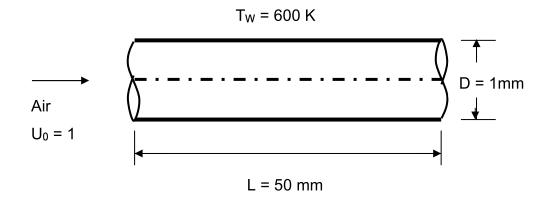
## 15.2.5. Step-wise methodology of FLUENT usage for a problem

### 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.

```
FLUENT [2d, segregated, lam]
                                                                                  - 🗆 ×
File Grid Define Solve Adapt Surface Display
                                     Plot Report Parallel Help
      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. user@che-lab-hysys
                            Fri Oct 3 18:32 che-lab-hysys
License for fluent expires 26-aug-2004.
Loading "C:\/.cxlayout"
Done.
> |
```

#### Step 2: Choosing the Solver Formulation

FLUENT provides three different solver formulations:

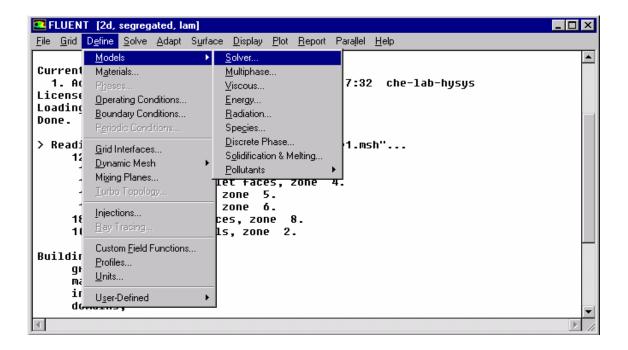
- Segregated
- Coupled implicit
- Coupled explicit

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

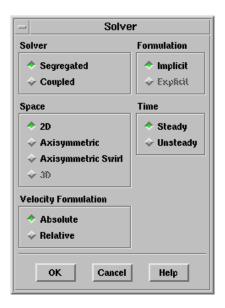
### User Inputs for Solver Selection

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

**Define**  $\rightarrow$  **Models**  $\rightarrow$  **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.

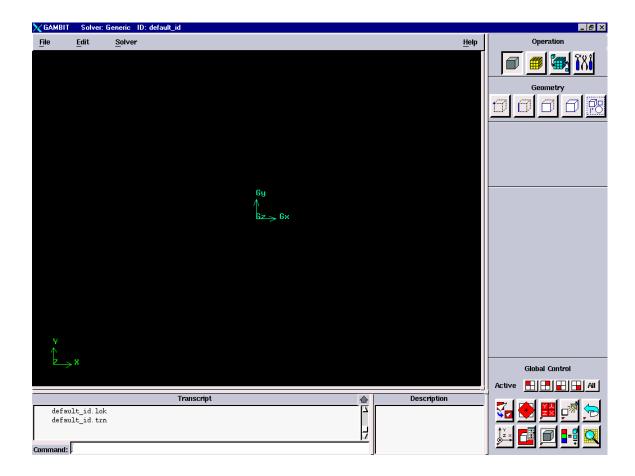
#### Step 3: Reading Mesh Files

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

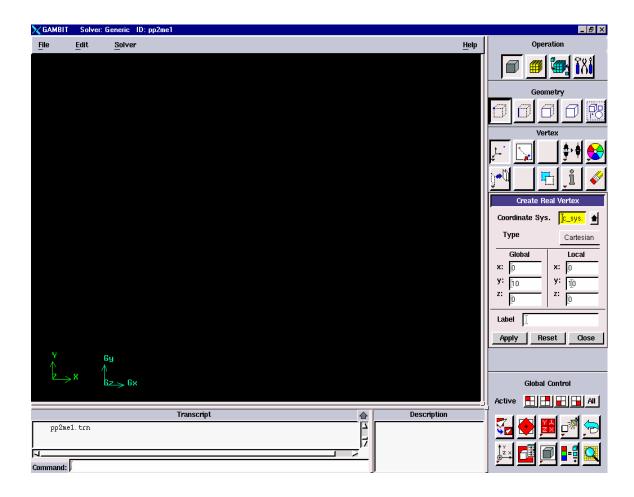
GAMBIT is used to create 2D and 3D structured/unstructured/hybrid grids. GAMBIT is a software package designed to help analysts and designers build and mesh models for computational fluid dynamics (CFD) and other scientific applications. GAMBIT receives user input by means of its graphical user interface (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.

