

**INSTED®**

**CFD Test Problems**



TTC Technologies, Inc.  
Centereach, NY 11720 USA

**© 1993-2013 TTC Technologies**

## Contents

Test Problem 1 Heat Conduction with Film Coefficient Boundary Conditions	2
Test Problem 2 Heat Conduction with Volumetric Heat Source	8
Test Problem 3 Axisymmetric Conduction in a Cylinder	13
Test Problem 4 Pressure Distribution in Viscous Flow of a Lubricant Bearing	18
Test Problem 5 Fluid Squeezed Between Parallel Plates	24
Test Problem 6 Natural Convection of Air in a Square Cavity: A Bench Mark Numerical Solution	29
Test Problem 7 Natural Convection Transport of Temperature and Two Non-Reacting Species	36

**TEST PROBLEM 1**  
**HEAT CONDUCTION WITH FILM COEFFICIENT BOUNDARY CONDITIONS**

**Problem Statement**

Analyze the heat conduction between concentric cylinders as shown in Figure 1-1. The wall of the inner cylinder is kept fixed at 140°C while that of the outer cylinder is exposed to an ambient at  $T_\infty = 20^\circ\text{C}$  and the heat transfer coefficient is  $h = 1.5 \text{ W/cm}^2 \cdot ^\circ\text{C}$ . The thermal conductivity,  $k$ , of the material should be taken as  $2 \text{ W/cm} \cdot ^\circ\text{C}$  in all directions.

**Governing Equations**

This problem is governed by the steady state heat conduction equation:

$$k\nabla^2 T + Q = 0,$$

where  $k$  is the thermal conductivity,  $T$  is temperature and  $Q$  is the volumetric heat generation rate.  $Q$  is assumed to be zero for this problem.

**CFD Type**

This is a heat conduction problem. Therefore, the values of the  $u$  and  $v$  velocities should be kept fixed during the solution stage. There are also no scalars to solve for this problem. This problem will be solved in dimensional form. That is, Type 3 solution method will be employed in the Solver (refer to the Users manual). We shall also take advantage of the spatial symmetry of the problem. Consequently, our computational model will be as shown below:

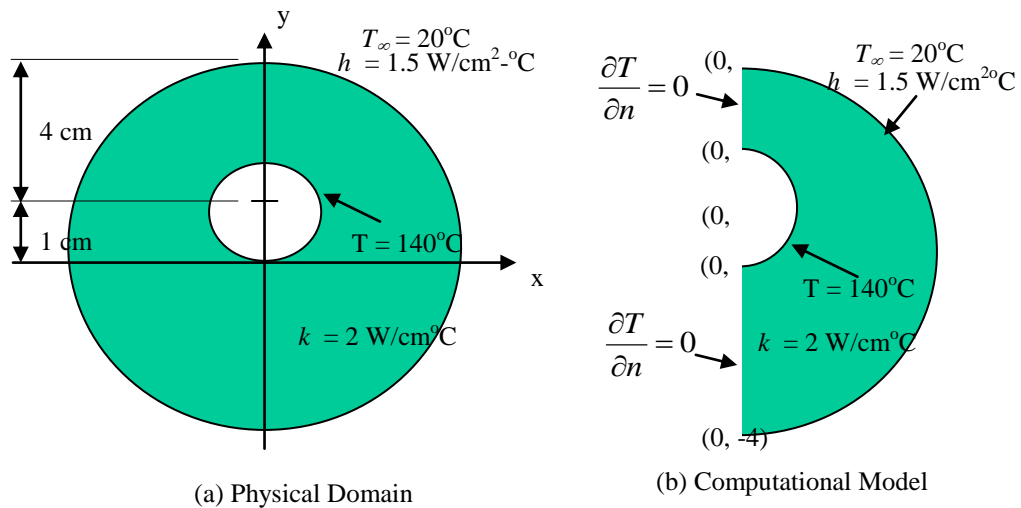


Figure 1.1 Physical and Computational Domain for Sample Problem 1.

## Initial Conditions

This problem will be analyzed as a transient one using the default initial conditions in INSTED. The default initial conditions will be zero or the minimum Dirichlet boundary conditions. If there are no Dirichlet conditions for a variable, INSTED sets the initial condition to zero for that variable at every nodal point.

## Boundary Conditions

Zero temperature gradient or

$$\frac{\partial T}{\partial n} = 0$$

is specified on the symmetry boundary at  $x = 0$ . Temperature is specified as 140 °C on the surface of the inner cylinder. A film coefficient boundary condition is used on the surface of the outer cylinder. This boundary condition can be written as

$$-k \frac{\partial T}{\partial n} = h(T - T_{\infty}) = q''$$

However, INSTED expects the value of  $k\partial T/\partial n$  to be specified as opposed to  $-k\partial T/\partial n$ . Thus,

$$k \frac{\partial T}{\partial n} = -h(T - T_{\infty}) = -q''$$

Thus, the 4 heat flux parameters as discussed in Chapter 3 of the User Manual are:

$$P_1 = -h = -1.5 \text{ W/cm}^2 \text{ } ^\circ\text{C}$$

$$P_2 = 1$$

$$P_3 = 20^\circ\text{C}$$

$$P_4 = 1$$

## Computational Grid

This problem will be analyzed with an unstructured mesh consisting of 113 9-node elements and 503 nodal points, including those at the centroids of the elements.

## Project Files

The files required to reproduce the results reported here for this sample problem are:

Seger1.mes	Input data for use in Mesh Generator
Seger.cfd	Input data for CFD solver
Htflxmat.dat	Film coefficient boundary condition data

## Location of Project Files

The project files above are located in the directory `\insted30\cfd\samples`.

## Model Building and Mesh Generation

Refer to Section 2.3 of the User's Manual for description of the INSTED/CFD (2D) Mesh Generation procedure. The status bar, Input Box, modeling tools, Main dialog box etc., are terms that are described in the manuals.

The boundary of the model for this sample problem will be composed of two arcs and two lines. These lines must form a counterclockwise loop in order to obtain the mesh of the area enclosed by the loop. The steps for creating the mesh for the model are as follows:

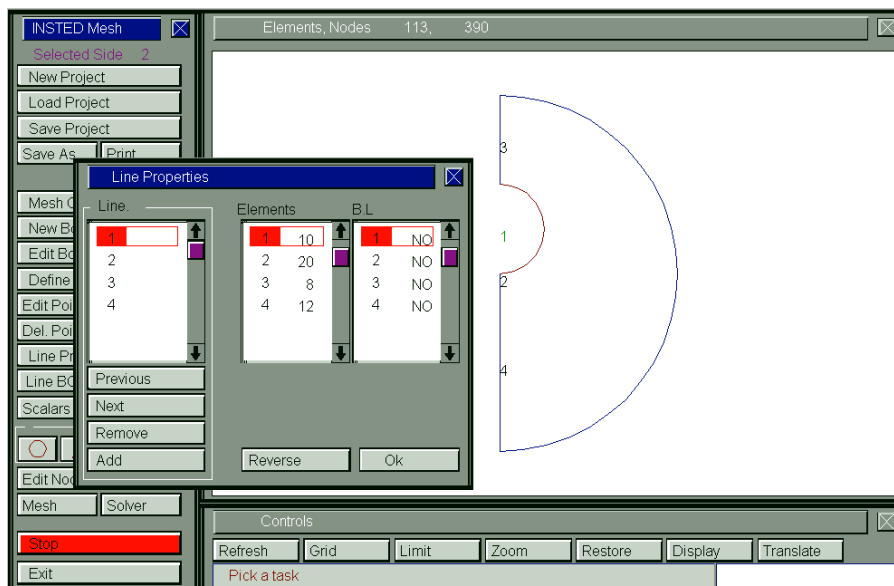
1. Start INSTED/CFD (2D) Mesh Generator.
2. Click the **New Project** button on the Main dialog box.
3. Click the **Limit** button on the 'Controls' dialog box.  
→ The 'Limits' dialog box appears.
4. Type in -6.5, 9.0, -5.0, 5.0 for the lower x, higher x, lower y and higher y screen limits in the 'Limits' dialog box.
5. Click the OK button to dismiss the 'Limits' dialog box.

### To draw the two arc segments

6. Click the Arc icon from the Modeling Tools that reside on the Main dialog box.  
→ Status bar displays the message: 'Pick Arc Center'.
7. Type in 0,1 in the Input Box and press Enter.  
→ Status bar displays the message: 'Pick Arc Start Point'.
8. Type in 0,0 in the Input Box and press Enter.  
→ Status bar displays the message: 'Pick Arc End Point'.
9. Type in 0,2 in the Input box and press Enter.
10. Repeat steps 5 to 8 using (0,0) for the second arc center and (0,-4), (0,4) for the start and end points respectively.

### To draw the two connecting lines

11. Click the Line icon from the Modeling Tools that reside on the Main dialog box.  
→ Status bar displays the message: 'Pick 1<sup>st</sup> Point'.
12. Type in 0,4 in the Input Box and press Enter.  
→ Status bar displays the message: 'Pick 2<sup>nd</sup> Point'
13. Type in 0,2 in the Input Box and press Enter.
14. Repeat Steps 10 to 12 using (0,0) and (0,-4) for the first and second points of the second line.



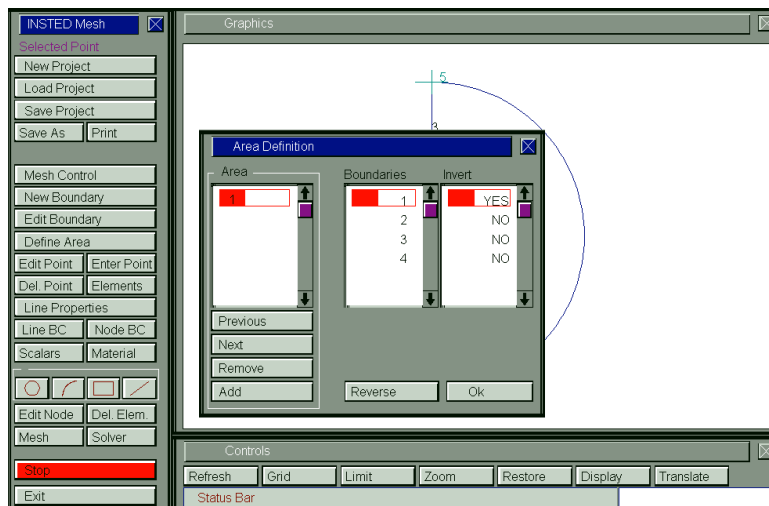
→ Notice that four lines have now been created up to this point. Line 1 is the smaller arc, while Line 2 is the bigger arc, Line 3 is the straight line closer to the top of the screen while Line 4 is the second straight line. The lines are labelled in the display on the 'Graphics' dialog box.

#### Defining number of elements per line

15. Click the **Line Properties** button on the Main dialog box,  
→ The 'Line Properties' dialog box appears.
16. Enter the values of 10, 20, 8, and 12 for the number of elements for lines 1, 2, 3, and 4 respectively as shown in the figure above. Press Enter after typing each value to move to the next line.
17. Click the OK button to dismiss this dialog box.

#### Defining the area to mesh

18. Click the **Edit Boundary** button on the Main dialog box,  
→ The 'Edit Boundary' dialog box appears. The dialog box indicates that four boundaries have been created as shown in the 'Bdry' listbox.
19. Scroll through the boundaries created to find out the constituent lines by clicking the **Next** button.
20. Exit the dialog box by clicking the **OK** button.
21. Click the **Define Area** button on the Main dialog box,  
→ The 'Area Definition' dialog box appears. This is shown in the figure below except that the first boundary is not yet inverted..
22. Click the **Add** button on this dialog box.  
→ An area listed as '1' is added in the 'Area' listbox.
23. Click the first line of the 'Boundaries' listbox.
24. Type 1 and press Enter. This is meant to include boundary (made of Arc line 1) in this area,
25. Repeat Step 20 to include boundaries 2, 3, and 4 in the definition of the current area.
26. Click the first line of the 'Invert' listbox to reverse the direction of Arc Line 1 (You may have realized from the order in which you entered coordinates (in Steps 6 through 14) that the arc represented by line 1 has a clockwise orientation relative to the loop.) Your dialog box should now be similar to the figure below.
27. Click the **OK** button to dismiss the 'Area Definition' dialog box.



