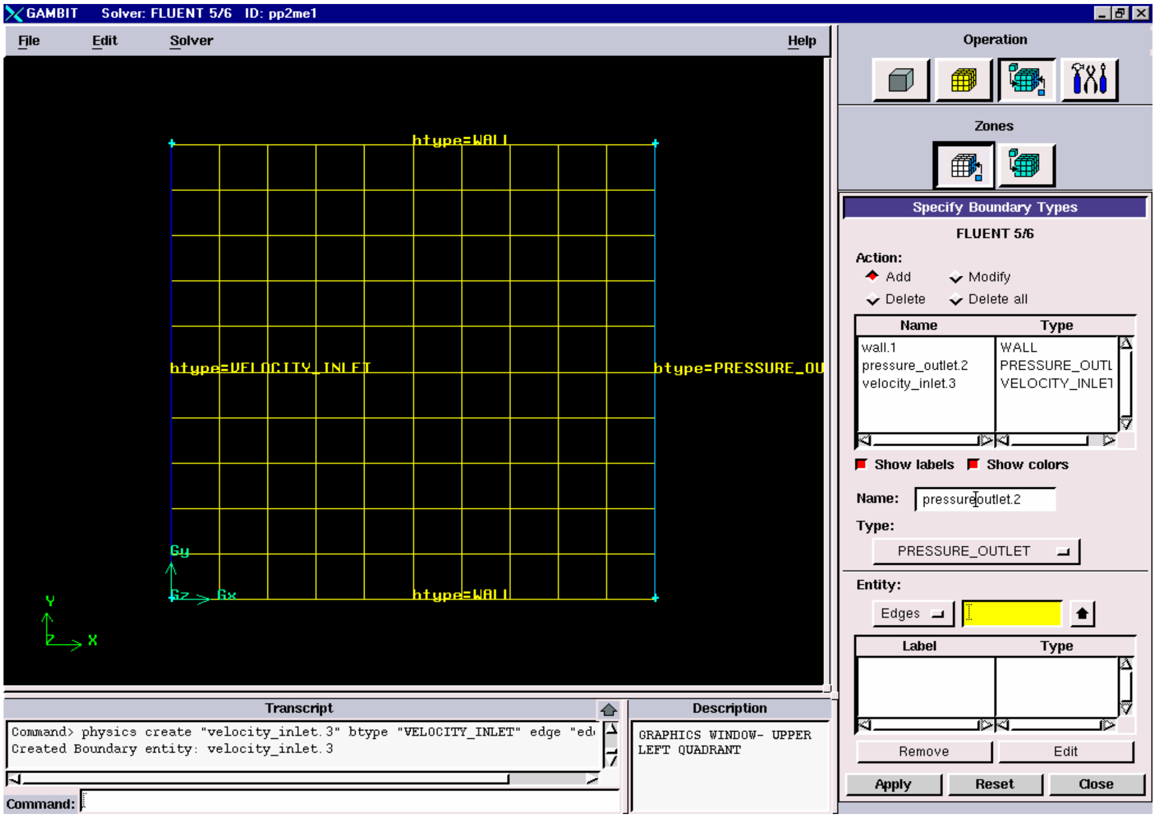


Specifying Zone Types

Zone-type specifications define the physical and operational characteristics of the model at its boundaries and within specific regions of its domain. There are two classes of zone-type specifications:

- Boundary types
- Continuum types

Boundary-type specifications define the physical and operational characteristics of the model at those topological entities that represent model boundaries. For example, if you assign an INFLOW boundary type specification to a face entity that is part of three-dimensional model, the model is defined such that material flows into the model domain through the specified face.



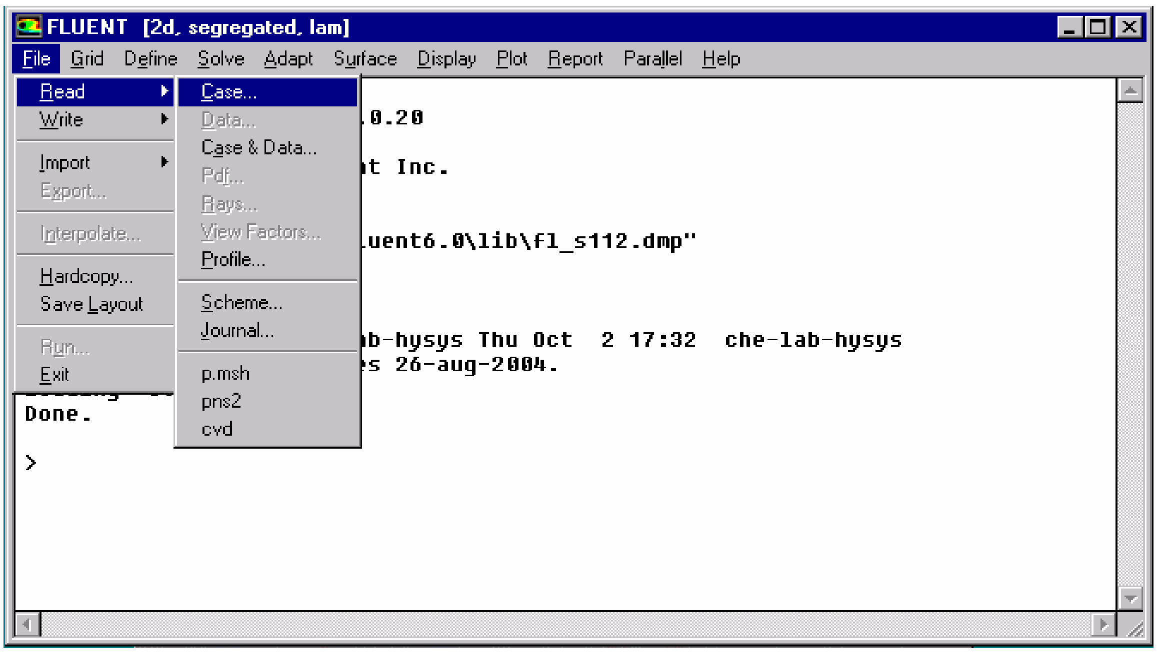
Export the Mesh File to Fluent

The mesh file is then exported to fluent by opening the file menu and then exporting the mesh file **File** → **Export** → **mesh…**

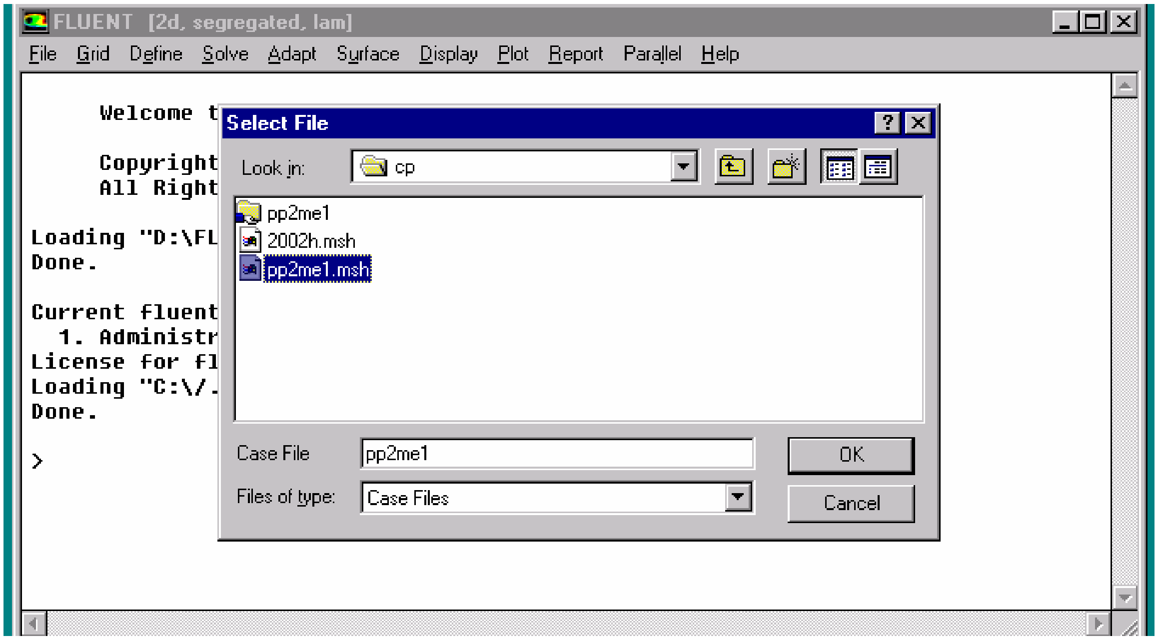
Step 4: Reading And Importing Gambit Mesh Files

FLUENT 5/6 grid created in GAMBIT, is read into FLUENT using the File/Read/Case...Menu item**. File →Read →Case...**

Selecting the Case... menu item will open the Select File dialog box, in which you will specify the name of the file to be read. If you have saved a neutral file from GAMBIT, rather than a FLUENT grid file, you can import it into FLUENT using the File/Import/GAMBIT... menu item. **File →Import →GAMBIT...**