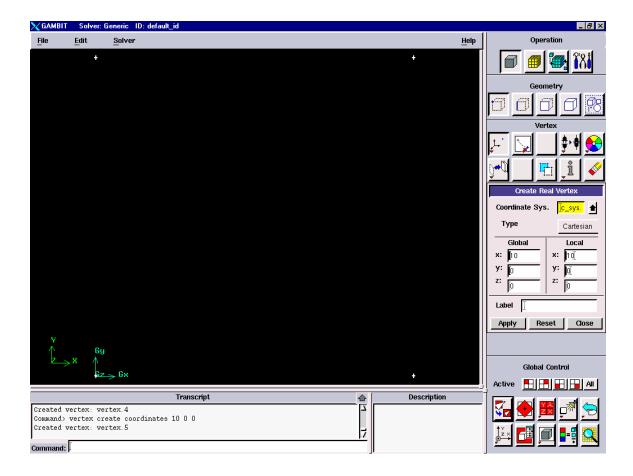


GAMBIT modeling forms require you to specify a location in space relative to a specified coordinate system. For example, the **Create Real Vertex** form shown in Fig. 2 requires you to specify the three coordinates describing the point at which the vertex is to be created.

Create Real Vertex				
Coordinate Sys.	Coordinate Sys. 🔽 sys.1 🔒			
Туре	Cartesian 🗆			
Global	Local			
<b>x:</b> [0	<b>x:</b> 0			
<b>y:</b> [0	<b>y:</b> 0			
<b>z:</b> [0	<b>z:</b> [0			
Label				
Apply Re	set Close			

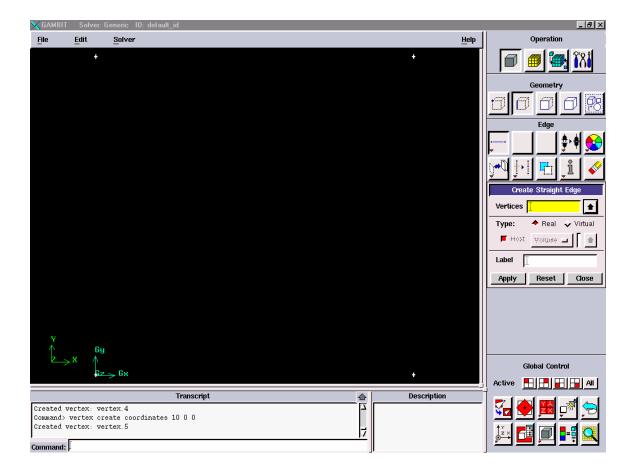
## Fig. 15.8. Creating Real Vertex

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



## **To create Edges**

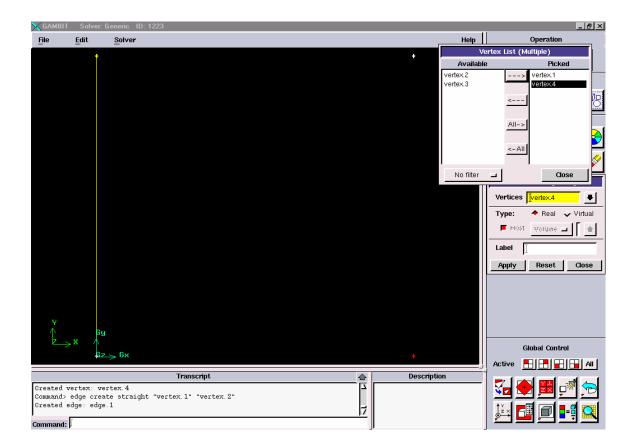
The following commands are available on the Geometry/Edge subpad.



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

Create Straight Edge						
Vertices	Vertices [					
Type:	🔵 Real 🛛 🌔 Virtual					
📕 Host	Volume 🗆 👔					
Label						
Apply	Reset	Close				

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.

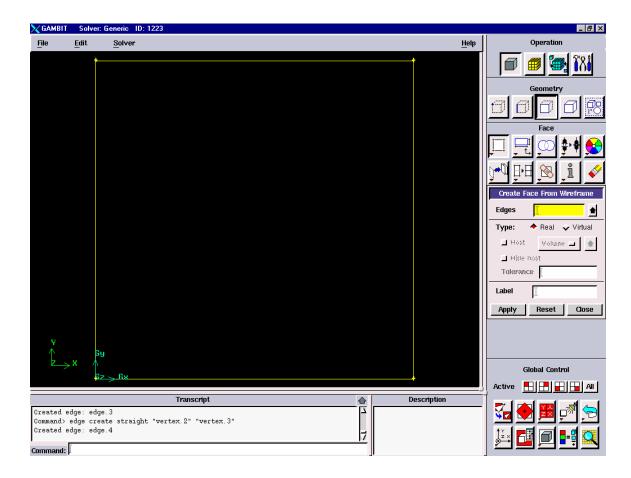


Similarly all edges are created.