### Applying boundary conditions

28. Click the **Line BC** button on the Main dialog box,  
→ Status bar displays message: 'Pick Side'.
29. Click the Line 1 on the graphics screen or type 1 and Press Enter in the Input Box  
→ The 'Line BC' dialog box appears with the heading -"Boundary Condition, Side 1" (confirm from the title of the dialog box that data is being received for Line 1. This will not be the case if you accidentally selected some other line.).
30. Click the arrow for temperature and select the Dirichlet condition from the list.  
→ An input box appears beside the boundary condition listbox once Dirichlet condition is selected.
31. Enter a value of 140 in the input box.
32. Click the **OK** button to dismiss the dialog box.
33. Repeat this procedure for Line 2 but only click the Nusselt No. condition for this line, leaving the remaining variables intact.

### Mesh and Confirm Mesh Results

34. Click the **Mesh** button on the Main dialog box,  
→ A mesh of the internal of our loop of lines is displayed on the 'Graphics' dialog box.
35. Click the **Display** button on the 'Controls' dialog box.
36. From the listbox, select 'Nusselt No.'
37. Click the **Ok** button on this dialog box to view nodes with Nusselt number boundary conditions.
38. Click the **Refresh** button twice on the 'Controls' dialog box to return the screen to its original condition.

**Note:** You should have the same result as if you loaded the file 'seger1.mes' located in the directory \insted30\cfd\samples\seger1.mes. You may also choose to save your current work. Refer to the sections 'Loading a file' and 'Saving a file' in the first chapter of the Users Manual.

### **Obtaining a Solution to the Problem**

39. Click the **Solver** button on the INSTED/CFD (2D) Mesh Generator Main dialog box.  
→ The INSTED/CFD (2D)Solver program is launched. The mesh generated from the Mesh Generation program is automatically loaded.
40. Click the **Load Project** button on the Main dialog box of the Solver program.
41. Type \*.\* in the filename field of the operating system 'Open File' dialog box that appears, and press Enter.  
→ A list of files in the current directory is displayed.
42. Locate the file 'Seger.cfd' from the 'Samples' directory.
43. Click this file and press Enter.
44. Click the **Prob. Description** button on the Main dialog box and inspect the loaded parameters.
45. Click the OK button to dismiss the 'Prob. Description' dialog box.
46. Click the **Material Properties** button on the Main dialog box and inspect the loaded material properties. Note particularly the heat flux parameters.
47. Click the OK button to dismiss the 'Material Properties' dialog box.
48. Click the Solve button on the Main dialog box  
→ Program begins to generate a solution to the problem. The graphic screen is converted to one that displays the status of the solution process.
49. Wait for the solution to complete.

## Analyzing the Result

(Refer to the User's Manual for general details about INSTED/CFD (2D) Post-Processing.)

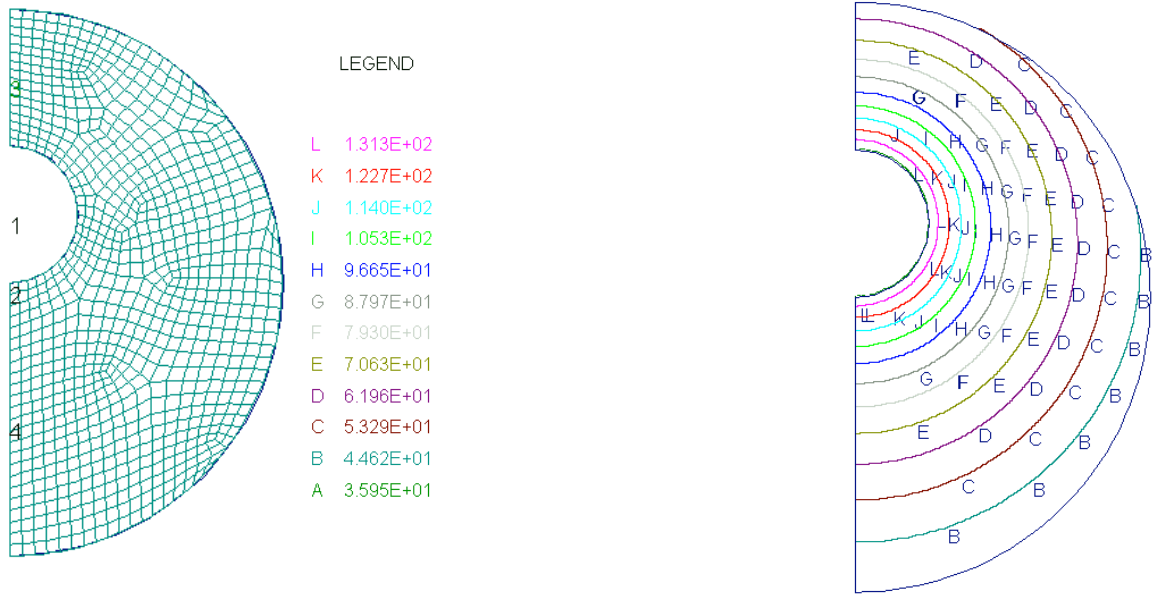
50. Click the **Post Processor** button on the INSTED/CFD (2D) Solver Main dialog box.  
→ The INSTED/CFD (2D) Post-processor program is launched. The results from the INSTED/CFD (2D) Solver are automatically loaded.
51. Click the **Display** button on the 'Controls' dialog box.  
→ The 'Display' dialog box appears.
52. Select the most recent print step for viewing from the 'Time Step' listbox.
53. Click the **Ok** button to dismiss the 'Display' dialog box.
54. View the Contour Plot of Temperature.
55. Locate and open the file 'lola.out' in the working directory (\cfd for the current example).
56. Inspect and compare the values at selected nodal points.

Table 1.1 Heat Conduction Between Two Cylinders, with Film Coefficient Boundary Conditions

x-coord	y-coord	Seegerlind	INSTED	Difference
0.0	-2.0	68.97	67.70	< 2 %
0.0	-4.0	36.58	36.22	< 1 %
0.0	2.0	140	140	< 1 %
0.0	3.0	88.48	87.10	< 2 %
0.0	4.0	55.71	55.36	< 1 %
1.0	1.0	140	140	< 1 %
0.707	0.293	140	140	< 1 %
4.0	0.0	42.35	42.10	< 1 %

File: *lola.out*





Number of nodes = 503  
 Number of 9 -node elements = 113

(a) Computational Model

(b) Contour Plot of Temperature

Figure 1.2 Mesh and temperature plot for Sample Problem 1.

**Reference:**

Seegerlind, L. J., 1984. Applied Finite Element Analysis. Second Edition. John Wiley & Sons, Ltd. New York. 405-408.

TEST PROBLEM 2  
HEAT CONDUCTION WITH VOLUMETRIC HEAT SOURCE

**Problem Statement**

Analyze the time-dependent equation

$$\frac{\partial T}{\partial t} - \nabla^2 T = 1,$$
$$\Omega = \{(x, y) : 0 \leq (x, y) \leq 1\}$$

subject to the boundary conditions, for  $t \geq 0$ ,

$$T = 0 \text{ on lines } x = 1 \text{ and } y = 1$$

$$\frac{\partial T}{\partial n} = 0 \text{ on lines } x = 0 \text{ and } y = 0$$

and the initial boundary conditions for  $(x, y)$  in  $W$

$$T = 0 ; t = 0$$

**Physical and Computational Domain**

The physical and computational domain is shown below:

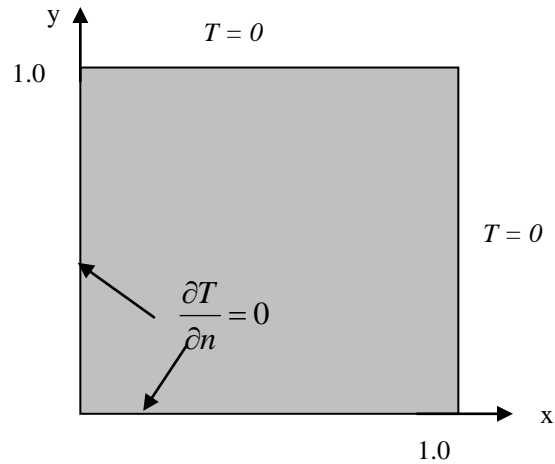


Figure 2.1 Physical and Computational Domain

**Governing Equations**

This is written above as part of the problem statement.

## Initial Condition

The problem will be analyzed as a transient one, for which the initial condition,  $T = 0$ , is used everywhere. This is also the default initial condition in INSTED for this problem.

## Boundary Conditions

This is written above as part of the problem statement.

## Computational Grid

This test problem will be analyzed with a mesh consisting of 64 9-node elements and 289 nodal points, including the nodes at the centroids of the elements.

## Project Files

The files required to reproduce our results for this sample problem are:

reddy.mes	Input data for use in Mesh Generator to generate mesh
reddy.cfd	Input data for CFD solver

## Location of Project Files

The project files above are located in the directory `\insted30\cfd\samples`.

## CFD Type

This is a transient heat conduction problem. Therefore, the velocities are not solved. There are also no scalars. Since this problem is posed in terms of non-dimensional parameters, one could in principle analyze this problem using CFD Type 1 or Type 2. However, the temperature equation for this problem resembles the form in CFD Type 1 if  $k = H = 1$  ( $H$  is non-dimensional volumetric heat source). Moreover, for Type 2, it is not clear what  $Re$  value must be used to obtain the right temperature equation and at the same time ensure a motionless state. The reason for this is the appearance of the Peclet number  $Pe = RePr$ , in the temperature equation for CFD Type 2. We have selected CFD Type 1 for this problem to avoid any ambiguities.

## Model Building and Mesh Generation

Refer to INSTED/CFD (2D) manual for the description of the INSTED Mesh Generation GUI. The status bar, Input Box, modeling tools, Main dialog box are discussed in the manuals.

The INSTED model for this sample problem is a square in which the four enclosing lines are oriented in a counter-clockwise sense. The rectangle modeling tool in INSTED will be used to produce a square with counter-clockwise orientation. The steps for generating the mesh for the sample problem are as follows:

1. Start INSTED/CFD (2D) Mesh Generator. Skip this step if the program is already open. However, you should close other INSTED/CFD (2D) programs that may have been opened through the Mesh Generator.

2. Click the **New Project** button on the Main dialog box.
3. Click the **Limit** button on the 'Controls' dialog box.  
→ The 'Limits' dialog box appears.
4. Type in -0.2, 1.2, -0.2, 1.2 for the lower x, higher x, lower y and higher y screen limits in the 'Limits' dialog box.
5. Click the **OK** button on the 'Limits' dialog box.

#### To draw the square

6. Click the Rectangle tool button among the Modeling Tools that reside on the Main dialog box.  
→ Status bar displays the message: 'Pick 1<sup>st</sup> Corner Point'.
7. Type in 0,0 in the Input box and press Enter.  
→ Status bar displays the message: 'Pick 2<sup>nd</sup> Corner Point'
8. Type in 1,1 in the Input box and press Enter.

#### Defining number of elements per line

9. Click the **Line Properties** button on the Main dialog box,  
→ The 'Line Properties' dialog box appears.
10. Enter the values of 8, 8, 8, and 8 for the number of elements for lines 1, 2, 3, and 4 respectively. Press Enter after typing each value to move to the next line in the listbox.
11. Click the OK button to dismiss this dialog box.