Selecting the File From the Directory

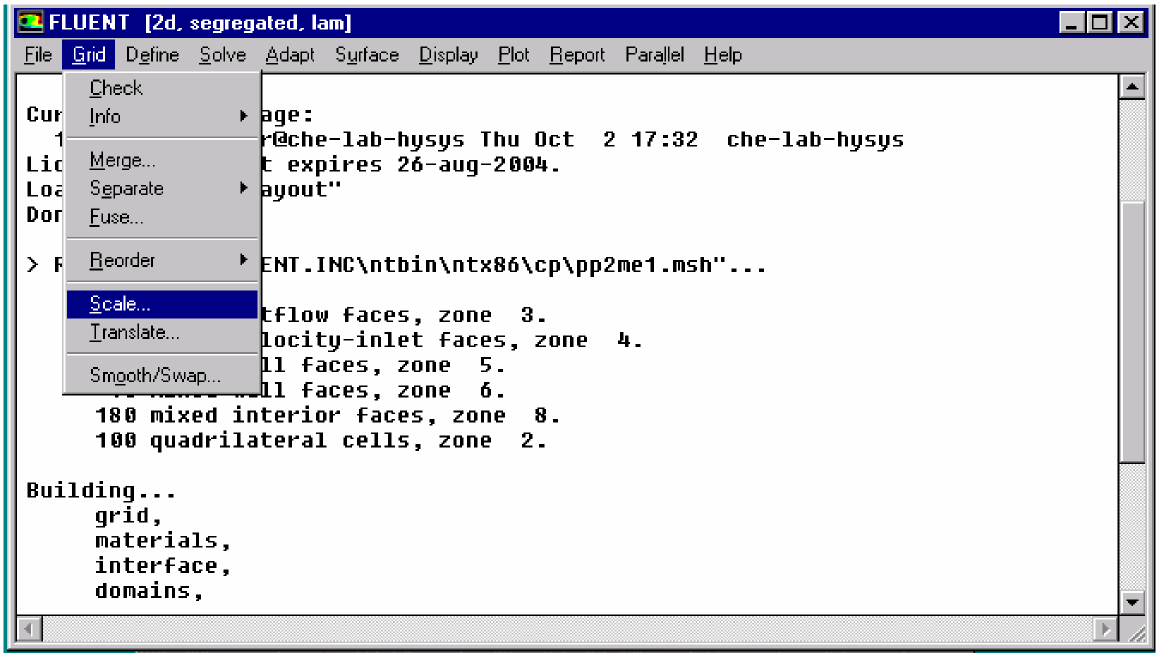


After selecting the corresponding mesh file the FLUENT window looks like this

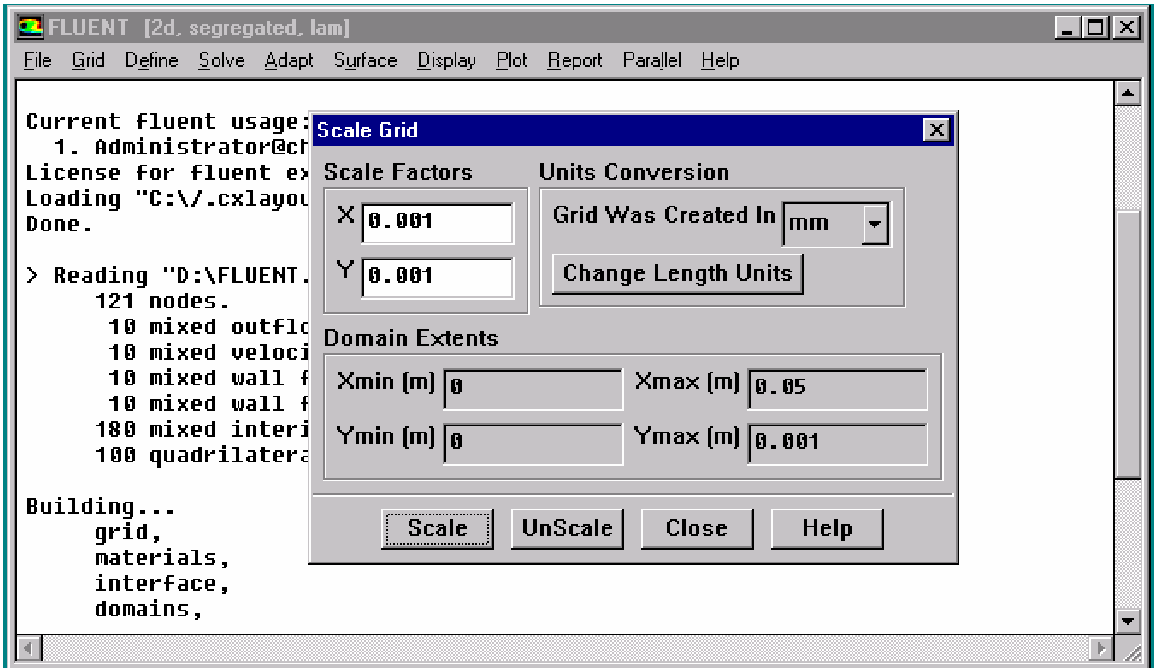
```
     Welcome to Fluent 6.0.20

     Copyright 2001 Fluent Inc.
     All Rights Reserved

Loading "D:\FLUENT.INC\fluent6.0\lib\fl_s112.dmp"
Done.

Current fluent usage:
  1. Administrator@che-lab-hysys Thu Oct  2 17:32  che-lab-hysys
License for fluent expires 26-aug-2004.
Loading "C:\/.cxlayout"
Done.

> Reading "D:\FLUENT.INC\ntbin\ntx86\cp\pp2me1.msh"...
     121 nodes.
      10 mixed outflow faces, zone  3.
      10 mixed velocity-inlet faces, zone  4.
      10 mixed wall faces, zone  5.
      10 mixed wall faces, zone  6.
     180 mixed interior faces, zone  8.
     100 quadrilateral cells, zone  2.

Building...
     grid,
     materials,
     interface,
     domains,
     zones,
        default-interior
        wall.7
        wall.8
        velocity_inlet.9
        outflow.10
        fluid
     shell conduction zones,
Done.
```

Step 5: Scaling the Grid

The grid scaling is done as shown below

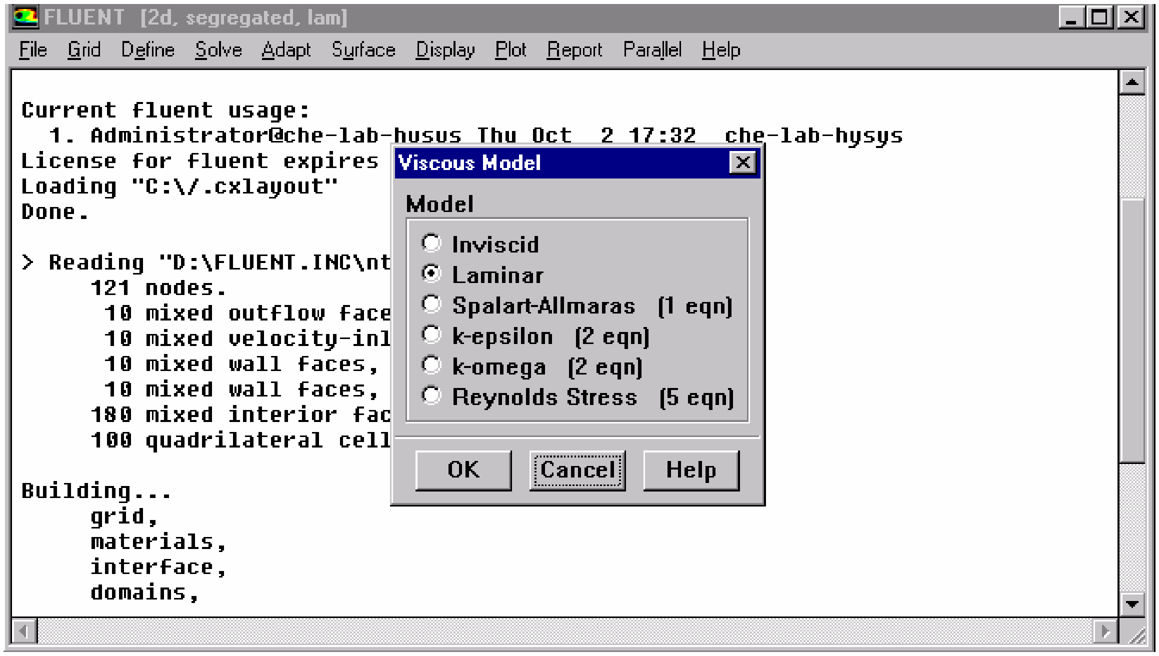


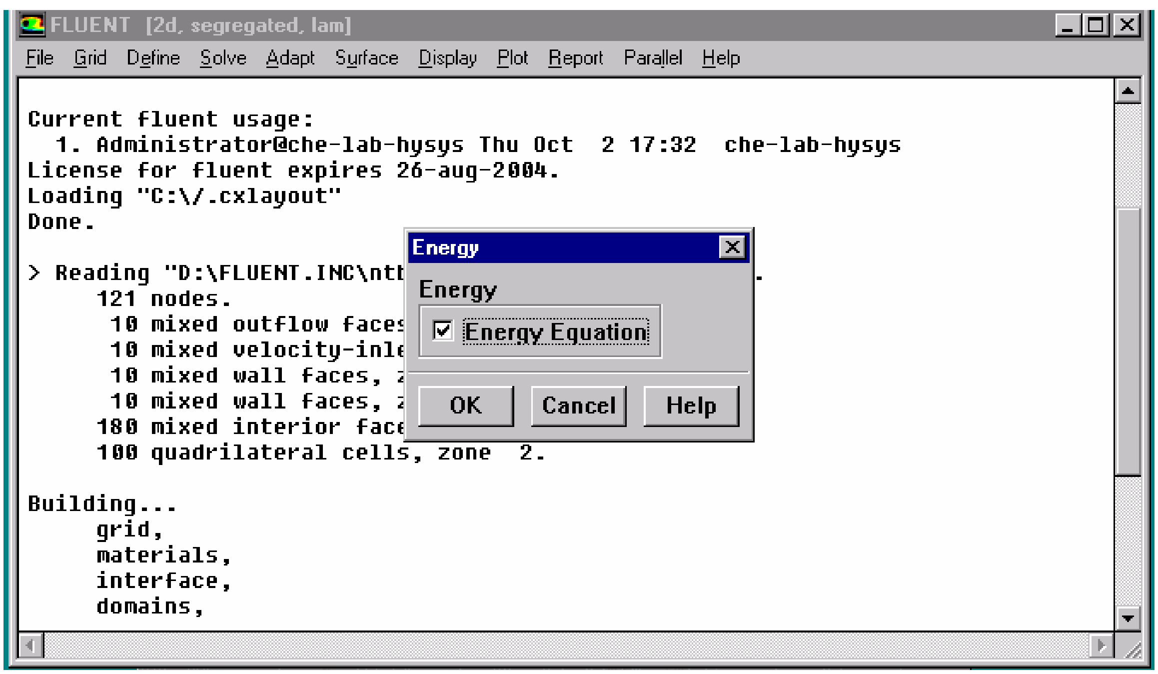
After clicking the Scale, the window appears as shown below. Enter the dimension of the problem using Units Conversion in the Scale Grid form.