🗸 GAMB	BIT Solver	: Generic ID: pp2me1		_ 8 >
File	Edit	Solver	Help	Operation
		+		
				Geometry
				Edge
				<u>; \$ \$ \$ \$ \$</u>
				🚰 <u>† † 🗄 i 🎸</u>
				Create Straight Edge
				Vertices
				Type: ◆ Real ♀ Virtual
				Label
				Apply Reset Close
		By		
¥ A				
e_	⇒×	βz → 6x		Global Control
				Active 🔠 🖪 🖬 🗛 All
Prostor	d edge: edg	Transcript description		🚬 🛋 🔤 🔜 🦕
,reated	u euge: eag	7		
1				🖾 📑 🗖 📑
omman	id:			

## > To create Face

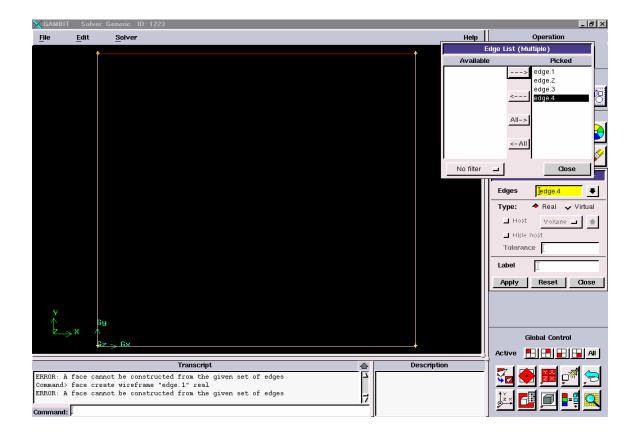
**Create Face From Wireframe** command allows you to create a face from three or more existing edges. The following commands are available on the **Geometry/Face** subpad.



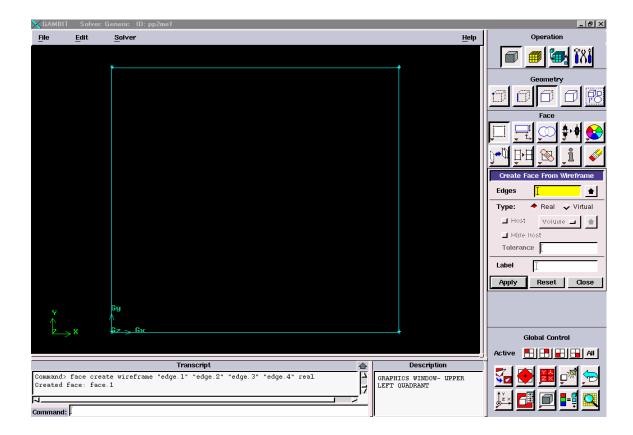
To create a face by means of the **Create Face From Wireframe** option, you must specify the following parameters:

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

The face is created as shown.

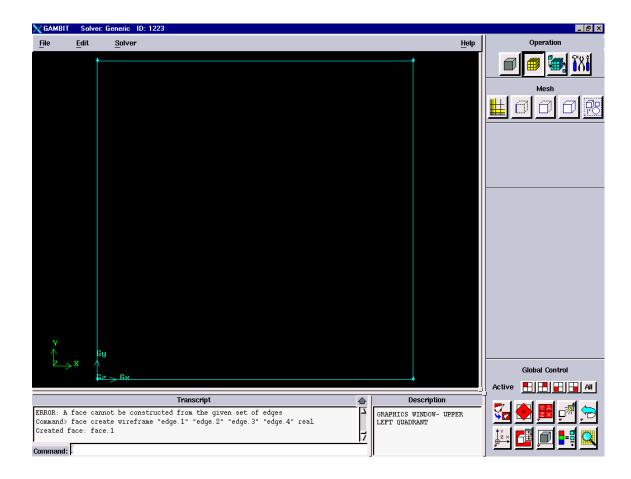


The final face window looks like this.



### 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.



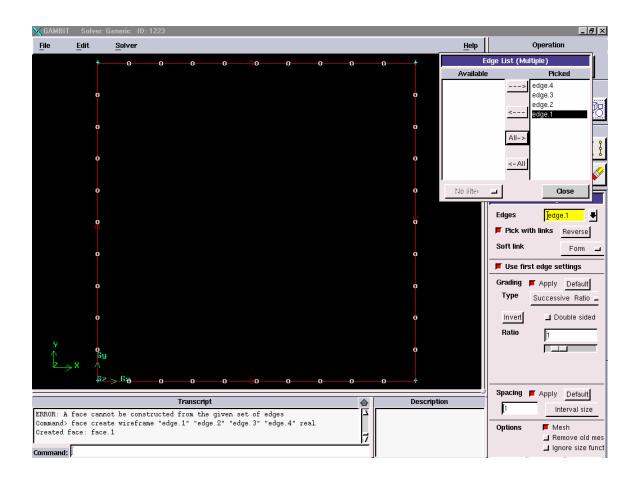
### **Mesh Edges**

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



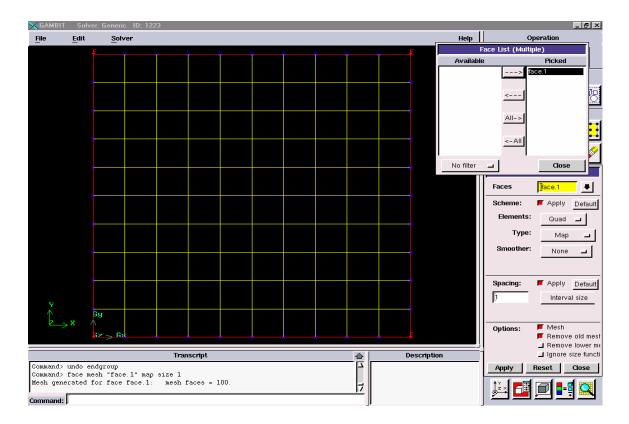
#### **Mesh Faces**

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.