#### Defining the area to mesh

12. Click the **Define Area** button on the Main dialog box,  
→ The 'Area Definition' dialog box appears.
13. Click the **Add** button on this dialog box.  
→ An area listed as '1' is added in the 'Areas' listbox.
14. Click the first line of the 'Boundaries' listbox.
15. Type 1 and press Enter. This is meant to include boundary 1 (made of Rectangle with lines 1,2,3, and 4) in this area,
16. Click the **OK** button to dismiss the 'Area Definition' dialog box.

#### Applying boundary conditions

17. Click the **Line BC** button on the Main dialog box,  
→ Status bar displays message: 'Pick Side'.
18. Click the Line 3 on the graphics screen or type 3 and Press Enter in the Input Box  
→ The 'Line BC' dialog box appears with the heading -"Boundary Condition, Side 3" (confirm from the title of the dialog box that data is being received for Line 3. This will not be the case if you accidentally selected some other line.).
19. Click the temperature boundary condition listbox's drop-down arrow and select the Dirichlet condition from the list.  
→ An input box appears beside the boundary condition listbox once Dirichlet condition is selected.
20. Enter a value of 0 in the input box.
21. Click the **OK** button to dismiss the dialog box.
22. Repeat this procedure for Line 2.

#### Mesh and Confirm Mesh Results

23. Click the **Mesh** button on the Main dialog box,  
→ A mesh of the internal of our loop of lines is displayed on the 'Graphics' dialog box.
24. Click the **Display** button on the 'Controls' dialog box.
25. From the listbox, select 'Dirichlet, Temp'
26. Click the **Ok** button on this dialog box to view nodes with Dirichlet temperature boundary conditions.

27. Click the **Refresh** button twice on the 'Controls' dialog box to return the screen to its original condition.

**Note:** You should have the same result as if you loaded the file `\insted30\cfd\samples\reddy.mes`. You may also choose to save your current work. In this case, refer to the sections 'Loading a file' and 'Saving a file' in the first chapter of the Users Manual.

### Obtaining a Solution to the Problem

28. Click the **Solver** button on the Main dialog box.
  - The INSTED Solver program is launched. The mesh generated from the Mesh Generation program is automatically loaded.
29. Click the **Load Project** button on the Main dialog box of the Solver program.
30. Type \*.\* in the filename section of the operating system 'Open File' dialog box that appears, and press Enter.
  - A list of files in the current directory is displayed.
31. Locate the file 'reddy.cfd' from the 'Samples' directory.
32. Click this file and press Enter.
33. Click the **Prob. Description** button on the Main dialog box and inspect the loaded parameters.
34. Click the OK button to dismiss the 'Prob. Description' dialog box.
35. Click the **Material Properties** button on the Main dialog box and inspect the loaded material properties. Note particularly the heat flux parameters.
36. Click the OK button to dismiss the 'Material Properties' dialog box.
37. Click the Solve button on the Main dialog box
  - Program begins to generate a solution to the problem. The graphic screen is converted to one that displays the status of the solution process.
38. Wait for the solution to complete.

### Analyzing the Result

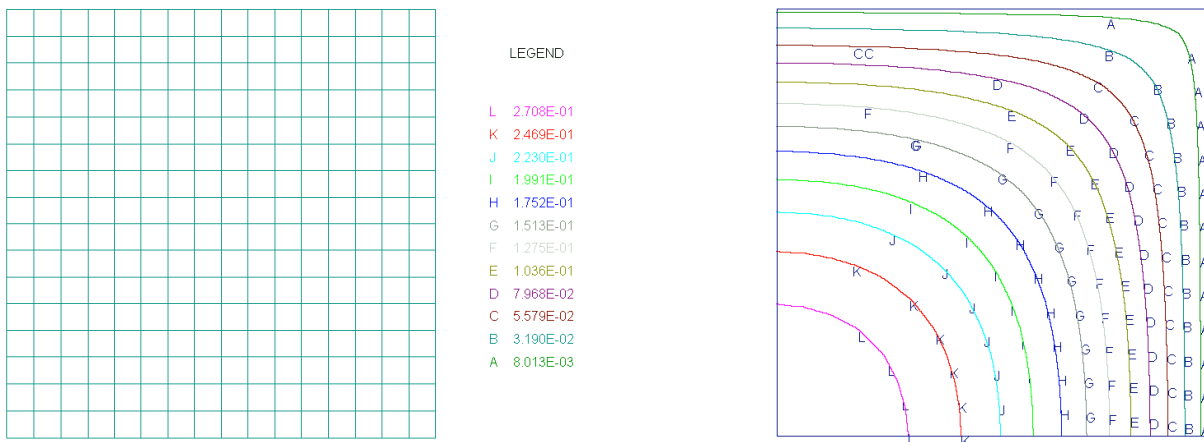
(Refer to the User's Manual for general details about INSTED/CFD (2D) Post-Processing.)

39. Click the **Post Processor** button on the Main dialog box.
  - The INSTED Postprocessor program is launched. The results from the
40. Click the **Display** button on the 'Controls' dialog box.
  - The 'Display' dialog box appears.
41. Select the most recent time step for viewing from the 'Time Step' listbox.
42. Click the **Ok** button to dismiss the 'Display' dialog box.
43. View the Contour Plot of Temperature.
44. Locate and open the file 'lola.out' in the working directory (`\cfd` for the current example).
45. Inspect and compare the values at selected nodal points.

**Table 1.1 Comparison of Results with Source**

<b>x-coord</b>	<b>y-coord</b>	<b>Exact Solution</b>	<b>INSTED</b>	<b>Difference</b>
0.0	0.0	0.2947	0.2947	< 1 %
0.0	0.25	0.2789	0.2789	< 1 %
0.0	0.5	0.2293	0.2293	< 1 %
0.0	0.75	0.1397	0.1397	< 1 %
0.0	1.0	0.0	0.0	< 1 %

File: *lola.out*



Number of nodes = 289  
 Number of 9-node elements = 64

**Figure 1.2 Mesh and temperature plot for Sample Problem 1.**

**Reference:**

Reddy, J. N., 1984. An Introduction to the Finite Element Method. McGraw Hill Book Co. New York. 303-305.



TEST PROBLEM 3  
**AXISYMETRIC CONDUCTION IN A CYLINDER**

**Problem Statement**

Consider a cylinder of height  $L = 0.02$  m, radius  $r = 0.01$ m, thermal conductivity  $k = 25\text{W/cm}^\circ\text{C}$  and constant internal heat generation  $Q''' = 5 \times 10^8 \text{ W/m}^3$ . The top and bottom faces and the curved surface of the cylinder are maintained at  $T_0 = 100^\circ\text{C}$ . Calculate the axisymmetric temperature distribution in the  $(r, z)$  plane of the cylinder.

**Physical and Computational Domain**

The physical domain is shown in Fig. 3.1 (a)

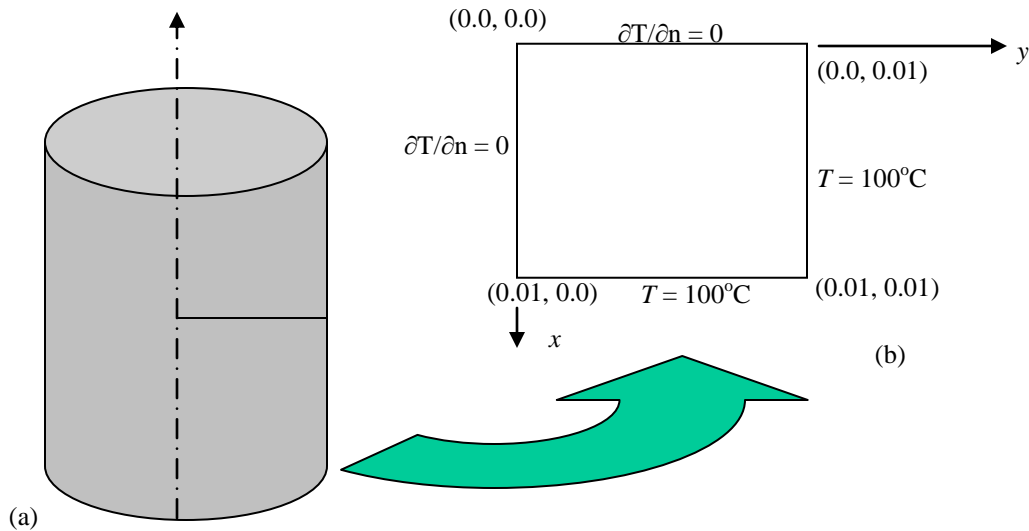


Figure 3.1 Physical Domain for the Axisymmetric Conduction Problem

The computational model is shown in Fig. 3.1 (b).

**Governing Equations**

The present sample problem is a conduction one with the equation

$$k \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial T}{\partial r} \right) + \frac{\partial^2 T}{\partial z^2} \right] + Q''' = 0$$

where  $k = 25\text{W/cm}^\circ\text{C}$  is the thermal conductivity and  $Q''' = 5 \times 10^8 \text{ W/m}^3$  is the volumetric heat generation. The coordinates  $(r, z) \equiv (y, x)$  are the radial and axial coordinates, respectively.



You must specify the axial coordinate ( $z$ ) as the  $x$ - coordinate in INSTED and the radial coordinate as the  $y$ - coordinate. This is the convention for axisymmetrical problems in INSTED.

### CFD Type

This problem will be solved as CFD Type 3; that is, without non-dimensionalization. We will also take advantage of symmetry to simplify the problem. As a result, the computational domain shown above was derived based on symmetry along the  $r$ - and  $z$ -axis.

### Initial Condition

The problem will be analyzed as a transient one, for which the initial condition,  $T = 0$ , is used everywhere. This is also the default initial condition in INSTED.

### Boundary Conditions

This is illustrated in Figure 3.1 (b) and summarized below:

$$\begin{aligned}x = 0 : \frac{\partial T}{\partial n} &= 0 \\x = 0.01 : T &= 100^{\circ} C \\y = 0 : \frac{\partial T}{\partial n} &= 0 \\y = 0.01 : T &= 100^{\circ} C\end{aligned}$$

Also note that there is additional heat generation  $Q''' = 5 \times 10^8 \text{ W/m}^3$ .

### Computational Grid

This problem will be analyzed with an unstructured mesh consisting of 25 9-node elements and 121 nodal points, including those at the centroids of the elements.

### Project Files

The files required to reproduce our results for this sample problem are:

axiscond.mes	Input data for use in Mesh Generator to generate mesh
axiscond.cfd	Input data for CFD solver

### Location of Project Files

The project files above are located in the directory `\insted30\cfd\samples`.

### Model Building and Mesh Generation

The model for this sample problem will be composed of a square with a counter-clockwise orientation. The rectangle modeling tool in INSTED will be used to produce the square.

1. Start INSTED/CFD (2D) Mesh Generator.
2. Click the **New Project** button on the Main dialog box.
3. Click the **Limit** button on the 'Controls' dialog box.  
→ The 'Limits' dialog box appears.
4. Type in -0.002, 0.012, -0.002, 0.012 for the lower x, higher x, lower y and higher y screen limits in the 'Limits' dialog box.
5. Click the OK button to dismiss this dialog box.

#### To draw the square

6. Click the Rectangle tool button among the Modeling Tools that reside on the Main dialog box.  
→ Status bar displays the message: 'Pick 1<sup>st</sup> Corner Point'.
7. Type in 0,0 in the Input box and press Enter.  
→ Status bar displays the message: 'Pick 2<sup>nd</sup> Corner Point'
8. Type in 0.01,0.01 in the Input box and press Enter.