Step 6: Defining the Model

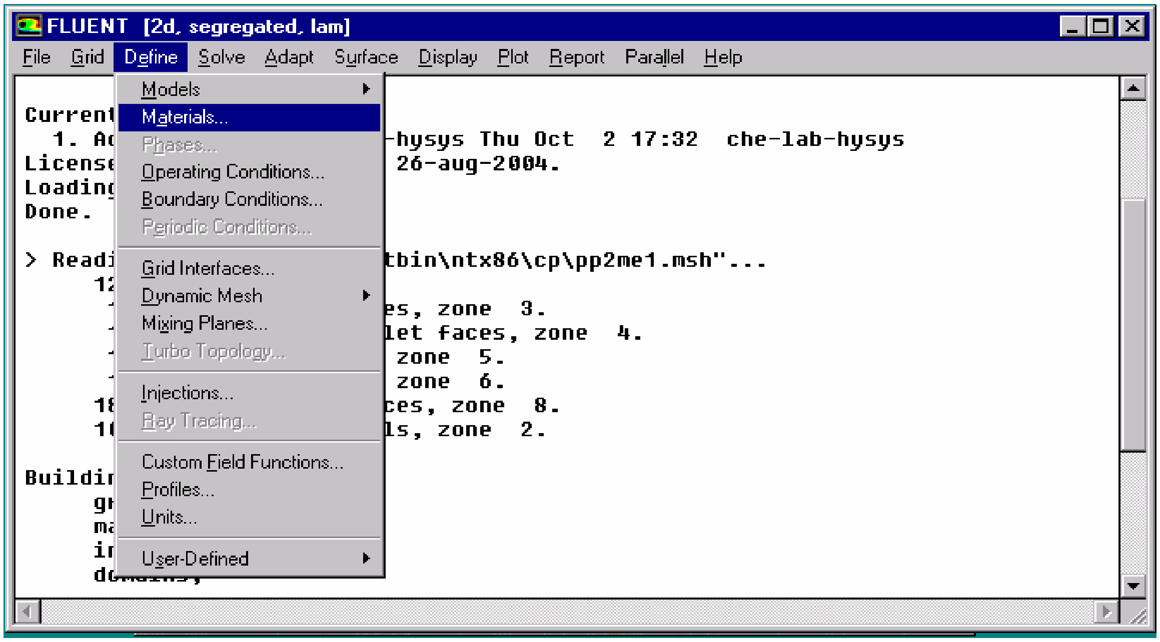
➢ **Selecting the Viscous and Energy model**



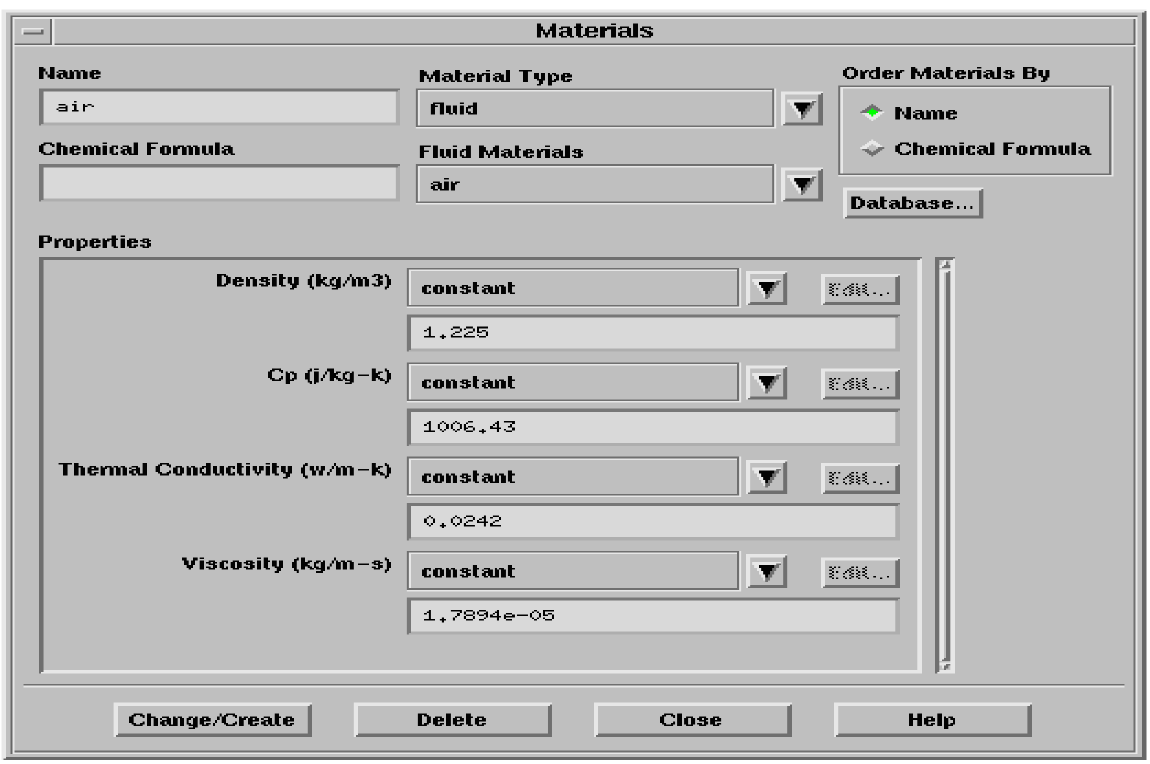


Step 7: Selecting The Materials

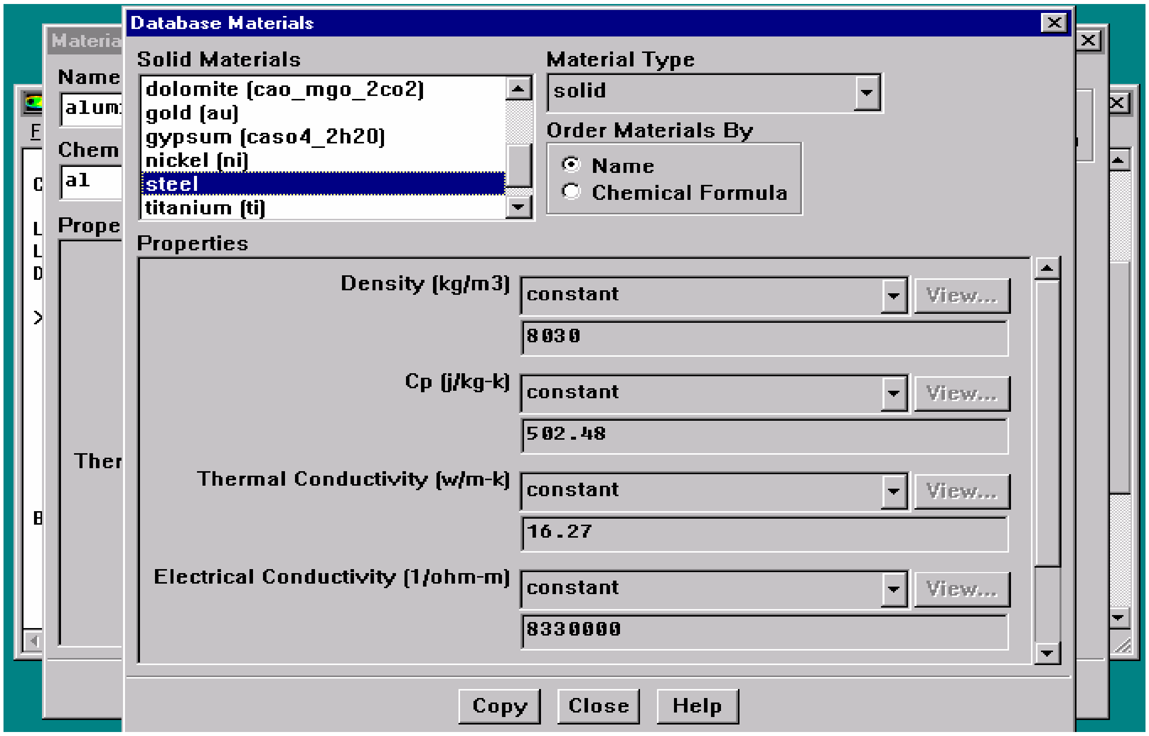
The **Materials** panel allows you to create new materials, copy materials from the global database, and modify material properties. **Define** → **Materials...**