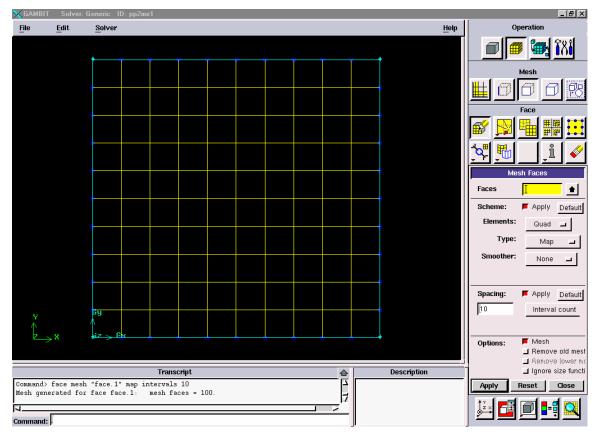
To mesh a face, you must specify the following parameters:

- 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.

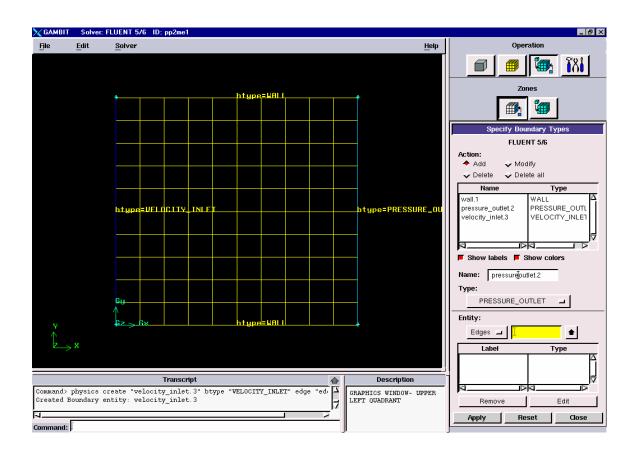


**Specifying Zone Types** 

**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**  $\rightarrow$  **Export**  $\rightarrow$  **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  $\rightarrow$  Read  $\rightarrow$  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  $\rightarrow$  Import  $\rightarrow$  GAMBIT...

E FLUENT [2d,	segregated, lan	1	_ 🗆 🗵
<u>File G</u> rid D <u>e</u> fine	<u>S</u> olve <u>A</u> dapt S	S <u>u</u> rface <u>D</u> isplay <u>P</u> lot <u>R</u> eport Parallel <u>H</u> elp	
<u>R</u> ead ► <u>W</u> rite ►	<u>C</u> ase Data	.0.20	<b></b>
<u>I</u> mport ► E <u>x</u> port	C <u>a</u> se & Data Pd <u>f</u> <u>B</u> ays	it Inc.	
Interpolate	<u>M</u> eys ⊻iew Factors <u>P</u> rofile	.uent6.0\lib\fl_s112.dmp"	
Save Layout	<u>S</u> cheme Journal	ıb-hysys Thu Oct 2 17:32 che-lab-hysys	
Exit Done.	p.msh pns2 cvd	rs 26-aug-2004.	
> <sup>-</sup>			
<b>T</b>			

Selecting the File From the Directory

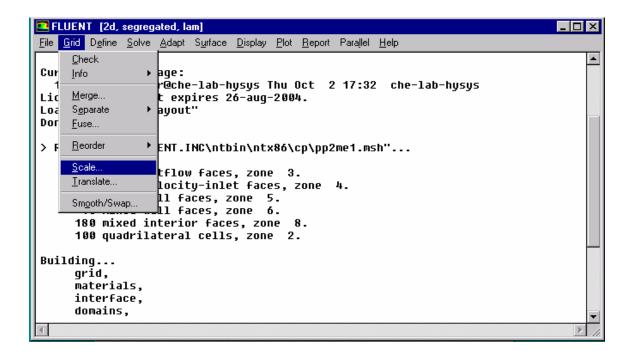
💶 FLUENT 🛛 [2d, se	egregated, lam]	- 🗆 🗵
<u>File G</u> rid D <u>e</u> fine <u>S</u>	olve <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>P</u> lot <u>R</u> eport Para <u>l</u> lel <u>H</u> elp	
Welcome	t Select File	<b></b>
Copyrigh All Righ	t Look jn: 🔄 cp 💽 🖻 📸 📰 📰	
Loading "D:\F Done.		
Current fluen 1. Administ License for f	r l	
Loading "C:\/ Done.		
>	Case File pp2me1 OK	
	Files of type: Case Files Cancel	
4		×