#### Defining number of elements per line

9. Click the **Line Properties** button on the Main dialog box,  
→ The 'Line Properties' dialog box appears.
10. Enter the values of 5, 5, 5, and 5 for the number of elements for lines 1, 2, 3, and 4 respectively. Press Enter after typing each value to move to the next line in the listbox.
11. Click the OK button to dismiss this dialog box.

#### Defining the area to mesh

12. Click the **Define Area** button on the Main dialog box,  
→ The 'Area Definition' dialog box appears.
13. Click the **Add** button on this dialog box.  
→ An area listed as '1' is added in the 'Areas' listbox.
14. Click the first line of the 'Boundaries' listbox.
15. Type 1 and press Enter. This is meant to include boundary (made of Rectangle with lines 1,2,3, and 4) in this area,
16. Click the **OK** button to dismiss the 'Area Definition' dialog box.

#### Applying boundary conditions

17. Click the **Line BC** button on the Main dialog box,  
→ Status bar displays message: 'Pick Side'.
18. Click the Line 2 on the graphics screen or type 2 and Press Enter in the Input Box  
→ The 'Line BC' dialog box appears with the heading -"Boundary Condition, Side2" (confirm from the title of the dialog box that data is being received for Line 2. This will not be the case if you accidentally selected some other line.).
19. Click the drop down list box for temperature and select a Dirichlet boundary condition from the list and enter a value of 100.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
20. Click the **OK** button to dismiss the dialog box.
21. Repeat the procedure (17-20) for Line 3.

#### Mesh and Confirm Mesh Results

22. Click the **Mesh** button on the Main dialog box,  
→ A mesh of the internal of our loop of lines is displayed on the 'Graphics' dialog box.
23. Click the **Display** button on the 'Controls' dialog box.
24. From the listbox, select 'Dirichlet, Temp'
25. Click the **Ok** button on this dialog box to view nodes with Dirichlet temperature boundary conditions.
26. Click the **Refresh** button on the 'Controls' dialog box to return the screen to its original condition.

**Note:** You should have the same result as if you loaded the file  
\\insted30\cfd\samples\axiscond.mes. You may also choose to save your current work. In this case,  
refer to the sections 'Loading a file' and 'Saving a file' in the first chapter of the Users Manual.

### Obtaining a Solution to the Problem

27. Click the **Solver** button on the INSTED/CFD (2D) Mesh Generator Main dialog box.  
→ The INSTED/CFD (2D) Solver program is launched. The mesh generated from the Mesh Generation program is automatically loaded.
28. Click the **Load Project** button on the Main dialog box of the Solver program.
29. Type \*.\* in the filename section of the operating system 'Open File' dialog box that appears, and press Enter.  
→ A list of files in the current directory is displayed.
30. Locate the file 'axiscond.cfd' from the 'Samples' directory.
31. Click this file and press Enter.
32. Click the **Prob. Description** button on the Main dialog box and inspect the loaded parameters.
33. Click the OK button to dismiss the 'Prob. Description' dialog box.
34. Click the **Material Properties** button on the Main dialog box and inspect the loaded material properties. Note particularly the heat flux parameters.
35. Click the OK button to dismiss the 'Material Properties' dialog box.
36. Click the Solve button on the Main dialog box  
→ Program begins to generate a solution to the problem. The graphic screen is converted to one that displays the status of the solution process.
37. Wait for the solution to complete.

### Analyzing the Result

(Refer to the User's Manual for general details about INSTED/CFD (2D) Post-Processing.)

38. Click the **Post Processor** button on the INSTED/CFD (2D) Solver Main dialog box.  
→ The INSTED/CFD (2D) Post-processor program is launched. The results from the
39. Click the **Display** button on the 'Controls' dialog box.  
→ The 'Display' dialog box appears.
40. Select the most recent time step for viewing from the 'Time Step' listbox.
41. Click the **Ok** button to dismiss the 'Display' dialog box.
42. View the Contour Plot of Temperature.
43. Locate and open the file 'lola.out' in the working directory (\cfd for the current example).
44. Inspect and compare the values at selected nodal points.

**Table 3.1 Comparison of Results**

<b>r (z = 0.009)</b>	<b>Reddy</b>	<b>INSTED</b>	<b>Difference</b>
0.000	195.86	195.21	< 1 %
0.001	195.03	194.96	< 1 %
0.002	193.06	192.68	< 1 %
0.003	189.78	189.44	< 1 %
0.004	185.07	184.73	< 1 %
0.005	178.74	178.40	< 1 %
0.006	170.50	170.12	< 1 %
0.007	159.91	159.50	< 1 %
0.008	146.20	145.62	< 1 %
0.009	127.81	127.23	< 1 %
0.010	100.0	100.0	< 1 %

<b>r (z = 0.0)</b>	<b>Reddy</b>	<b>INSTED</b>	<b>Difference</b>
0.000	504.64	501.30	< 1 %
0.001	499.82	497.70	< 1 %
0.002	488.52	486.81	< 1 %
0.003	469.93	468.50	< 1 %
0.004	443.70	442.49	< 1 %
0.005	409.43	408.43	< 1 %
0.006	366.68	365.88	< 1 %
0.007	314.92	314.33	< 1 %
0.008	253.61	253.22	< 1 %
0.009	182.16	181.97	< 1 %
0.010	100.0	100.0	< 1 %

**Reference:**

Reddy, J. N. & Gartling, D. K. (1994). The Finite Element Method in Heat Transfer and Fluid Dynamics. CRC Press, Boca Raton, Florida, USA. 69-70.

## TEST PROBLEM 4

### PRESSURE DISTRIBUTION IN A VISCOUS FLOW OF A LUBRICANT BEARING

#### Problem Statement

Calculate the pressure distribution in a lubricant flowing inside a slider bearing (Figure 4.1). The slider bearing consists of a short sliding pad moving at a velocity  $u = U = 30$  relative to a stationary pad inclined at a small angle with respect to the stationary pad, as the small gap between the two pads is filled with a lubricant. Since the ends of the bearing are generally open, the pressure there is atmospheric, which will be taken as zero. The pressure distribution inside the bearing is, in general, a function of  $x$  and  $y$ , and is set up in the gap.

An exact solution, assuming that the pressure does not vary in  $y$ , can be shown as:

$$P = \frac{6\mu U_0 (h_2 - h_1)(h - h_1)}{h^2 (h_2^2 - h_1^2)}$$

$$u = \left( U_0 - \frac{1}{2\mu} h^2 \frac{dP}{dx} \frac{y}{h} \right) \left( 1 - \frac{y}{h} \right)$$

$$\tau_{xy} = \mu \frac{\partial u}{\partial y} = \frac{dP}{dx} \left( y - \frac{h}{2} \right) - \mu \frac{U_0}{h}$$

$$h(x) = h_2 + \frac{h_2 - h_1}{L} x$$

#### CFD Type

This problem will be solved as CFD Type 3; that is, in the dimensional form.

#### Physical and Computational Domain

The physical domain is shown in the figure below:

In the figure,  $h_2 = 2h_1 = 8 \times 10^{-4}$ ,  $L = 0.36$ ,  $\mu = 8 \times 10^{-4}$ ,  $U_0 = 30$ ,  $\rho = 1$ .

Caution: You should be aware of the large aspect ratio of the physical system under consideration. This quantity is equal to  $L/h_1 = 900$ . Under this condition, the truly automatic mesh generation method in INSTED defaults to the standard structured mesh generation approach.

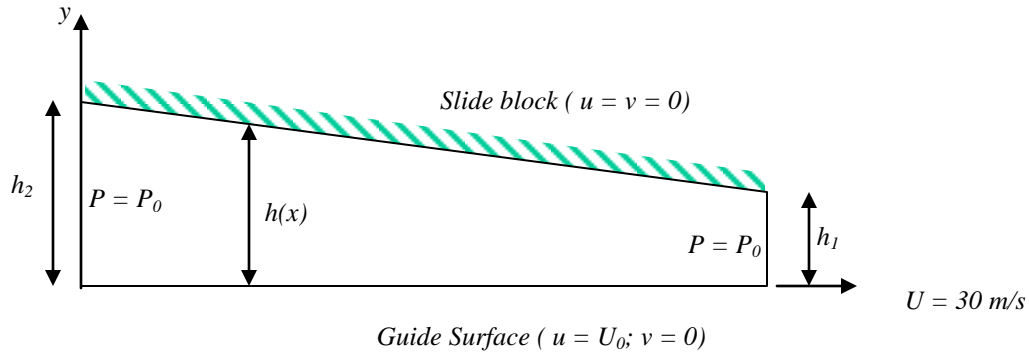


Figure 4.1 Physical domain for the bearing problem

The computational domain will have the same orientation and dimensions as the physical domain shown in this figure.

### Computational Grid

This problem will be analyzed with an unstructured mesh consisting of 60 9-node elements and 347 nodal points, including those at the centroids of the elements.

### Governing Equations

The governing equations for CFD Type 3: Engineering CFD, are used on the assumption that the data is given in dimensional quantities. (Note that no units have been specified.) Isothermal flow conditions are assumed. The governing equations are:

$$\begin{aligned} \nabla \cdot \mathbf{u} &= 0 \\ \frac{\partial u}{\partial t} + \mathbf{u} \cdot \nabla u &= -\frac{1}{\rho_0} \frac{\partial p}{\partial x} + \frac{1}{\rho_0} \nabla \cdot \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^*) \Big|_x \\ \frac{\partial v}{\partial t} + \mathbf{u} \cdot \nabla v &= -\frac{1}{\rho_0} \frac{\partial p}{\partial y} + \frac{1}{\rho_0} \nabla \cdot \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^*) \Big|_y \end{aligned}$$

### Initial Condition

The default initial conditions in INSTED as described in Sample Problem 1 is used for this problem. That is,  $u(x, y, 0) = v(x, y, 0) = 0$ .

### Boundary Conditions

The appropriate boundary conditions for this sample problem is as follows:

$$\text{Left } (x = 0): t_x = t_y = 0$$

$$\text{Right } (x = 36): t_x = t_y = 0$$

Bottom ( $y = 0$ ):  $u = 30, v = 0$

Top:  $u = 0, v = 0$

## Project Files

The files required to reproduce our results for this sample problem are:

slider1.mes	Input data for use in Mesh Generator
slider.cfd	Input data for CFD solver

## Location of Project Files

The project files above are located in the directory `\insted30\cfd\samples`.

## Model Building and Mesh Generation

The CFD model for this sample problem will be generated using a structured mesh generation procedure consisting of eight sides. The **New Block** modeling tool in INSTED will give us a structured block. Note that the points of the block must be entered also in a counter-clockwise direction. This is illustrated in the instructions below.

1. Start INSTED/CFD (2D) Mesh Generator.
2. Click the **New Project** button on the Main dialog box.
3. Click the **Limit** button on the 'Controls' dialog box.  
→ The 'Limits' dialog box appears.
4. Type in -0.02, 0.4, -0.0002, 0.001 for the lower  $x$ , higher  $x$ , lower  $y$  and higher  $y$  screen limits in the 'Limits' dialog box.
5. Click the OK button to dismiss this dialog box.

### To draw the object

6. Click the **New Boundary** button on the Main dialog box.  
→ Status bar displays the message: 'Pick Point 1'.
  7. Type in 0,0 in the Input box and press Enter.  
→ Status bar displays the message: 'Pick Point 2'.
  8. Type in 0.18,0.0 in the Input box and press Enter.
  9. Type in 0.36,0.0 in the Input box and press Enter.
  10. Type in 0.36, 2.0E-04 in the Input box and press Enter.
  11. Type in 0.36, 4.0E-04 in the Input box and press Enter.
  12. Type in 0.18, 4.0E-04 in the Input box and press Enter.
  13. Type in 0.0, 8.0E-04 in the Input box and press Enter.
  14. Type in 0.0, 4.0E-04 in the Input box and press Enter.
  15. Type in 0.0,0.0 in the Input box and press Enter.  
→ The slider shape as illustrated in the Figure 4.1 is obtained.
- ### Defining number of elements per line
16. Click the **Line Properties** button on the Main dialog box,  
→ The 'Line Properties' dialog box appears.
  17. Enter the values of 9, 3, 9, and 3 for the number of elements for lines 1, 2, 3, and 4 respectively. Press Enter after typing each value to move to the next line I the listbox.
  18. Click the OK button to dismiss this dialog box.