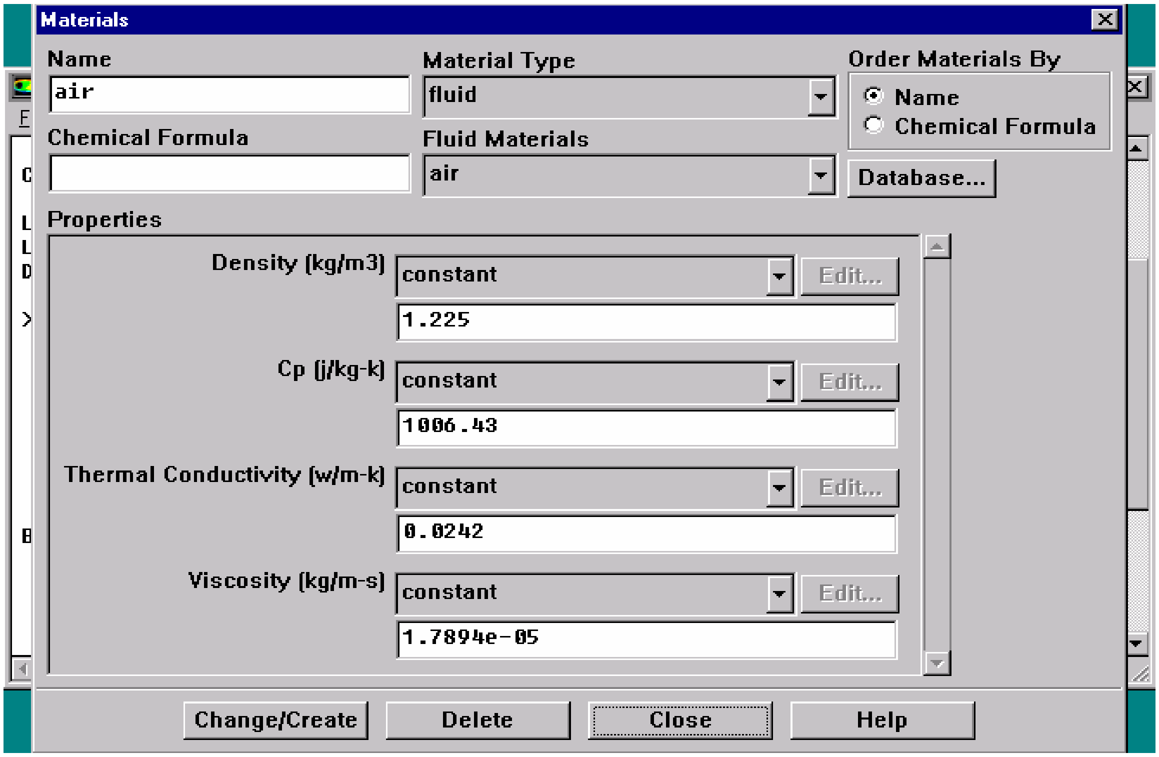


The Materials Panel looks as such



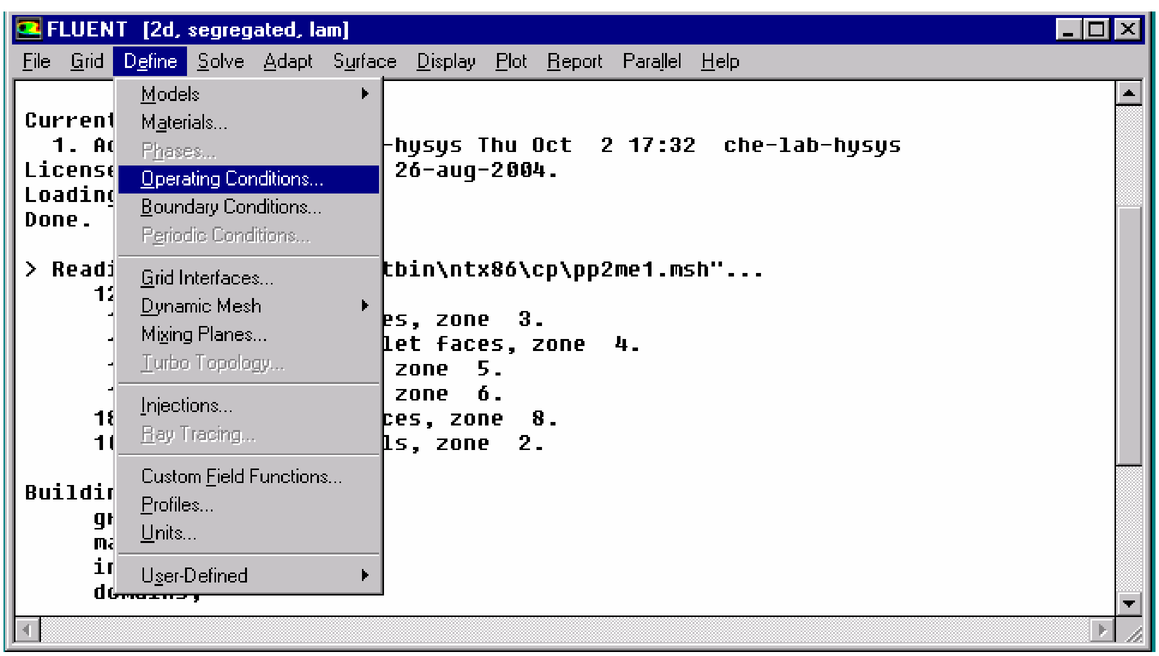
For this problem solid selected is steel and the fluid selected is air from the database as shown below.



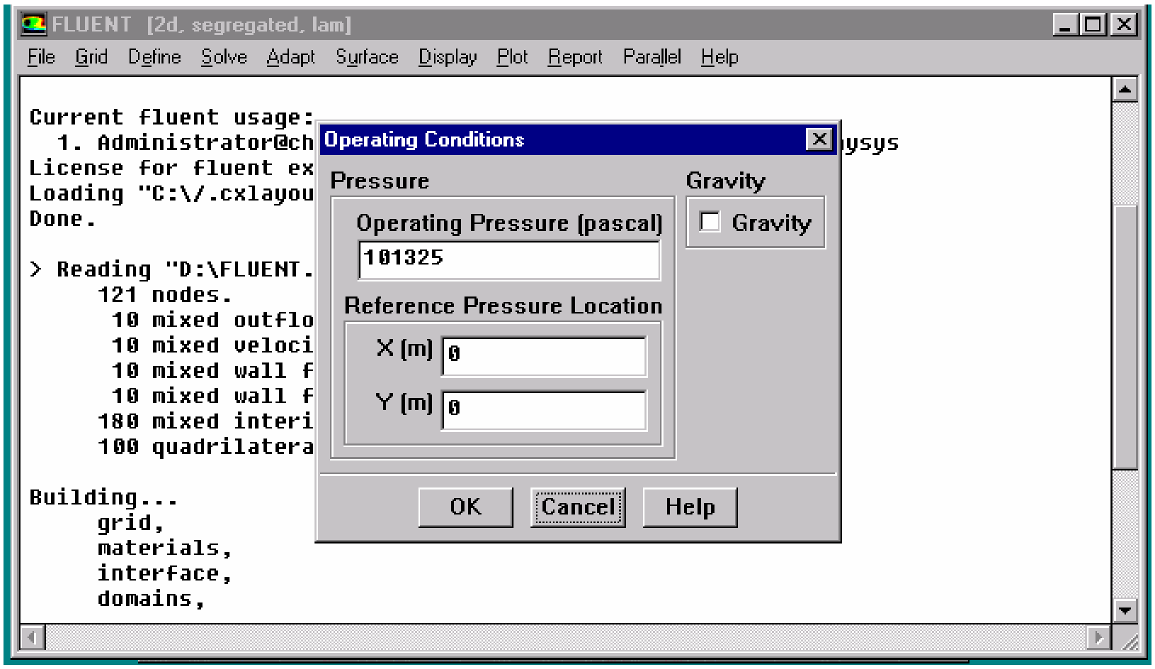


Step 8: Selecting the Operating Conditions

The operating conditions panel allows you to select appropriate operating conditions for the given problem. **Define → Operating Conditions...**



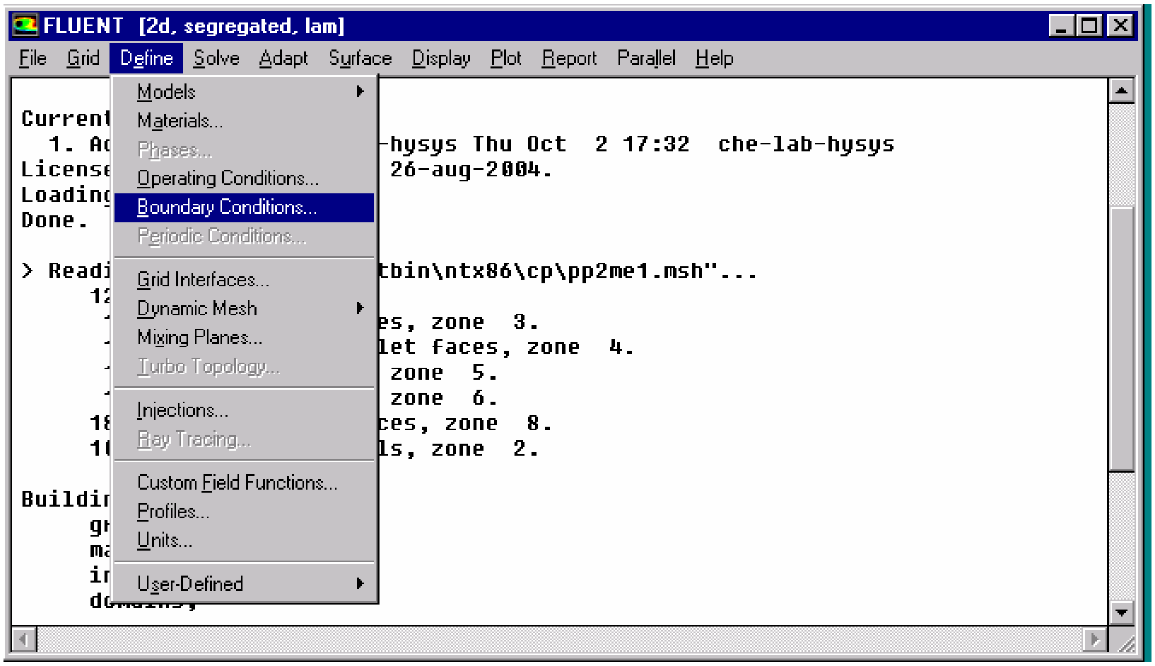
The Operating Condition panel looks as such



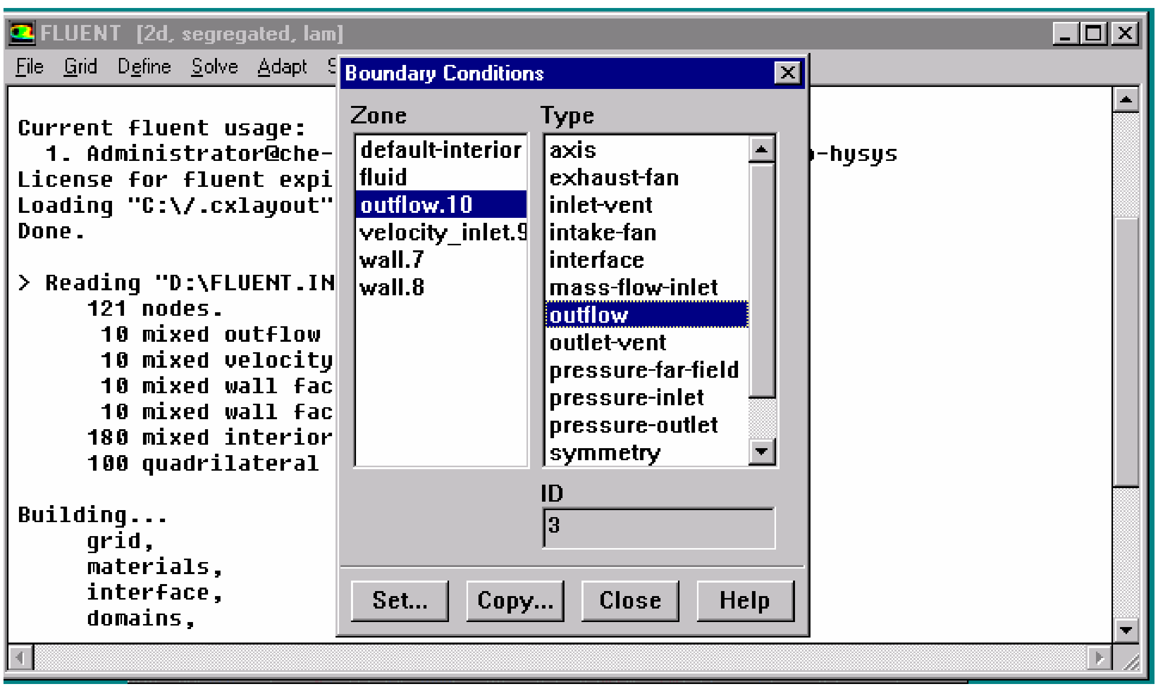
Step 9: Selecting the Boundary Conditions

The **Boundary Conditions** panel_allows you to change the boundary zone type for a given zone and open other panels to set the boundary condition parameters for each zone.