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



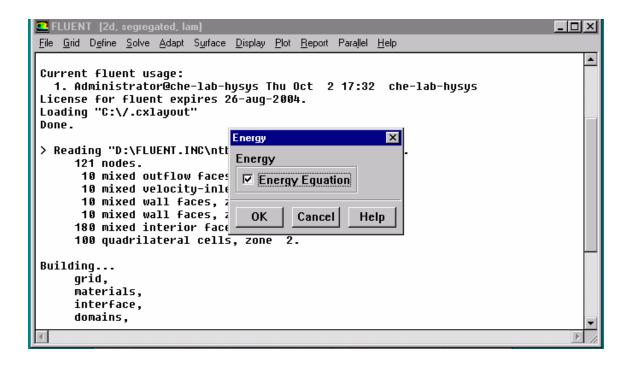
After clicking the Scale, the window appears as shown below. Enter the dimension of the problem using Units Conversion in the Scale Grid form.

FLUENT [2d, segregated, I	am]	_ 🗆 🗵
<u>File Grid Define Solve A</u> dapt	S <u>u</u> rface <u>D</u> isplay <u>P</u> lot <u>R</u> eport Para <u>l</u> lel <u>H</u> elp	
Current fluent usage: 1. Administrator@ct	Scale Grid 🔀	
License for fluent ex Loading "C:\/.cxlayou		
Done.	X 0.001 Grid Was Created In mm 👻	
> Reading "D:\FLUENT. 121 nodes.	Y 0.001 Change Length Units	
10 mixed outflc 10 mixed veloci	Domain Extents	
10 mixed wall f 10 mixed wall f	Xmin (m) 🔋 Xmax (m) 🖲 . 05	
180 mixed interi 180 quadrilatera	Ymin (m) 🕫 Ymax (m) 🖲 . 881	
Building grid, materials,	Scale UnScale Close Help	
interface, domains,		<b>•</b>
<u> </u>		

Step 6: Defining the Model

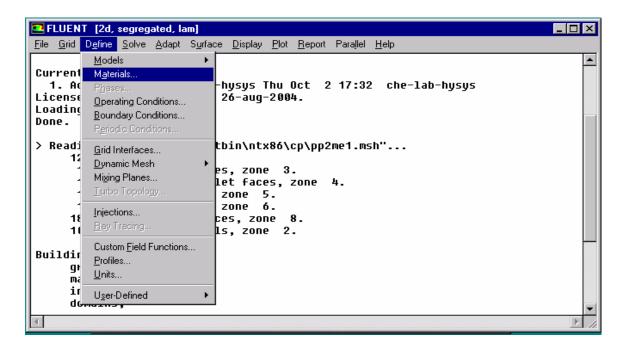
> Selecting the Viscous and Energy model

💶 FLUENT [2d, segregated, lam]	<u>- 🗆 ×</u>
<u>File Grid Define Solve Adapt Surface Display Plot Report Parallel Help</u>	
Current fluent usage: 1. Administrator@che-lab-husus Thu Oct 2 17:32 che-lab-hysys License for fluent expires Loading "C:\/.cxlayout" Done. > Reading "D:\FLUENT.INC\nt 121 nodes. 10 mixed outflow face 10 mixed velocity-inl 10 mixed wall faces, 10 mixed wall faces, 10 mixed interior fac 100 quadrilateral cell Building grid, materials, interface, domains,	



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**  $\rightarrow$  **Materials...** 



The Materials Panel looks as such

- Materials			
Name air	Material Type	Order Materials By	
Chemical Formula	Fluid Materials		
Properties			
Density (kg/m3)	constant 🛛 🝸	Edit	
	1,225		
Cp (j/kg-k)	constant 🗾 🔻	Editor	
	1006,43		
Thermal Conductivity (w/m–k)	constant 🗾 🝸	Edit	
	0.0242		
Viscosity (kg/m–s)		Editor	
	1.7894e-05		
Change/Create	Delete Close	Help	

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

	Б	Materia	Database Materials	×	<b>V</b> I	
			Solid Materials Material Type		×	
	-	Name alum:	dolomite (cao_mgo_2co2) 🔺 solid 🗸			×I
	Ē		gold (au) gypsum (caso4_2h20) Order Materials By			Г
ſ		Chem al	nickel (ni)   Name			
	C		steel  Chemical Formula titanium (ti)			
	L L	Prope	Properties			
	D		Density (kg/m3)	4		
	>					
			8 03 0			
			Cp (j/kg-k) constant View			
			502.48			
		Ther	I			
			Thermal Conductivity (w/m-k) constant View			$\vdash$
	8		16.27			
			Electrical Conductivity (1/ohm-m)			
			constant			
ŀ			8330000			F
				┙	_	
			Copy Close Help			