### Defining the area to mesh

19. Click the **Define Area** button on the Main dialog box,  
→ The 'Area Definition' dialog box appears.
20. Click the **Add** button on this dialog box.  
→ An area listed as '1' is added in the 'Areas' listbox.
21. Click the first line of the 'Boundaries' listbox.
22. Type 1 and press Enter. This is meant to include boundary (made of Rectangle with lines 1,2,3, and 4) in this area,
23. Click the **OK** button to dismiss the 'Area Definition' dialog box.

### Applying boundary conditions

24. Click the **Line BC** button on the Main dialog box,  
→ Status bar displays message: 'Pick Side'.
25. Click the Line 1 on the graphics screen or type 1 and Press Enter in the Input Box  
→ The 'Line BC' dialog box appears with the heading -"Boundary Condition, Side1" (confirm from the title of the dialog box that data is being received for Line 1. This will not be the case if you accidentally selected some other line.).
26. Click the drop down list box for u-velocity and select a Dirichlet boundary condition from the list and enter a value of 30.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
27. Click the drop down list box for v-velocity and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
28. Click the **OK** button to dismiss the dialog box.
29. Repeat the procedure (20-24) for Line 3 setting Dirichlet conditions for u-velocity and v-velocity with values 0.0 and 0.0 respectively.

### Mesh and Confirm Mesh Results

30. Click the **Mesh** button on the Main dialog box,  
→ A mesh of the internal of our loop of lines is displayed on the 'Graphics' dialog box.
31. Click the **Display** button on the 'Controls' dialog box.
32. From the listbox, select 'Dirichlet, Temp'
33. Click the **Ok** button on this dialog box to view nodes with Dirichlet temperature boundary conditions.
34. Click the **Refresh** button on the 'Controls' dialog box to return the screen to it original condition.

### Turning on Node & Element Numbering

35. Click the Display button on the 'Controls' dialog box.  
→ The 'Display' dialog box appears
36. Click the 'Display Node Numbers' radio button to turn on display of node numbers
37. Click the 'Display Element Numbers' radio button to turn on display of node numbers
38. Click the **OK** button to dismiss the dialog box.

**Note:** You should have the same result as if you loaded the file \insted30\cfd\samples\slider.mes. You may also choose to save your current work. Refer to the sections 'Loading a file' and 'Saving a file' in the first chapter of the Users Manual.

### **Obtaining a Solution to the Problem**

39. Click the **Solver** button on the INSTED/CFD (2D) Mesh Generator Main dialog box.



- The INSTED/CFD (2D) Solver program is launched. The mesh generated from the Mesh Generation program is automatically loaded.
- 40. Click the **Load Project** button on the Main dialog box of the Solver program.
- 41. Type \*.\* in the filename section of the operating system 'Open File' dialog box that appears, and press Enter.
- A list of files in the current directory is displayed.
- 42. Locate the file 'slider.cfd' from the 'Samples' directory.
- 43. Click this file and press Enter.
- 44. Click the **Prob. Description** button on the Main dialog box and inspect the loaded parameters.
- 45. Click the OK button to dismiss the 'Prob. Description' dialog box.
- 46. Click the **Material Properties** button on the Main dialog box and inspect the loaded material properties. Note particularly the heat flux parameters.
- 47. Click the OK button to dismiss the 'Material Properties' dialog box.
- 48. Click the Solve button on the Main dialog box
- Program begins to generate a solution to the problem. The graphic screen is converted to one that displays the status of the solution process.
- 49. Wait for the solution to complete.

### Analyzing the Result

(Refer to the User's Manual for general details about INSTED/CFD (2D) Post-Processing.)

- 50. Click the **Post Processor** button on the INSTED/CFD (2D) Solver Main dialog box.
- The INSTED/CFD (2D) Post-processor program is launched. The results from the Solver is automatically loaded (Note that the model does not appear clearly initially due to its high aspect ratio.)
- 51. Click the **Axis** button on the 'Controls' dialog box.
- The 'Axis' dialog box appears.
- 52. Enter a value of 1.2 for the x-scale.
- 53. Enter a value of 140 for the y-scale.
- 54. Click the **Ok** button to dismiss the dialog box.
- 55. Click the **Display** button on the 'Controls' dialog box.
- The 'Display' dialog box appears.
- 56. Select the most recent time step for viewing from the 'Time Step' listbox.
- 57. Click the **Ok** button to dismiss the 'Display' dialog box.
- 58. View the Contour Plot of Pressure.
- 59. Locate and open the file 'lola.out' in the working directory (\cfd for the current example).
- 60. Inspect and compare the values at selected nodal points.

### Comparison of INSTED Results with Analytical Solution

x-coordinate	Analytic Solution	INSTED Result	Difference
0.0063	472.46	468.52	< 1 %
0.0132	989.65	989.56	< 1 %
0.0207	1551.14	1551.10	< 1 %
0.0375	2804.01	2805.70	< 1 %
0.0567	4221.43	4224.40	< 1 %
0.0783	5785.07	5788.90	< 1 %

0.1023	7462.06	7466.00	< 1 %
0.1575	10886.40	10885.00	< 1 %
0.1887	12367.20	12357.00	< 1 %
0.2479	13465.98	13462.00	< 1 %
0.2775	12627.14	12627.00	< 1 %
0.3039	10634.62	10631.00	< 1 %
0.3471	3477.632	3465.40	< 1 %
0.3559	1188.77	1173.00	< 1 %

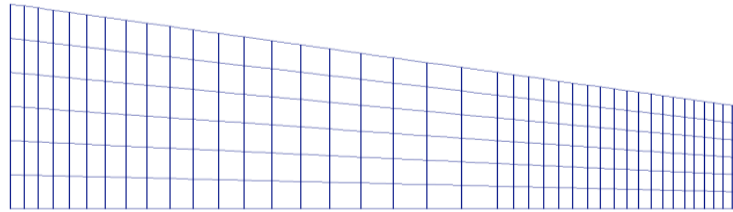


Figure 4.2 Computational Mesh of the Slider Problem

LEGEND

- L 1.245E+04
- K 1.132E+04
- J 1.018E+04
- I 9.050E+03
- H 7.917E+03
- G 6.784E+03
- F 5.651E+03
- E 4.518E+03
- D 3.385E+03
- C 2.252E+03
- B 1.120E+03
- A -1.340E+01

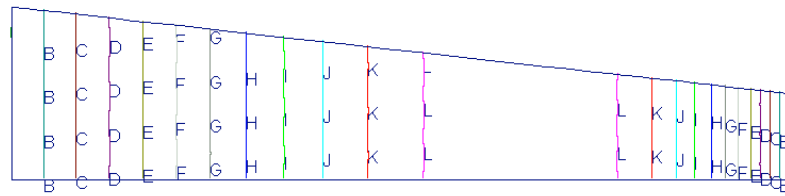


Figure 4.3 Plot of Pressure Contour for the Slider Problem

**Reference:**

Reddy, J. N. & Gartling, D. K. (1994). The Finite Element Method in Heat Transfer and Fluid Dynamics. CRC Press, Boca Raton, Florida, USA. 69-70.

TEST PROBLEM 5  
**FLUID SQUEEZED BETWEEN PARALLEL PLATES**

**Problem Statement**

The problem being considered is the (Stokes) flow of a viscous incompressible material squeezed between two long parallel plates. A plane flow (in the plane formed by the width of and the distance between the plates) is considered. Although this is a moving boundary problem, we wish to determine the velocity and pressure fields for a fixed distance between the plates, assuming that a state of plane flow exists. Results are compared with Penalty and mixed formulations (Reddy and Gartling, 1994) as well as with series solutions by Nadai (1963).

**CFD Type**

This problem will be solved as a CFD Type 2, posing it in the non-dimensional units. We will also take advantage of symmetry in the  $x$ - and  $y$ - directions.

**Physical and Computational Domain**

The physical and computational domain are shown below:

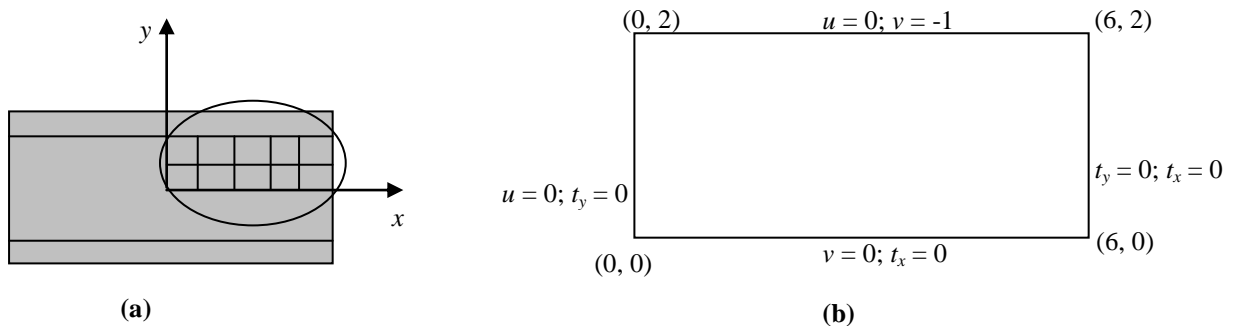


Figure 5.1 Physical (a) and Computational (b) model for the test problem on viscous flow between parallel plates

**Governing Equations**

The governing equations are the same as those presented in the main CFD manual for CFD Type 2: Forced/Mixed Convection except that the present case is isothermal and the Reynolds number is very small. This problem will be solved in an inertial frame since there is no rotation of the system. The resulting equations are:

$$\nabla \cdot \mathbf{u} = 0,$$

$$\frac{\partial u}{\partial t} + \mathbf{u} \cdot \nabla u = -\frac{\partial p}{\partial x} + \frac{1}{\text{Re}} \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^*)_x,$$

$$\frac{\partial v}{\partial t} + \mathbf{u} \cdot \nabla v = -\frac{\partial p}{\partial y} + \frac{1}{\text{Re}} \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^*),$$

$$\text{Re}_r = \text{Ra}_r = 0,$$

where  $(u, v)$  are the velocities in the coordinate directions, and  $\eta$  is the non-dimensional value of the absolute viscosity. The Reynolds number,  $\text{Re}$ , is arbitrarily takes as 0.001, consistent with Stokes flow requirement,  $\eta = 1$  and  $\text{Pr}$  is arbitrarily set to 7.0. The mathematical symbols in these equations take on their usual meaning and are explained in greater detail in the User's Manual.

### Initial Condition

The default initial conditions in INSTED are used for this problem. That is,  $u(x, y, 0) = v(x, y, 0) = 0$ .

### Boundary Conditions

$$x = 0: u = 0, t_y = 0$$

$$x = 6: t_x = 0, t_y = 0$$

$$y = 0: v = 0, t_x = 0$$

$$y = 2: u = 0, v = -1$$

### Project Files

The files required to reproduce our results for this sample problem are:

squeeze.mes	Input data for use in Mesh Generator to generate mesh
squeeze.cfd	Input data for CFD solver

### Location of Project Files

The project files above are located in the directory `\insted30\cfd\samples`.

### Model Building and Mesh Generation

The model for this sample problem will be composed of a square with a counter-clockwise orientation. The rectangle modeling tool in INSTED will be used to produce the square.