Define →Boundary Conditions...

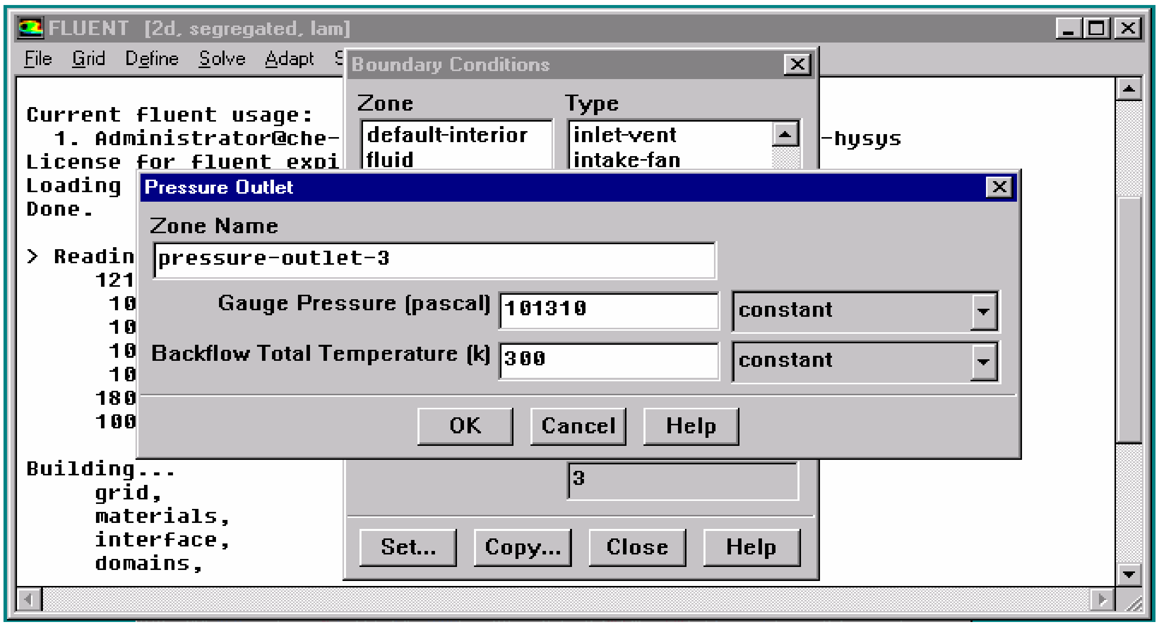


- **Selecting Outflow Conditions**



After specifying the zone as outflow.10 and type as Pressure Outlet the window appears as below where the outlet pressure is entered. Pressure outlet boundary conditions require the specification of a static (gauge) pressure at the outlet boundary. The value of static pressure specified is used only while the flow is subsonic. Should the flow become locally supersonic, the specified pressure is no longer used; pressure will be extrapolated from the flow in the interior. All other flow quantities are extrapolated from the interior.

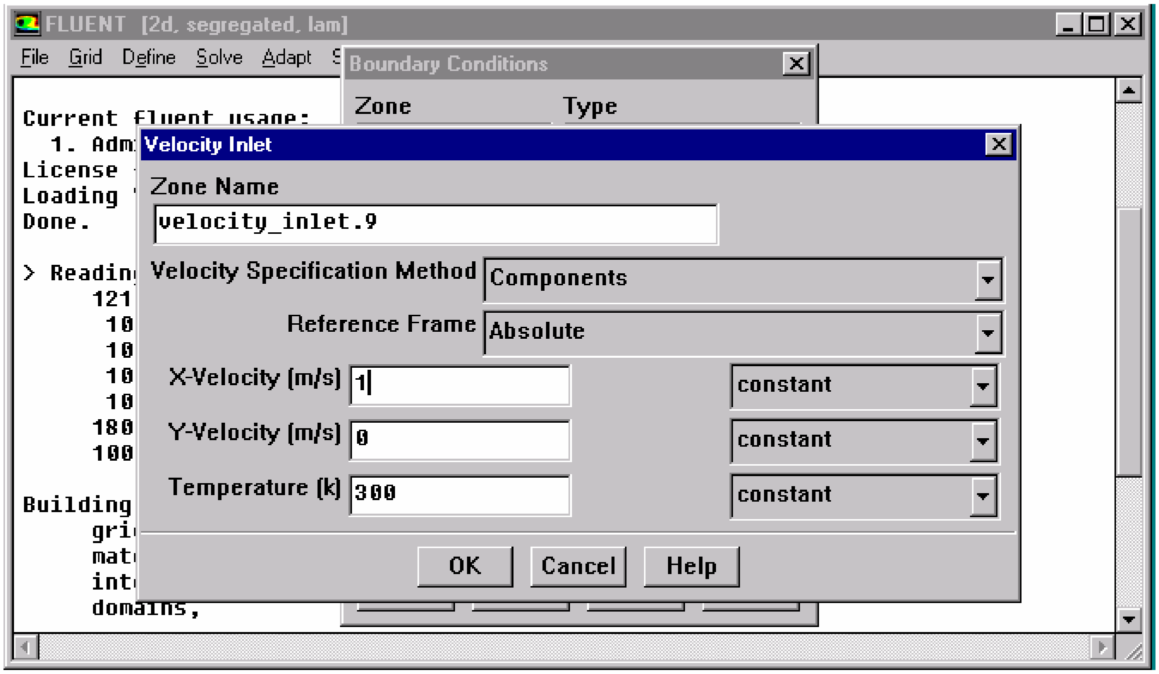
A set of ``backflow" conditions is also specified to be used if the flow reverses direction at the pressure outlet boundary during the solution process. Convergence difficulties will be minimized if you specify realistic values for the backflow quantities.



Similarly velocity inlet and wall conditions are entered.

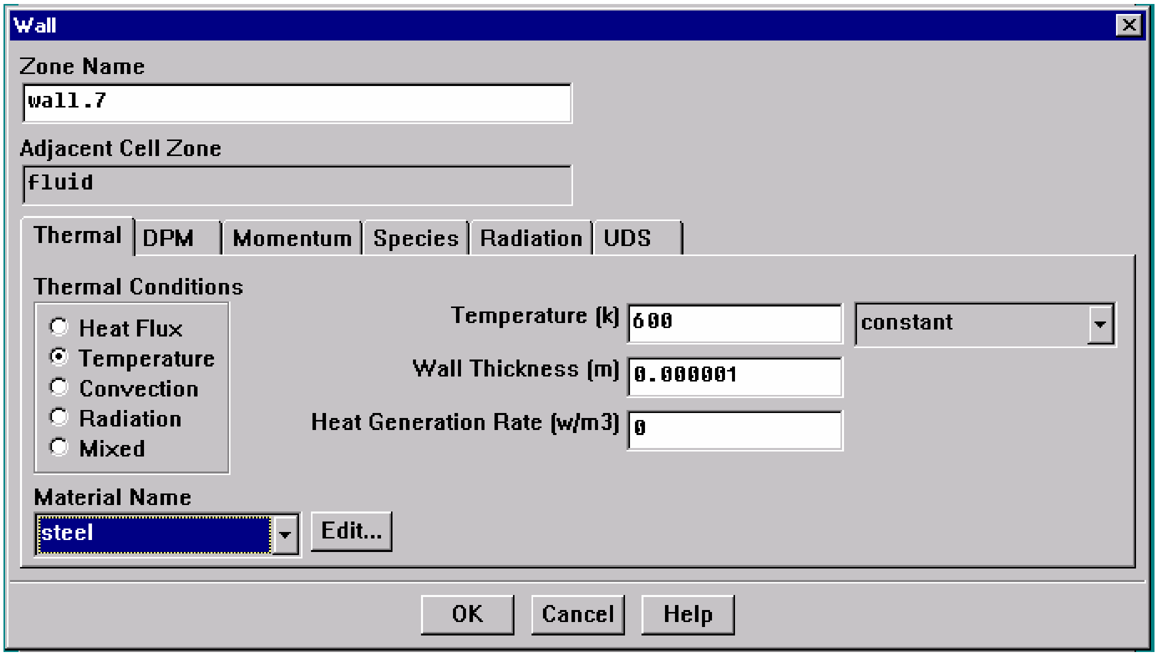
➢ Selecting the velocity inlet

Velocity inlet boundary conditions are used to define the flow velocity, along with all relevant scalar properties of the flow, at flow inlets.