	Materials	×	
	Name	Material Type Order Materials By	
Ē	air	fluid 🗸 🔍 🖲 Name	<
Ē	Chemical Formula	Fluid Materials	-
C		air ▼ Database	_
lι	Properties		
	Density (kg/m3)	constant ▼ Edit	
,		1.225	
	Cp (j/kg-k)	constant   Edit	
L		1006.43	
L	Thermal Conductivity (w/m-k)	constant 👻 Edit	
8		0.0242	
L	Viscosity (kg/m-s)	constant 💌 Edit	
		1.7894e-05	-
	Change/Create	Delete Close Help	8

Step 8: Selecting the Operating Conditions

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

🖸 FLUENT [2d, segregated, lam]				
<u>F</u> ile <u>G</u> rid	D <u>e</u> fine <u>S</u> olve <u>A</u> dapt S <u>u</u> rfa	ce <u>D</u> isplay <u>P</u> lot <u>R</u> eport Para <u>l</u> lel <u>H</u> elp		
Current 1. Ac License Loadin <u>c</u> Done.	Phases Operating Conditions	-hysys Thu Oct 2 17:32 che-lab-hysys 26-aug-2004.	<b></b>	
> Readi 12 - - - - - - - - - - - - - - - - - -	Grid Interfaces         Dynamic Mesh         Mixing Planes         Turbo Topology         Injections         Bay Tracing	tbin\ntx86\cp\pp2me1.msh" es, zone 3. let faces, zone 4. zone 5. zone 6. ces, zone 8. ls, zone 2.		
Buildir gr mi ir dG	Custom Eield Functions Profiles <u>U</u> nits Uger-Defined			

The Operating Condition panel looks as such

💶 FLUENT 🛛 [2d, segregated, la	im]		
<u>File G</u> rid D <u>e</u> fine <u>S</u> olve <u>A</u> dapt	S <u>u</u> rface <u>D</u> isplay <u>P</u> lot	<u>R</u> eport Para <u>l</u> lel	Help
Current fluent usage: 1. Administrator@ch License for fluent ex Loading "C:\/.cxlayou Done. > Reading "D:\FLUENT. 121 nodes. 10 mixed outflo 10 mixed veloci 10 mixed wall f 180 mixed interi 100 quadrilatera	Operating Conditions Pressure Operating Press 101325 Reference Press X (m) 0 Y (m) 0	sure (pascal)	Gravity Gravity
Building grid, materials, interface, domains,	ОК	Cancel Ho	elp
<u>र</u>			F /

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**  $\rightarrow$  **Boundary Conditions...** 

💶 FLUENT [2d, segregated, lam]	_ 🗆 🗡
<u>File Grid Define</u> Solve Adapt Surface Display Plot Report Parallel Help	
Models       Materials         1. Ac       Phases         License       Operating Conditions         Done.       Boundary Conditions	
<pre>&gt; Readi Grid Interfaces 12 Dynamic Mesh Miging Planes 14 Jurbo Topology 18 Injections 14 Injections 14 Eay Tracing 15, zone 2.</pre>	
Buildir Gr Gr User-Defined Gr Gr Gr Gr Gr Gr Gr Gr Gr Gr	<b>_</b>

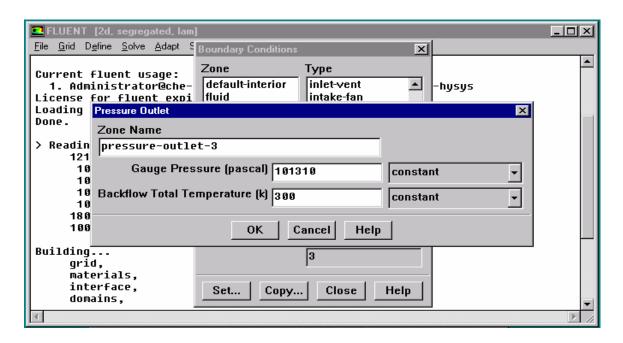
## Selecting Outflow Conditions

Eile     Grid     Define     Solve     Adapt     S       Boundary Conditions     Image: Solve     Zone     Type	
Current fluent usage: 1. Administrator@che- License for fluent expi Loading "C:\/.cxlayout" Done. > Reading "D:\FLUENT.IN 121 nodes. 10 mixed outflow 10 mixed velocity 10 mixed wall fac 180 mixed interior 190 quadrilateral	•
Building grid, materials, interface, domains, ID 3 Close Help	

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.