1. Start INSTED/CFD (2D) Mesh Generator.
2. Click the **New Project** button on the Main dialog box.
3. Click the **Limit** button on the 'Controls' dialog box.  
→ The 'Limits' dialog box appears.
4. Type in -0.5, 6.5, -0.2, 2.2 for the lower x, higher x, lower y and higher y screen limits in the 'Limits' dialog box.
5. Click the OK button to dismiss this dialog box.

#### To draw the square

6. Click the Rectangle tool button among the Modeling Tools that reside on the Main dialog box.

→ Status bar displays the message: 'Pick 1<sup>st</sup> Corner Point'.

7. Type in 0,0 in the Input box and press Enter.

→ Status bar displays the message: 'Pick 2<sup>nd</sup> Corner Point'

8. Type in 6, 2 in the Input box and press Enter.

#### Defining number of elements per line

9. Click the **Line Properties** button on the Main dialog box,

→ The 'Line Properties' dialog box appears.

10. Enter the values of 12, 4, 12, and 4 for the number of elements for lines 1, 2, 3, and 4 respectively. Press Enter after typing each value to move to the next line in the listbox.

11. Click the OK button to dismiss this dialog box.

#### Defining the area to mesh

12. Click the **Define Area** button on the Main dialog box,

→ The 'Area Definition' dialog box appears.

13. Click the **Add** button on this dialog box.

→ An area listed as '1' is added in the 'Areas' listbox.

14. Click the first line of the 'Boundaries' listbox.

15. Type 1 and press Enter. This is meant to include boundary 1 (made of Rectangle with lines 1,2,3, and 4) in this area.

16. Click the **OK** button to dismiss the 'Area Definition' dialog box.

#### Applying boundary conditions

17. Click the **Line BC** button on the Main dialog box,

→ Status bar displays message: 'Pick Side'.

18. Click the Line 3 on the graphics screen or type 3 and Press Enter in the Input Box

→ The 'Line BC' dialog box appears with the heading -"Boundary Condition, Side 3" (confirm from the title of the dialog box that data is being received for Line 3. This will not be the case if you accidentally selected some other line.).

19. Click the drop down list box for u-velocity and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.

20. Click the drop down list box for v-velocity and select a Dirichlet boundary condition from the list and enter a value of -1.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.

21. Click the **OK** button to dismiss the dialog box.

22. Repeat the procedure (17-20) for Line 4 setting Dirichlet conditions only for u-velocity with a value of 0.0.

23. Repeat the procedure (17-20) for Line 1 setting Dirichlet conditions only for v-velocity with a value of 0.0.

#### Mesh and Confirm Mesh Results

24. Click the **Mesh** button on the Main dialog box,

→ A mesh of the internal of our loop of lines is displayed on the 'Graphics' dialog box.

25. Click the **Display** button on the 'Controls' dialog box.

26. From the listbox, select 'Dirichlet, Temp'

27. Click the **Ok** button on this dialog box to view nodes with Dirichlet temperature boundary conditions.

28. Click the **Refresh** button on the 'Controls' dialog box to return the screen to its original condition.

**Note:** You should have the same result as if you loaded the \insted30\cfd\samples\axiscond.mes. You may also choose to save your current work. Refer to the sections 'Loading a file' and 'Saving a file' in the first chapter of the Users Manual.

## Obtaining a Solution to the Problem

29. Click the **Solver** button on the INSTED/CFD (2D) Mesh Generator Main dialog box.  
→ The INSTED/CFD (2D) Solver program is launched. The mesh generated from the Mesh Generation program is automatically loaded.
30. Click the **Load Project** button on the Main dialog box of the Solver program.
31. Type \*.\* in the filename section of the operating system 'Open File' dialog box that appears, and press Enter.  
→ A list of files in the current directory is displayed.
32. Locate the file 'squeeze.cfd' from the 'Samples' directory.
33. Click this file and press Enter.
34. Click the **Prob. Description** button on the Main dialog box and inspect the loaded parameters.
35. Click the OK button to dismiss the 'Prob. Description' dialog box.
36. Click the **Material Properties** button on the Main dialog box and inspect the loaded material properties. Note particularly the heat flux parameters.
37. Click the OK button to dismiss the 'Material Properties' dialog box.
38. Click the Solve button on the Main dialog box  
→ Program begins to generate a solution to the problem. The graphic screen is converted to one that displays the status of the solution process.
39. Wait for the solution to complete.

## Analyzing the Result

(Refer to the User's Manual for general details about INSTED/CFD (2D) Post-Processing.)

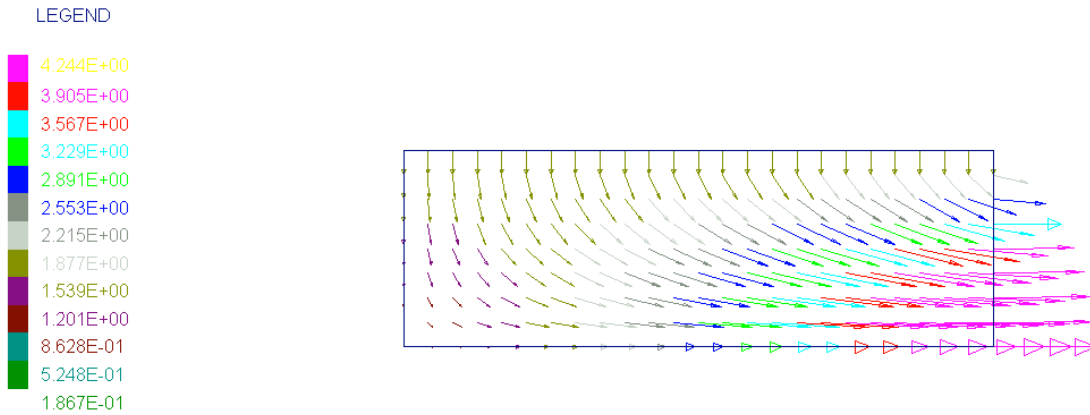
40. Click the **Post Processor** button on the INSTED/CFD (2D) Solver Main dialog box.  
→ The INSTED/CFD (2D) Postprocessor program is launched. The results from the
41. Click the **Display** button on the 'Controls' dialog box.  
→ The 'Display' dialog box appears.
42. Select the most recent time step for viewing from the 'Time Step' listbox.
43. Click the **Ok** button to dismiss the 'Display' dialog box.
44. View the Vector Plot for velocity.
45. Locate and open the file 'lola.out' in the working directory (\cfd for the current example).
46. Inspect and compare the values at selected nodal points.

**Table 6.1 Comparison of Results**

The velocities  $u(x)$  at  $y = 0$  obtained from INSTED are compared in the table below with those from the references.

x- coord	Series Solution (Nadai)	Penalty 9-node (Reddy)	Mixed Model 9-node (Reddy)	INSTED 12 x 4  Re = 0.001	Difference*
1.0	.7500	.7505	.7497	.7500	< 1 %
2.0	1.500	1.499	1.503	1.502	< 1 %
3.0	2.250	2.256	2.256	2.256	< 1 %
4.0	3.000	3.024	3.020	3.026	< 1 %
4.5	3.375	3.431	3.429	3.422	< 1 %
5.0	3.750	3.803	3.816	3.811	< 2 %
5.5	4.125	4.108	4.120	4.135	< 1 %
6.0	4.500	4.194	4.236	4.243	< 6 %

\* Relation to the exact solution



**Figure 5.2 Vector Plot of Velocity for Fluid Squeezed Between Parallel Plates**

**Reference:**

Reddy, J. N. & Gartling, D. K. (1994). The Finite Element Method in Heat Transfer and Fluid Dynamics. CRC Press, Boca Raton, Florida, USA. 69-70.

Nadia, A. 1963. Theory of Flow and Fracture of Solids, Volume II. McGraw Hill, New York.

TEST PROBLEM 6  
**NATURAL CONVECTION OF AIR IN A SQUARE CAVITY: A BENCHMARK  
 NUMERICAL SOLUTION**

**Problem Statement**

The problem being considered is that of the two-dimensional flow of a Boussinesq fluid of Prandtl number 0.71 in an upright square cavity of side  $L$  ( $L = 1$  in non-dimensional variables). Both velocity components are zero on all the boundaries. The horizontal walls are insulated, and the left and right vertical sides are at temperatures  $T_h$  and  $T_c$ , respectively, which are 1 and 0 in non-dimensional units.

The solutions to this problem, namely, the velocities, the temperature and the rate of heat transfer, are to be obtained for Rayleigh numbers,  $Ra = 10^4$ .

**CFD Type**

This problem will be solved as CFD Type 1; that is, using non-dimensional units for free convection.

**Physical and Computational Domain**

The physical and computational domain are shown below:

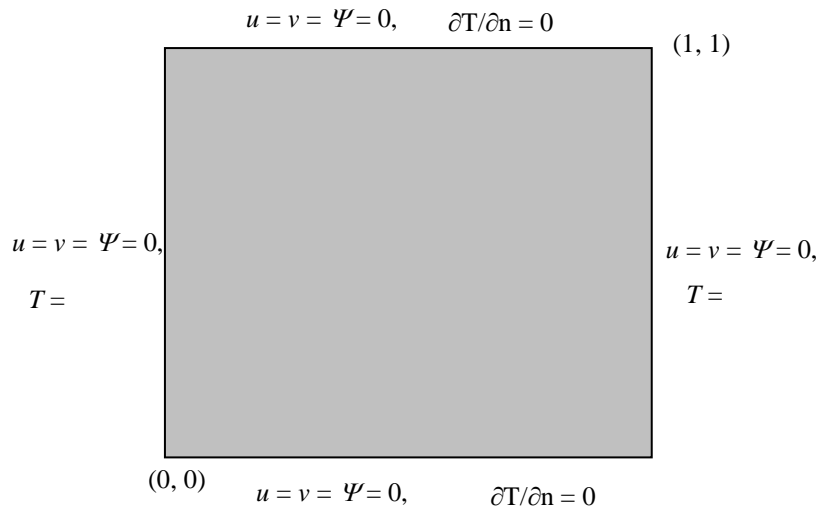


Figure 6.1 Physical Domain for Natural Convection Problem



## Governing Equations

The governing equations are the same as those presented in the main CFD manual for CFD Type 1: Free Convection. However, this problem will be solved in an inertial frame, the rotational terms in the equations have to be set to zero. That is:

$$\text{Re}_r = \text{Ra}_r = 0.$$

The volumetric heat source,  $H$ , should also be set to zero, and you should take the  $y$ -coordinate as pointing in the negative direction of the normal gravity vector. The governing equation can be written as

$$\begin{aligned}\nabla \cdot \mathbf{u} &= 0 \\ \frac{\partial u}{\partial t} + \mathbf{u} \cdot \nabla u &= -\frac{\partial p}{\partial x} + \text{Pr} \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^*)_x \\ \frac{\partial v}{\partial t} + \mathbf{u} \cdot \nabla v &= -\frac{\partial p}{\partial y} + \text{Pr} \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^*)_y + \text{Ra Pr} T \\ \frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T &= \nabla \cdot k \nabla T\end{aligned}$$

where  $(u, v)$  are the velocities in the  $x$ - and  $y$ -coordinate directions, and  $\eta$  and  $k$  are the non-dimensional value of the absolute viscosity and thermal conductivity respectively. The Prandtl number,  $Pr$ , is taken as 0.71 while a Rayleigh number of  $10^4$  is used. The mathematical symbols in these equations take on their usual meaning and are explained in greater detail in the Users Manual.

## Initial Condition

The default initial conditions in INSTED is used for this problem. That is,  $u(x, y, 0) = v(x, y, 0) = 0$ .