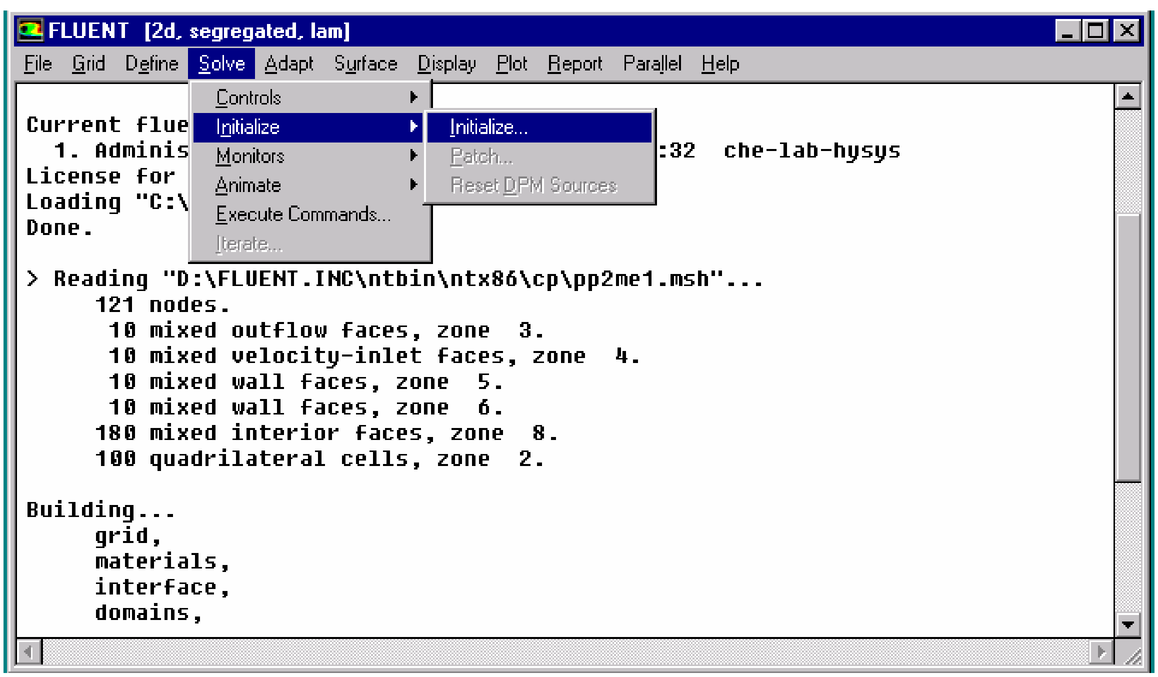
➢ Selecting the wall conditions

Wall boundary conditions are used to bound fluid and solid regions. The shear stress and heat transfer between the fluid and wall are computed based on the flow details in the local flow field.

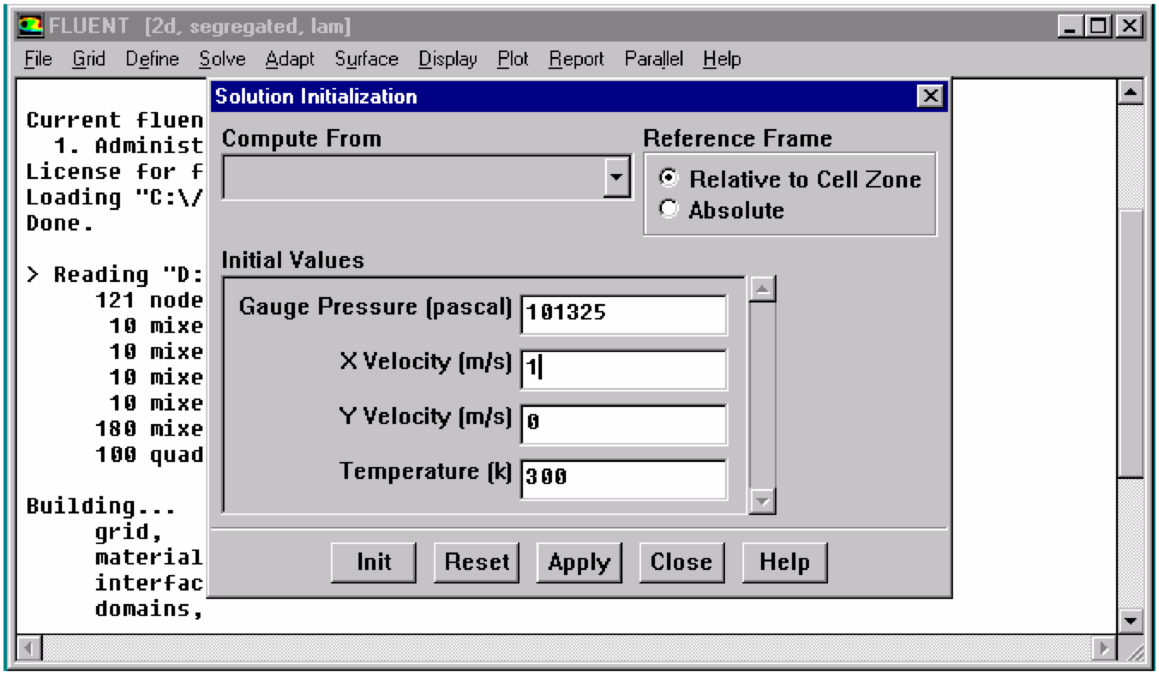


Step 10: Solution Initialization

After entering all the parameters, the problem is initialized as shown below.

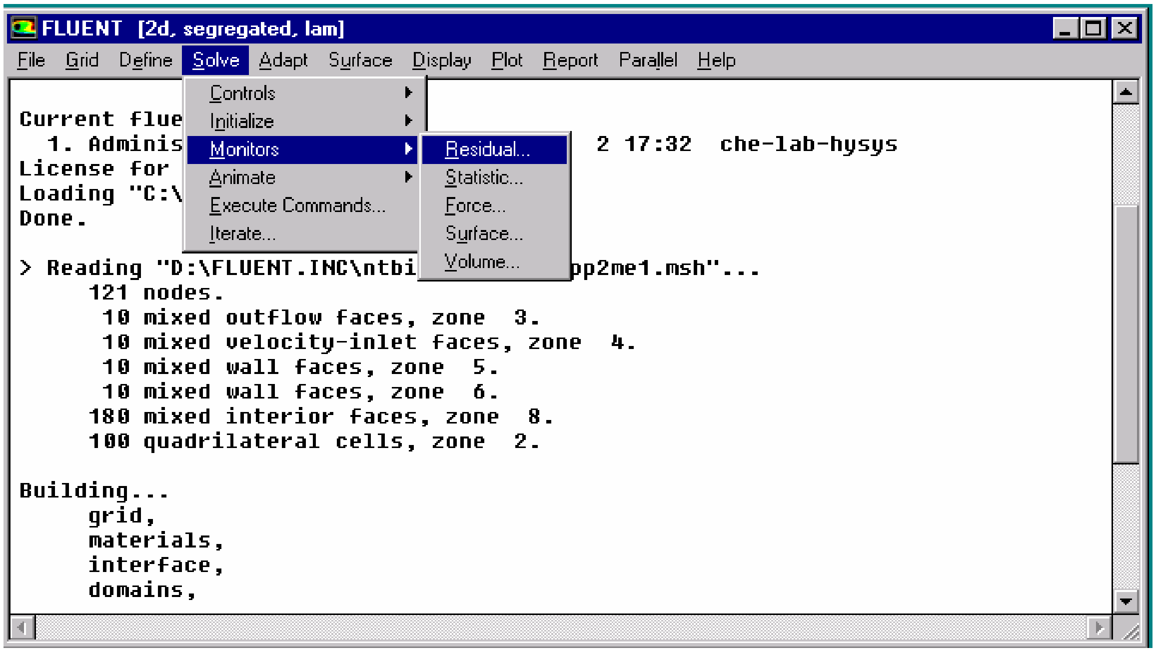


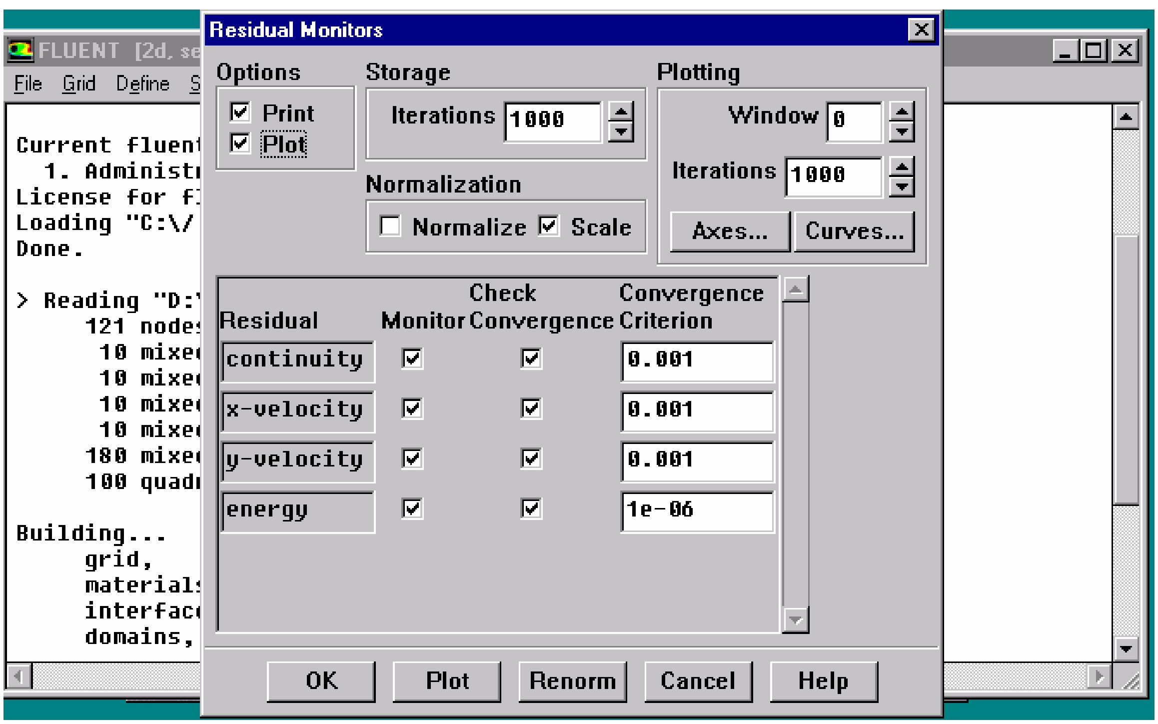
The initial values for the pressure and velocity are entered.