FLUENT	[2d, segregated, lam]	_ O ×
<u>File G</u> rid D <u>e</u>	efine <u>S</u> olve Adapt <sup>S</sup> Boundary Conditions	
Current f	Tuent usage: Zone Type	<b></b>
	Velocity Inlet	
License · Loading '	Zone Name	
Done.	velocity_inlet.9	
> Reading	Velocity Specification Method Components	
10	Reference Frame Absolute	
10 10	X-Velocity (m/s) 1 constant	
180 100	Y-Velocity (m/s) g	
Building	Temperature (k) 300 constant	
gri mati inti		
domä	ains,	-
4		▶ //

# > 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.

Wall		×
Zone Name		
wall.7		
Adjacent Cell Zone		
fluid		
Thermal DPM	Momentum Species Radiation	UDS
Thermal Condition		
C Heat Flux	Temperatur	e (k) 600 constant 👻
• Temperature	Wall Thickness	
C Convection C Radiation		
O Mixed	Heat Generation Rate (w)	(m3) <mark>0</mark>
Material Name		
steel	Edit	
	ОК Саг	ncel Help

Step 10: Solution Initialization

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

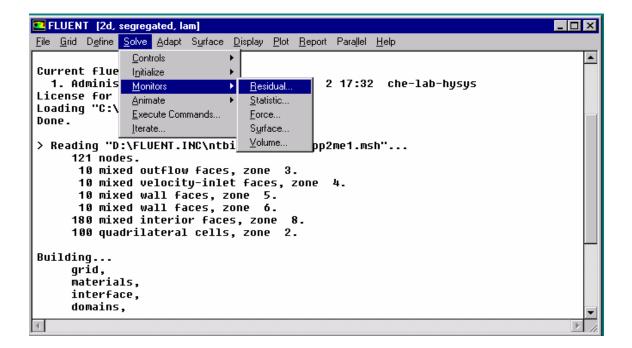
E FLUENT (2d	, segregated, lam]	-	- 🗆 ×
<u>File G</u> rid D <u>e</u> fine	: <u>S</u> olve Adapt S <u>u</u> rface <u>D</u> isplay <u>P</u> lot <u>R</u>	eport Parallel <u>H</u> elp	
	Controls		<b></b>
Current flu			
1. Admini  License for		:32 che-lab-hysys	
Loading "C:	Animate Preset DHM 5	ources	
Done.	Execute Commands		
	Iterate		
	D:\FLUENT.INC\ntbin\ntx86\cp	\pp2me1.msh"	
121 no 10 mi	ues. xed outflow faces, zone  3.		
	xed velocity-inlet faces, zo	ne 4.	
10 mi	xed wall faces, zone 5.		
	xed wall faces, zone 6.		
	xed interior faces, zone 8. adrilateral cells, zone 2.		
100 40			
Building			
grid,	_		
materi interf			
domain			
	- ;		
<u>تا</u>			

The initial values for the pressure and velocity are entered.

💶 FLUENT 🛛 [2d, se	egregated, lam]	<u>- 0 ×</u>
<u>File G</u> rid D <u>e</u> fine <u>S</u>	olve <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>P</u> lot <u>R</u> eport Parallel <u>H</u> elp	
	Solution Initialization	<b></b>
Current fluen 1. Administ	Compute From Reference Frame	
License for f Loading "C:\/ Done.	Relative to Cell Zone     Absolute	
> Reading "D:	Initial Values	
121 node 10 mixe	Gauge Pressure (pascal) 101325	
10 mixe 10 mixe	X Velocity (m/s) 1	
10 mixe 180 mixe	Y Velocity (m/s) 0	
100 quad	Temperature (k) 300	
Building qrid,		
material interfac	Init Reset Apply Close Help	
domains,		•
4		

## Step 11: Residual Monitoring

After initializing Residual Monitoring is done to view the convergence plot



	Residual Monitors 🛛 🗙					
Elle <u>Grid Define S</u> <u>File Grid Define S</u> Current fluent 1. Administr License for f: Loading "C:\/ Done.	Print Plot	Storage Iterations   Normalizatio 「Normaliz	<b>_</b>	Plotting Window 0 Iterations 1000 Axes Curves	4 4 4 1 1 1 1 1 1 1 1 1 1 1 1 1	
> Reading "D:' 121 node: 10 mixeo 10 mixeo 10 mixeo 10 mixeo 180 mixeo 100 quado	continuity x-velocity y-velocity	য	vergence Cri Ø Ø Ø Ø	. 991		
Building grid, material: interfac: domains,	energy OK	Plot	Renorm	Cancel Help		

# Step 12: Iterate

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

FLUENT [2d, segregated, lam]	
<u>File Grid De</u> fine <u>Solve</u> Adapt Surface <u>D</u> isplay <u>P</u> lot <u>R</u> eport Parallel <u>H</u> elp	
Current flue 1. Adminis License for Loading "C:\ Done.	
Building grid, materials, interface, domains,	•

💶 FLUENT [2d, segregated, lan	1]	- D ×
<u>File G</u> rid D <u>e</u> fine <u>S</u> olve <u>A</u> dapt	S <u>u</u> rface <u>D</u> isplay <u>P</u> lot <u>R</u> eport Para <u>l</u> lel <u>H</u> elp	
Current fluent usage: 1. Administrator@che- License for fluent exp Loading "C:\/.cxlayout Done. > Reading "D:\FLUENT.II 121 nodes. 10 mixed outflow 10 mixed velocity 10 mixed wall fai 10 mixed wall fai 180 mixed interior 100 quadrilateral Building grid, materials, interface, domains,	Lab-hysys Thu Oct 2 17:32 che-lab-hysys Iterate Iteration Number of Iterations 500 Reporting Interval 1 UDF Profile Update Interval 1 Iterate Apply Close Help	
		▼ ▶ //