## Boundary Conditions

Two-dimensional flow of a Boussinesq fluid of Prandtl number 1, in an upright square cavity is described in non-dimensional terms by

$$0 \leq x \leq 1, \quad 0 \leq y \leq 1$$

with  $y$  vertically upwards. The problem assumes that both components of the velocity are zero on all boundaries and the boundaries are  $y = 0$  and  $y = 1$  are insulate,

$$\frac{\partial T}{\partial n} = 0,$$

and that  $T = 1$  at  $x = 0$  and  $T = 0$  at  $x = 1$ .

## Computational Grid

Two different mesh models are used - an 8 x 8 square mesh and a 10 x 10 boundary layer mesh with 4 layers, 0.3 depth and a packing factor of 1.04. The 8 x 8 square is consists of 64 elements and 289 nodes including the nodes at the center while the boundary layer mesh includes 81 elements and 361 nodes.

## Project Files

The files required to reproduce our results for this sample problem are:

Devahl1.mes	Input data for 8 x 8 mesh of model
Devahl2.mes	Input data for boundary layer mesh of model
Devahl.cfd	Input data for CFD solver

## Location of Project Files

The project files above are located in the directory \insted30\cfd\samples.

## Model Building and Mesh Generation

The mesh boundary for this problem is a square with counter-clockwise orientation. The rectangle modeling tool in INSTED will be used to produce the square. Details of mesh generation follow:

1. Start INSTED/CFD (2D) Mesh Generator.
2. Click the **New Project** button on the Main dialog box.
3. Click the **Limit** button on the 'Controls' dialog box.  
→ The 'Limits' dialog box appears.
4. Type in -0.2, 1.2, -0. 2, 1.2 for the lower  $x$ , higher  $x$ , lower  $y$  and higher  $y$  screen limits in the 'Limits' dialog box.
5. Click the OK button to dismiss this dialog box.

### To draw the square

6. Click the Rectangle tool button among the Modeling Tools that reside on the Main dialog box.  
→ Status bar displays the message: 'Pick 1<sup>st</sup> Corner Point'.
7. Type in 0,0 in the Input box and press Enter.  
→ Status bar displays the message: 'Pick 2<sup>nd</sup> Corner Point'
8. Type in 1, 1 in the Input box and press Enter.

### Defining number of elements per line

9. Click the **Line Properties** button on the Main dialog box,  
→ The 'Line Properties' dialog box appears.
10. Enter the values of 8, 8, 8, and 8 for the number of elements for lines 1, 2, 3, and 4 respectively. Press Enter after typing each value to move to the next line I the listbox.
11. Click the OK button to dismiss this dialog box.

### Defining the area to mesh

12. Click the **Define Area** button on the Main dialog box,  
→ The 'Area Definition' dialog box appears.
13. Click the **Add** button on this dialog box.  
→ An area listed as '1' is added in the 'Areas' listbox.
14. Click the first line of the 'Boundaries' listbox.

15. Type 1 and press Enter. This is meant to include boundary (made of Rectangle with lines 1,2,3, and 4) in this area,

16. Click the **OK** button to dismiss the 'Area Definition' dialog box.

#### Applying boundary conditions

17. Click the **Line BC** button on the Main dialog box,

→ Status bar displays message: 'Pick Side'.

18. Click the Line 1 on the graphics screen or type 1 and Press Enter in the Input Box

→ The 'Line BC' dialog box appears with the heading -"Boundary Condition, Side 1" (confirm from the title of the dialog box that data is being received for Line 1. This will not be the case if you accidentally selected some other line.).

19. Click the drop down list box for u-velocity and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.

20. Click the drop down list box for v-velocity and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.

21. Click the drop down list box for stream function and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.

22. Click the **OK** button to dismiss the dialog box.

23. Repeat the procedure (17-21) for Line 3.

24. Click the **Line BC** button on the Main dialog box,

→ Status bar displays message: 'Pick Side'.

25. Click the Line 4 on the graphics screen or type 4 and Press Enter in the Input Box

→ The 'Line BC' dialog box appears with the heading -"Boundary Condition, Side 4" (confirm from the title of the dialog box that data is being received for Line 4. This will not be the case if you accidentally selected some other line.).

26. Click the drop down list box for u-velocity and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.

27. Click the drop down list box for v-velocity and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.

28. Click the drop down list box for stream function and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.

29. Click the drop down list box for temperature and select a Dirichlet boundary condition from the list and enter a value of 1.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.

30. Click the radio button 'Nusset No. BC on Side' to indicate that you wish to calculate the Nusset number along this line.

31. Click the **OK** button to dismiss the dialog box.

32. Repeat the procedure (24-29) for Line 2 setting Dirichlet conditions for temperature with a value of 0.0 while executing step 29.

33. Click the **OK** button to dismiss the dialog box.

#### Mesh and Confirm Mesh Results

34. Click the **Mesh** button on the Main dialog box,

→ A mesh of the internal of our loop of lines is displayed on the 'Graphics' dialog box.

35. Click the **Display** button on the 'Controls' dialog box.

36. From the listbox, select 'Dirichlet, Temp'

37. Click the **Ok** button on this dialog box to view nodes with Dirichlet temperature boundary conditions.
38. Click the **Refresh** button on the 'Controls' dialog box to return the screen to its original condition.

**Note:** You should have the same result as if you loaded the file `\insted30\cfd\samples\devahl1.mes`. You may also choose to save your current work. Refer to the sections 'Loading a file' and 'Saving a file' in the first chapter of the Users Manual.

### Obtaining a Solution to the Problem

39. Click the **Solver** button on the INSTED/CFD (2D) Mesh Generator Main dialog box.  
→ The INSTED/CFD (2D) Solver program is launched. The mesh generated from the Mesh Generation program is automatically loaded.
  40. Click the **Load Project** button on the Main dialog box of the Solver program.
  41. Type \*.\* in the filename section of the operating system 'Open File' dialog box that appears, and press Enter.  
→ A list of files in the current directory is displayed.
  42. Locate the file 'devahl1.cfd' from the 'Samples' directory.
  43. Click this file and press Enter.
  44. Click the **Prob. Description** button on the Main dialog box and inspect the loaded parameters.
  45. Click the OK button to dismiss the 'Prob. Description' dialog box.
  46. Click the **Material Properties** button on the Main dialog box and inspect the loaded material properties. Note particularly the heat flux parameters.
  47. Click the OK button to dismiss the 'Material Properties' dialog box.
- Selecting Sample Points for History Data
48. Click the **Select Points** button on the Main dialog box.
  49. Select about eight nodal points by clicking the points on the graphics display
  50. Click the **Stop** button on the Main dialog box to stop selecting points
  51. Click the Solve button on the Main dialog box  
→ Program begins to generate a solution to the problem. The graphic screen is converted to one that displays the status of the solution process.
  52. Wait for the solution to complete.

### Analyzing the Result

(Refer to the User's Manual for general details about INSTED/CFD (2D) Post-Processing.)

53. Click the **Post Processor** button on the INSTED/CFD (2D) Solver Main dialog box.  
→ The INSTED/CFD (2D) Postprocessor program is launched. The results from the
54. Click the **Display** button on the 'Controls' dialog box.  
→ The 'Display' dialog box appears.
55. Select the most recent time step for viewing from the 'Time Step' listbox.
56. Click the **Ok** button to dismiss the 'Display' dialog box.
57. View the Vector plot for velocity.
58. View the Contour plots for velocities, temperature, stream function, and pressure.
59. View the time-history data for the selected sample points.
60. Locate and open the file 'lola.out' in the working directory (\cfd for the current example).
61. Inspect and compare the values at selected nodal points.

### Generating the Boundary Layer Mesh

62. Click the **Line Properties** button on the Main dialog box,  
→ The 'Line Properties' dialog box appears.
63. Click the 'B.L.' listbox for all four lines that form the boundary of the object.  
→ The listbox should now have the text 'YES' on every line.
64. Click the OK button to dismiss this dialog box.

### Specifying Boundary Layer Parameters

65. Click the Mesh Control button on the Main dialog box.  
→ The 'Mesh Control' dialog box appears.
66. Click the radio button 'Boundary Layer Mesh' to indicate that you wish to generate a boundary layer mesh.
67. Enter the values 4, 0.3 and 1.04 in the input boxes for the number of layers, depth, and packing factor.
68. Click the OK button to dismiss this dialog box.
69. Repeat steps 34 through 61 for this mesh.

Table 6.1 Comparison of Results

	Bench Mark Results	INSTED Results	Difference	INSTED file
$ \Psi _{\max}$	5.071	5.074	< 1 %	Psimax.out
$U_{\max}$	16.178	16.2	< 1 %	Lola.out
$V_{\max}$	19.617	19.6	< 1 %	Lola.out
$Nu$ at left	2.238	2.23	< 1 %	Numean.out
$Nu_{\max}$	3.528	3.53	< 1 %	Nusset.out

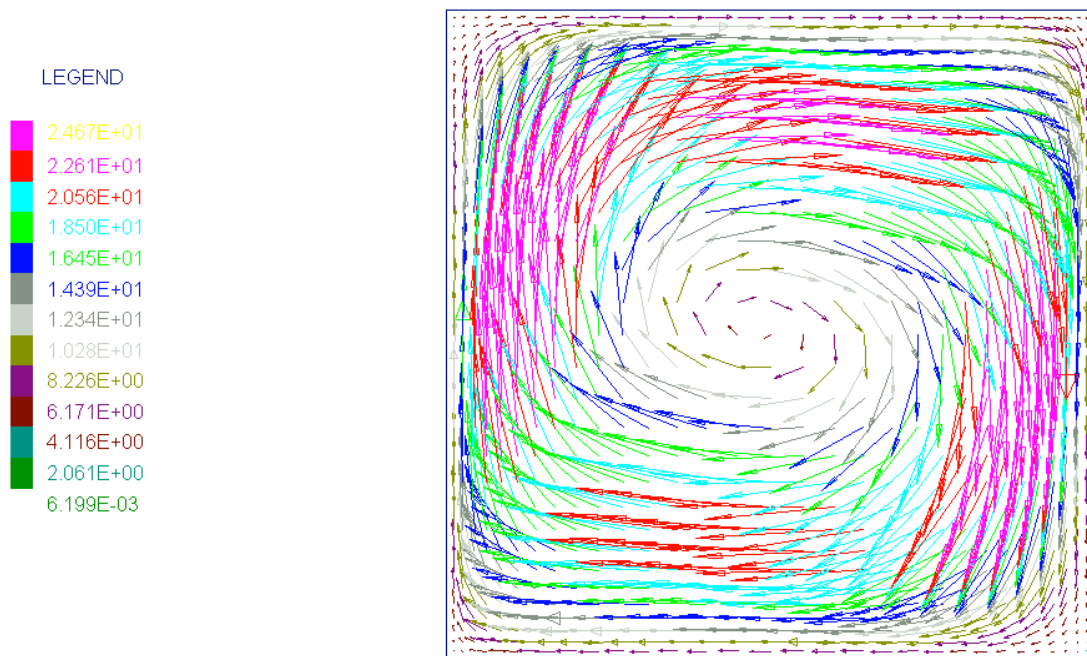


Figure 6.2 Vector Plot of Velocity for the Natural Convection Problem

## What's Next?

After reproducing the results for this test problem, go through the exercise again but this time, change the input data in the project file to study the effect on the solution. For example, during mesh generation, you could change the number of elements in each of the I and J directions. During analysis, you could locate the thermophysical properties, dimensionless parameters, solution parameters, etc. and change them to study the effect on the solutions.