Step 11: Residual Monitoring

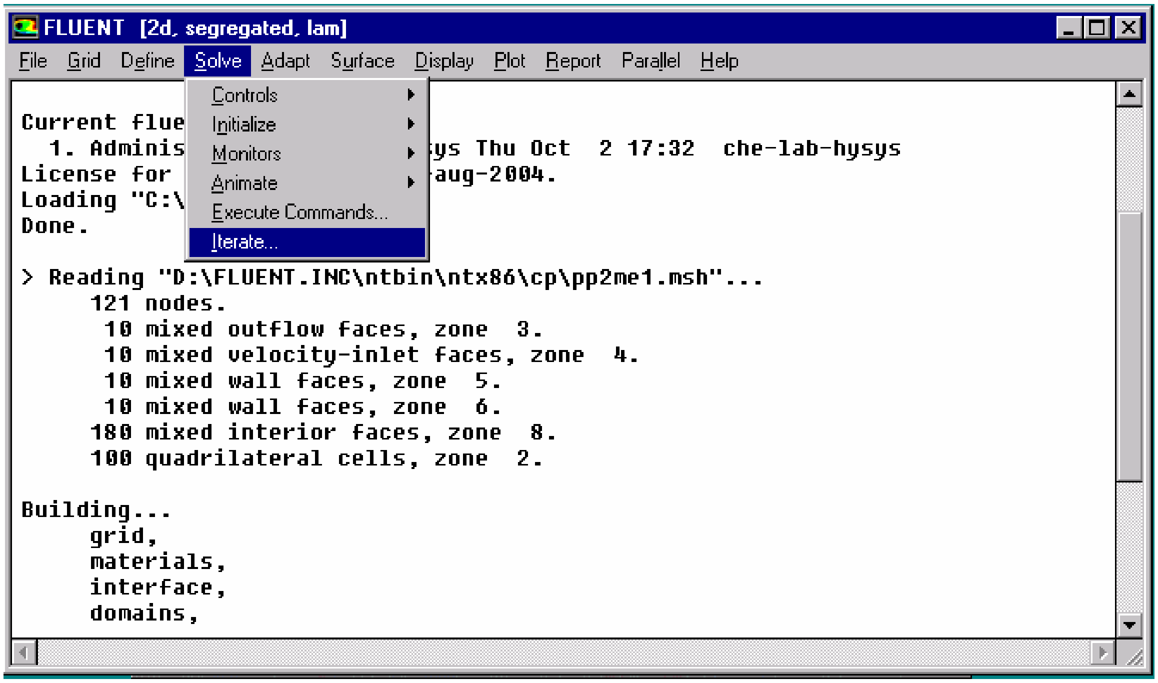
After initializing Residual Monitoring is done to view the convergence plot

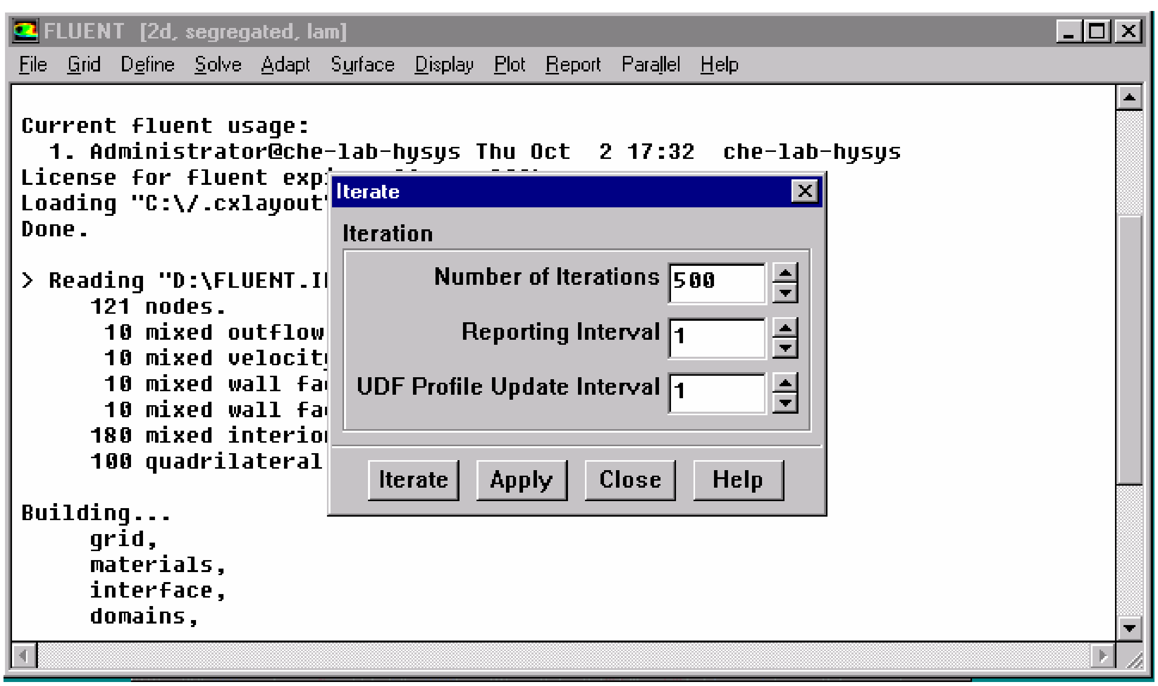

FLUENT [2d, segregated, lam]
File Grid Define Solve Adapt Surface Display Plot Report Parallel Help
Controls
Initialize
Monitors
Animate
Execute Commands...
Iterate...
Residual...
Statistic...
Force...
Surface...
Volume...
Current flue
1. Adminis
License for
Loading "C:\
Done.
2 17:32 che-lab-hysys
> Reading "D:\FLUENT.INC\ntbi
pp2me1.msh"...
121 nodes.
10 mixed outflow faces, zone 3.
10 mixed velocity-inlet faces, zone 4.
10 mixed wall faces, zone 5.
10 mixed wall faces, zone 6.
180 mixed interior faces, zone 8.
100 quadrilateral cells, zone 2.
Building...
grid,
materials,
interface,
domains,


Residual Monitors
Options
Print
Plot
Storage
Iterations 1000
Normalization
Normalize
Scale
Plotting
Window 0
Iterations 1000
Axes...
Curves...
Residual
Monitor
Check Convergence
Convergence Criterion
continuity 0.001
x-velocity 0.001
y-velocity 0.001
energy 1e-06
OK
Plot
Renorm
Cancel
Help

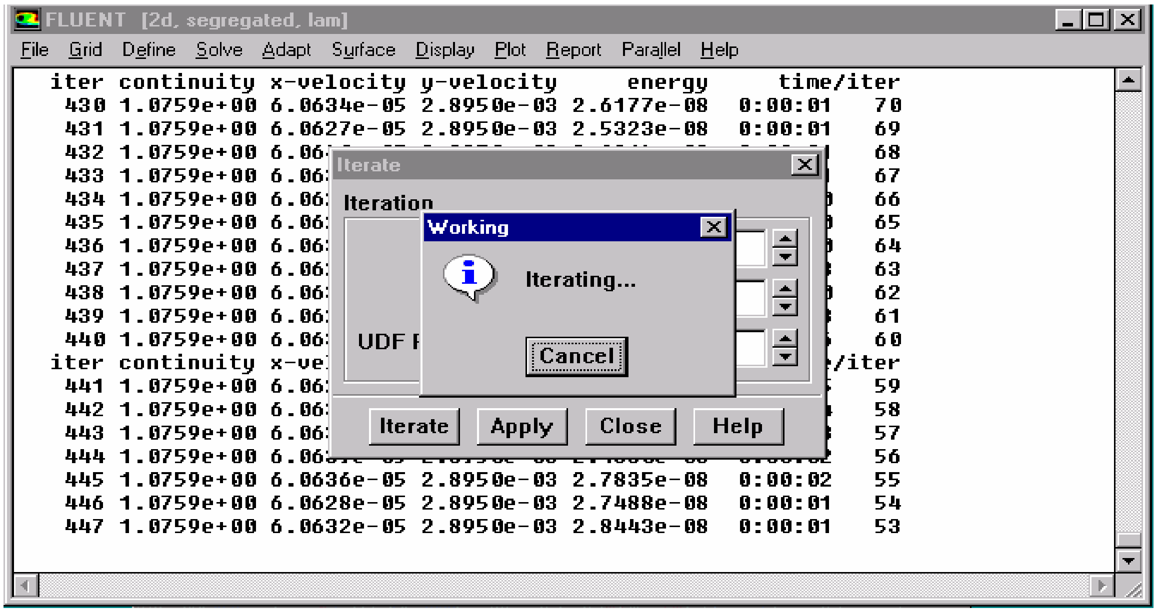
Step 12: Iterate

Iterations are carried out as shown by specifying the number of iterations in Iterate form.





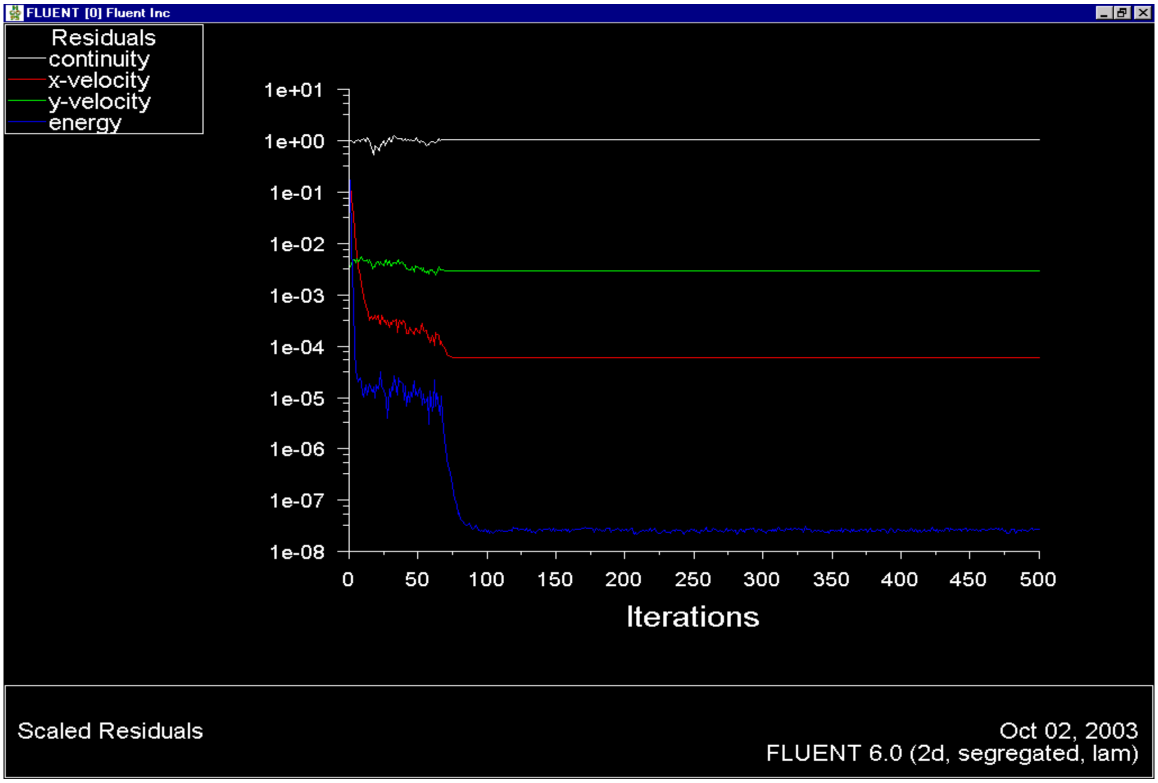
Step 13: Results Obtained



The final iterated values are shown.