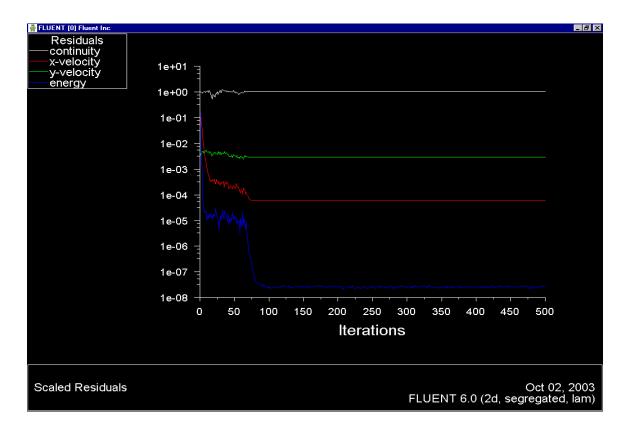
Step 13: Results Obtained

FLUENT [2d, segregated, lam]	- 🗆 🗵
<u>File Grid Define S</u> olve <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>P</u> lot <u>R</u> eport Para <u>l</u> lel <u>H</u> elp	
iter continuity x-velocity y-velocity energy time/iter         430 1.0759e+00 6.0634e-05 2.8950e-03 2.6177e-08 0:00:01 70         431 1.0759e+00 6.0627e-05 2.8950e-03 2.5323e-08 0:00:01 69         432 1.0759e+00 6.066         433 1.0759e+00 6.066         435 1.0759e+00 6.066         436 1.0759e+00 6.066         437 1.0759e+00 6.066         438 1.0759e+00 6.066	A
439       1.6759e+00       6.06;       0         440       1.6759e+00       6.06;       0         iter       continuity       x-ve:       441         441       1.0759e+00       6.06;       0         442       1.0759e+00       6.06;       0         443       1.0759e+00       6.06;       0         444       1.0759e+00       6.06;       0         443       1.0759e+00       6.06;       0         444       1.0759e+00       6.06;       0         445       1.0759e+00       6.06;       0         445       1.0759e+00       6.06;       0         446       1.0759e+00       6.06;36e-05       2.8950e-03       2.7835e-08       0:00:02       55         446       1.0759e+00       6.06;32e-05       2.8950e-03       2.7488e-08       0:00:01       54         447       1.0759e+00       6.06;32e-05       2.8950e-03       2.8443e-08       0:00:01       53	
4	•

The final iterated values are shown.

iter	continuity	x-velocity	y-velocity	energy	time/i	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/i	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/i	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



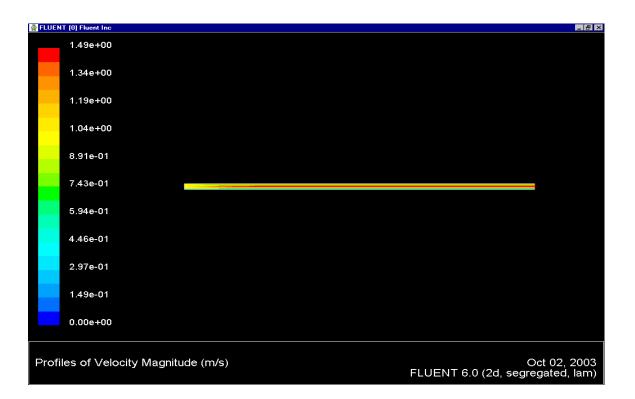
## Step 14: Displaying Results Using Contours

# Selecting the Contours

The required contours are selected from the contour form

Contours			×
Options	Contours Of		
<ul> <li>✓ Filled</li> <li>✓ Node Values</li> <li>✓ Global Range</li> </ul>	Velocity	-	
	Velocity Magni	tude 🚽	-
🗹 Auto Range	Min	Max	
🗖 Clip to Range	9	0	1
Draw Profiles		-	
Draw Grid	Surfaces	<u>i</u> l	
	default-interior pressure-outle velocity_inlet.9 wall.7 wall.8	t-3	
	Surface Types		=
Match	axis clip-surf		]
L]	exhaust-fan fan	•	1
Display Com	pute Close	Help	

a) Velocity Contours



## b) Temperature Contours

🙀 FLUEN	T [0] Fluent Inc	
	6.00e+02	
	5.70e+02	
	5.40e+02	
	5.10e+02	
	4.80e+02	
	4.50e+02	
	4.20e+02	
	3.90e+02	
	3.60e+02	
	3.30e+02	
	3.00e+02	
Profiles of Static Temperature (k)		Oct 02, 2003 FLUENT 6.0 (2d, segregated, lam)

### c) Pressure Contours