iter	continuity	x-velocity	y-velocity	energy	time/iter	
20	7.6475e-01	3.3940e-04	4.2017e-03	1.7491e-05	0:00:00	480
40	1.0601e+00	3.0801e-04	3.8247e-03	8.9988e-06	0:00:23	460
60	9.7681e-01	1.6271e-04	3.0241e-03	5.5699e-06	0:00:18	440
80	1.0759e+00	6.0608e-05	2.8950e-03	5.5318e-08	0:00:13	420
100	1.0759e+00	6.0629e-05	2.8950e-03	2.4148e-08	0:00:10	400
120	1.0759e+00	6.0636e-05	2.8950e-03	2.8806e-08	0:00:12	380
140	1.0759e+00	6.0639e-05	2.8950e-03	2.4742e-08	0:00:09	360
160	1.0759e+00	6.0628e-05	2.8950e-03	2.4204e-08	0:00:07	340
180	1.0759e+00	6.0637e-05	2.8950e-03	2.7697e-08	0:00:05	320
200	1.0759e+00	6.0620e-05	2.8950e-03	2.5786e-08	0:00:07	300
220	1.0759e+00	6.0627e-05	2.8950e-03	2.6415e-08	0:00:05	280
iter	continuity	x-velocity	y-velocity	energy	time/iter	
240	1.0759e+00	6.0642e-05	2.8950e-03	2.5785e-08	0:00:04	260
260	1.0759e+00	6.0627e-05	2.8950e-03	2.4980e-08	0:00:03	240
280	1.0759e+00	6.0633e-05	2.8950e-03	2.5725e-08	0:00:04	220
300	1.0759e+00	6.0632e-05	2.8950e-03	2.4837e-08	0:00:03	200
320	1.0759e+00	6.0636e-05	2.8950e-03	2.8518e-08	0:00:02	180
340	1.0759e+00	6.0625e-05	2.8950e-03	2.6510e-08	0:00:02	160
360	1.0759e+00	6.0637e-05	2.8950e-03	2.7261e-08	0:00:03	140
380	1.0759e+00	6.0636e-05	2.8950e-03	2.5510e-08	0:00:02	120
400	1.0759e+00	6.0644e-05	2.8950e-03	2.5552e-08	0:00:01	100
420	1.0759e+00	6.0639e-05	2.8950e-03	2.6662e-08	0:00:01	80
440	1.0759e+00	6.0631e-05	2.8950e-03	2.6715e-08	0:00:01	60
iter	continuity	x-velocity	y-velocity	energy	time/iter	
460	1.0759e+00	6.0635e-05	2.8950e-03	2.7527e-08	0:00:01	40
480	1.0759e+00	6.0633e-05	2.8950e-03	2.6826e-08	0:00:00	20
500	1.0759e+00	6.0638e-05	2.8950e-03	2.7095e-08	0:00:00	0

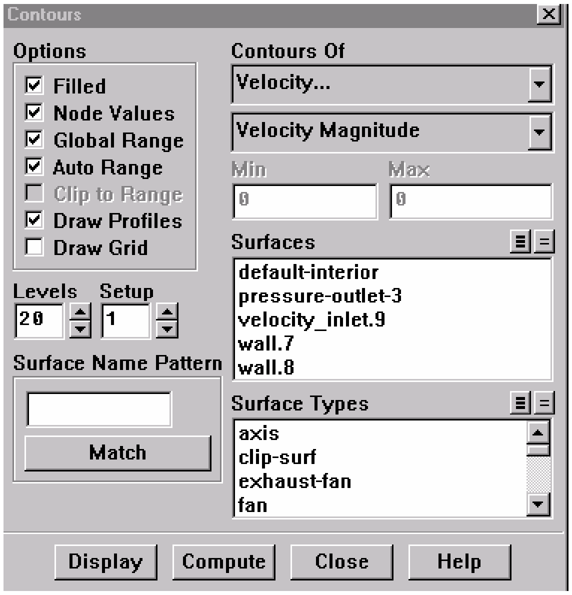
- **Residual Plot**



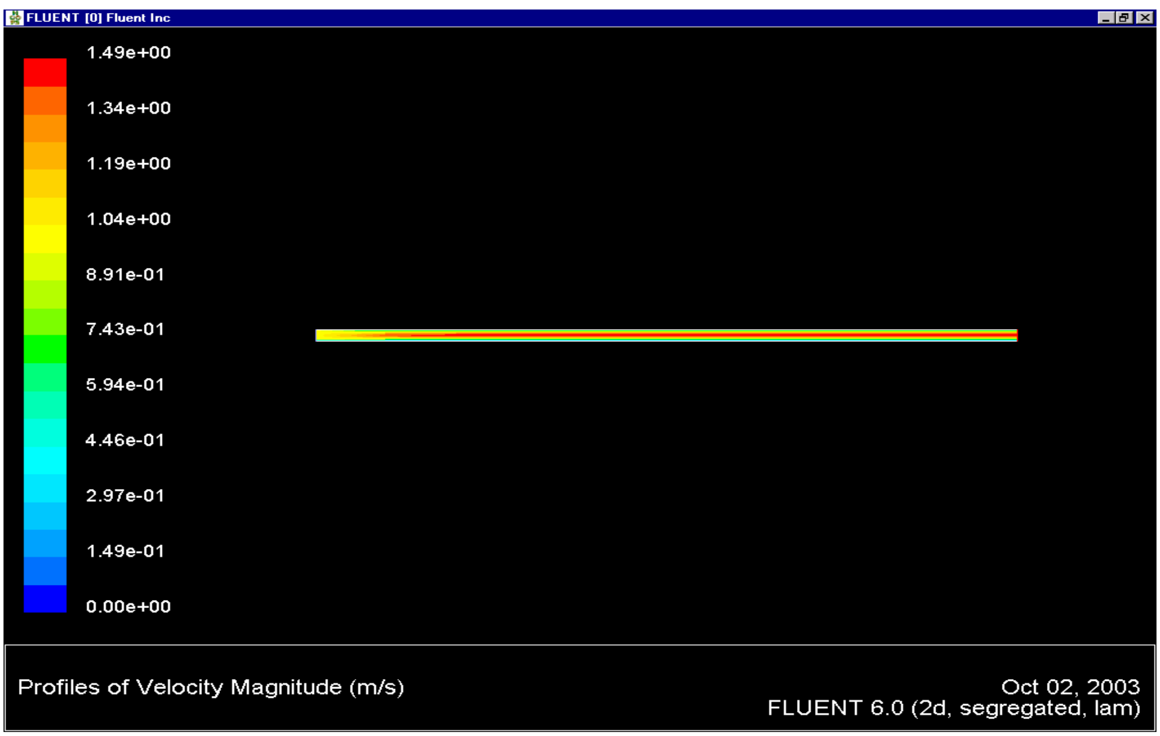
Step 14: Displaying Results Using Contours

➢ **Selecting the Contours**

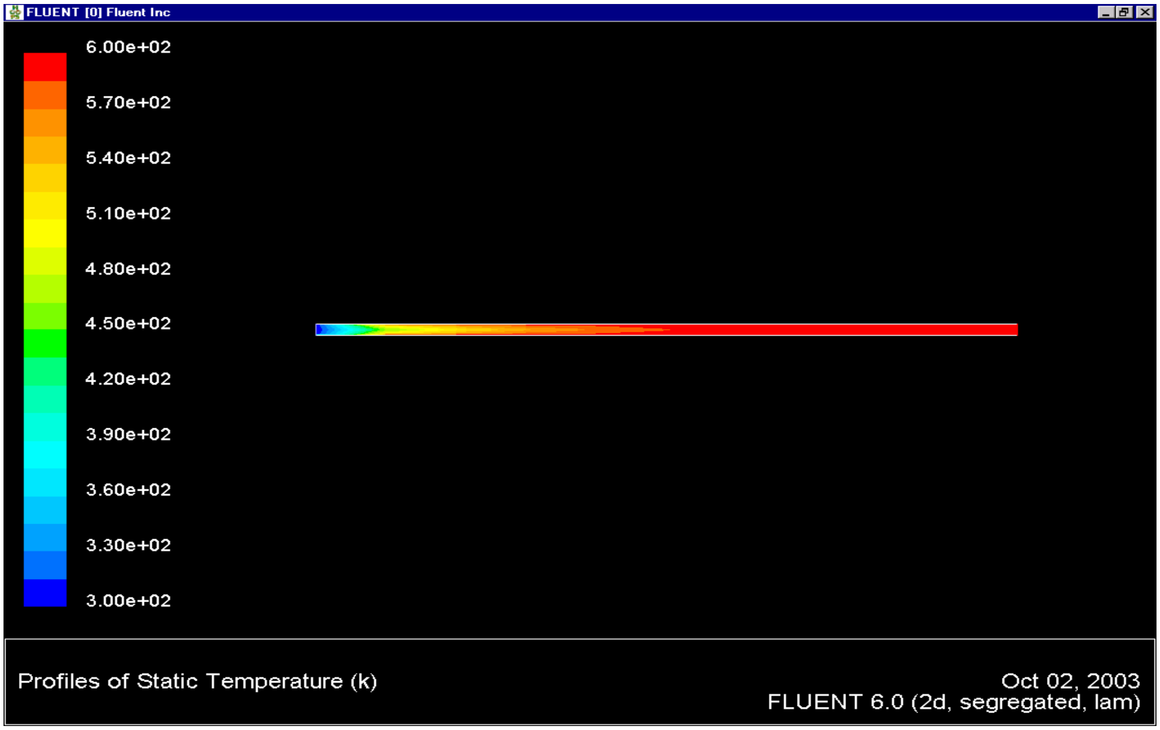
The required contours are selected from the contour form



a) Velocity Contours



b) Temperature Contours



c) Pressure Contours