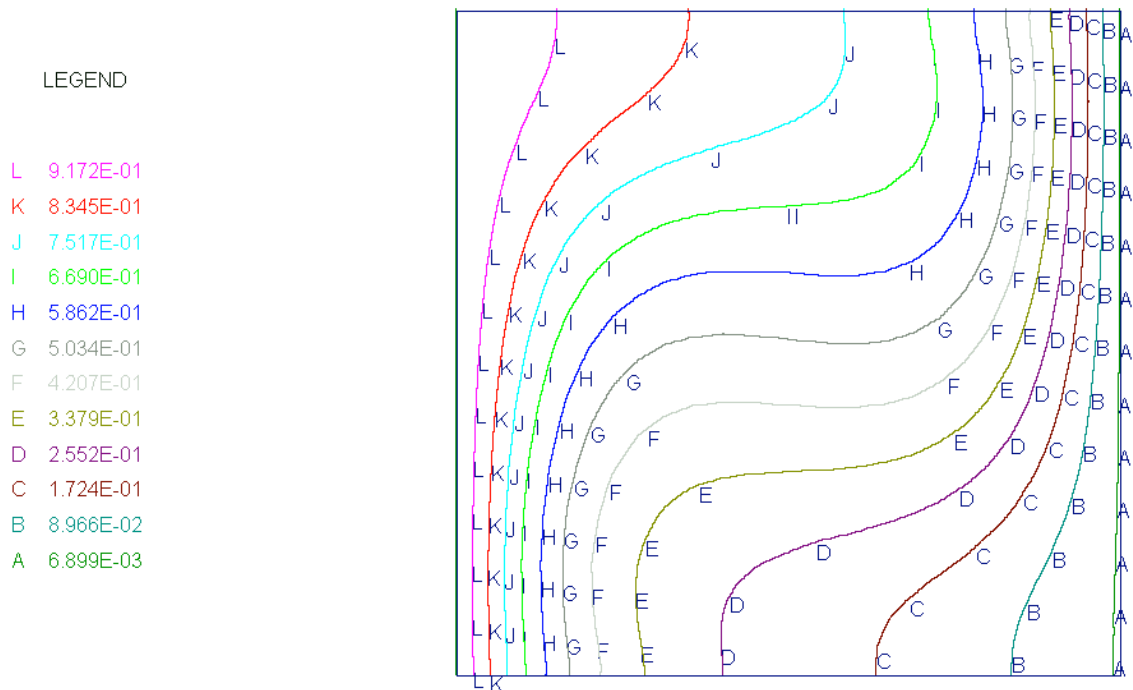


Figure 6.3 Contour Plot of Temperature for the Natural Convection Problem

## Reference:

Davis, G. de Vahl. (1983). Natural Convection in a Square Cavity. A comparison Exercise. *Int. J. Numerical Methods in Fluids*. Volume 3. 227-248.

Davis, G. de Vahl. (1983). Natural Convection in a Square Cavity. A Benchmark Numerical Solution. *Int. J. Numerical Methods in Fluids*. Volume 3. 249-264.

## TEST PROBLEM 7

### NATURAL CONVECTION TRANSPORT OF TEMPERATURE AND TWO NON-REACTING SPECIES

#### Problem Statement

The problem being considered is that of the two-dimensional flow of a Boussinesq fluid of Prandtl number 0.71 in an upright square cavity of side  $L$  ( $L = 1$  in non-dimensional variables). Both velocity components are zero on all the boundaries. The horizontal walls are insulated, and the left and right vertical sides are at temperatures  $T_h$  and  $T_c$ , respectively, which are 1 and 0 in non-dimensional units.

The natural convection flow is used to transport two chemical species (of different diffusivities) within the box. It is assumed that buoyancy only results from temperature differences. The solution at  $Ra = 10^4$  is desired.

#### CFD Type

This problem will be solved as a CFD Type 1 posing the solution in the non-dimensional units.

#### Physical and Computational Domain

The physical domain is as shown in Figure 6.1 except that we now have to include passive scalars which we assume have the same boundary conditions as temperature.

#### Governing Equations

The governing equations are the same as those presented in the main CFD manual for CFD Type 1: Free Convection. However, this problem will be solved in an inertial frame, the rotational terms in the equations have to be set to zero. That is,

$$Re_r = Ra_r = 0.$$

The volumetric heat source,  $H$ , should also be set to zero, and you should take the  $y$ -coordinate as pointing in the negative direction of the normal gravity vector. The governing equation can be written as

$$\begin{aligned}\nabla \cdot \mathbf{u} &= 0 \\ \frac{\partial u}{\partial t} + \mathbf{u} \cdot \nabla u &= -\frac{\partial p}{\partial x} + Pr \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^*)_x \\ \frac{\partial v}{\partial t} + \mathbf{u} \cdot \nabla v &= -\frac{\partial p}{\partial y} + Pr \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^*)_y + Ra Pr T \\ \frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T &= \nabla \cdot k \nabla T\end{aligned}$$

$$P_{11} \left( \frac{\partial \phi_1}{\partial t} + \mathbf{u} \cdot \nabla \phi_1 \right) = \nabla \cdot (P_{12} \nabla \phi_1) + P_{13}$$

$$P_{21} \left( \frac{\partial \phi_2}{\partial t} + \mathbf{u} \cdot \nabla \phi_2 \right) = \nabla \cdot (P_{22} \nabla \phi_2) + P_{23}$$

where  $(u, v)$  are the velocities in the coordinate directions, and  $\eta$  and  $k$  are the non-dimensional value of the absolute viscosity and thermal conductivity respectively. The parameters  $\eta$  and  $k$  take on the values of 1 for the test problem.  $\phi_1$  and  $\phi_2$  represent the concentrations of the two species to be calculated.  $P_{ij}$  ( $i = 1, 2; j = 1, 2, 3$ ) are the three parameters (index  $j$ ) for each of the two scalar equations (index  $i$ ).

For this sample problem,

$$P_{11} = P_{21} = 1$$

$$P_{12} = 25; P_{22} = 1$$

$$P_{13} = P_{23} = 0$$

The Prandtl number,  $Pr$ , is taken as 0.71 while the Rayleigh number is  $10^4$ . The mathematical symbols in these equations take on their usual meaning.

### Initial Condition

The default initial conditions in INSTED is used for this problem. That is,  $u(x, y, 0) = v(x, y, 0) = \phi_1(x, y, 0) = \phi_2(x, y, 0) = 0$ .

### Boundary Conditions

Two-dimensional flow of Boussinesq fluid of Prandtl number 0.71, in an upright square cavity is described in non-dimensional terms by

$$0 \leq x \leq 1, \quad 0 \leq y \leq 1$$

with  $y$  vertically upwards. The problem assumes that both components of the velocity are zero on all boundaries and the boundaries at  $y = 0$  and  $y = 1$  are insulated,

$$\frac{\partial T}{\partial n} = 0,$$

and that  $T = 1$  at  $x = 0$  and  $T = 0$  at  $x = 1$ . The boundary conditions on the scalars are the same as those for temperature.

### Computational Grid

An 8 x 8 square mesh of 9-node quadrilateral elements will be used. The total number of 64 elements is and the total number of nodes including those at the centroid of elements is 289.



## Project Files

The files required to reproduce our results for this sample problem are:

devscala.mes	Input data for 8 x 8 mesh of model
devscala.cfd	Input data for CFD solver

## Location of Project Files

The project files above are located in the directory \insted30\cfd\samples.

## Model Building and Mesh Generation

The mesh boundary for this problem is a square with counter-clockwise orientation. The rectangle modeling tool in INSTED will be used to produce the square. Details of mesh generation follow:

1. Start INSTED/CFD (2D) Mesh Generator.
2. Click the **New Project** button on the Main dialog box.
3. Click the **Limit** button on the 'Controls' dialog box.  
→ The 'Limits' dialog box appears.
4. Type in -0.2, 1.2, -0.2, 1.2 for the lower x, higher x, lower y and higher y screen limits in the 'Limits' dialog box.
5. Click the OK button to dismiss this dialog box.

### To draw the square

6. Click the Rectangle tool button among the Modeling Tools that reside on the Main dialog box.  
→ Status bar displays the message: 'Pick 1<sup>st</sup> Corner Point'.
7. Type in 0,0 in the Input box and press Enter.  
→ Status bar displays the message: 'Pick 2<sup>nd</sup> Corner Point'
8. Type in 1, 1 in the Input box and press Enter.

### Defining number of elements per line

9. Click the **Line Properties** button on the Main dialog box,  
→ The 'Line Properties' dialog box appears.
10. Enter the values of 8, 8, 8, and 8 for the number of elements for lines 1, 2, 3, and 4 respectively. Press Enter after typing each value to move to the next line in the listbox.
11. Click the OK button to dismiss this dialog box.

### Defining the area to mesh

12. Click the **Define Area** button on the Main dialog box,  
→ The 'Area Definition' dialog box appears.
13. Click the **Add** button on this dialog box.  
→ An area listed as '1' is added in the 'Areas' listbox.
14. Click the first line of the 'Boundaries' listbox.
15. Type 1 and press Enter. This is meant to include boundary (made of Rectangle with lines 1,2,3, and 4) in this area,
16. Click the **OK** button to dismiss the 'Area Definition' dialog box.

### Entering the Number of Scalars

17. Click on the Scalars button on the Main dialog box.  
→ The status bar displays the message - "Enter the number of scalars"
18. Enter a value of 2 in the Input box and press Enter.

### Applying boundary conditions

19. Click the **Line BC** button on the Main dialog box,  
→ Status bar displays message: 'Pick Side'.
20. Click the Line 1 on the graphics screen or type 1 and Press Enter in the Input Box  
→ The 'Line BC' dialog box appears with the heading -"Boundary Condition, Side 1" (confirm from the title of the dialog box that data is being received for Line 1. This will not be the case if you accidentally selected some other line.).
21. Click the drop down list box for u-velocity and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
22. Click the drop down list box for v-velocity and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
23. Click the drop down list box for stream function and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
24. Click the **OK** button to dismiss the dialog box.
25. Repeat the procedure (19-23) for Line 3.
26. Click the **Line BC** button on the Main dialog box,  
→ Status bar displays message: 'Pick Side'.
27. Click the Line 2 on the graphics screen or type 2 and Press Enter in the Input Box  
→ The 'Line BC' dialog box appears with the heading -"Boundary Condition, Side 2" (confirm from the title of the dialog box that data is being received for Line 2. This will not be the case if you accidentally selected some other line.).
28. Click the drop down list box for u-velocity and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
29. Click the drop down list box for v-velocity and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
30. Click the drop down list box for stream function and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
31. Click the drop down list box for temperature and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
32. Click the drop down list box for scalar 1 (Scalar 1 is the scalar that is selected by default for boundary condition input) and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
33. Change the selected scalar to Scalar 2 by clicking the drop down arrow of the Scalar listbox and clicking on Scalar 2.
34. Click the drop down list box for Scalar 2 and select a Dirichlet boundary condition from the list and enter a value of 0.0 in the accompanying Input box that appears as soon as Dirichlet condition is selected.
35. Click the **OK** button to dismiss the dialog box.
36. Repeat the procedure (26-35) for Line 4 setting Dirichlet conditions for temperature with a value of 1.0 while executing step 29.

### Mesh and Confirm Mesh Results

37. Click the **Mesh** button on the Main dialog box,

- A mesh of the internal of our loop of lines is displayed on the 'Graphics' dialog box.
- 38. Click the **Display** button on the 'Controls' dialog box.
- 39. From the listbox, select 'Dirichlet, Temp'
- 40. Click the **Ok** button on this dialog box to view nodes with Dirichlet temperature boundary conditions.
- 41. Click the **Refresh** button on the 'Controls' dialog box to return the screen to its original condition.

**Note:** You should have the same result as if you loaded the file 'devscala.mes' in the Samples directory of the \insted30\cfd\devscala.mes. You may also choose to save your current work. Refer to the sections 'Loading a file' and 'Saving a file' in the first chapter of the Users Manual.

### Obtaining a Solution to the Problem

- 42. Click the **Solver** button on the INSTED/CFD (2D) Mesh Generator Main dialog box.
  - The INSTED/CFD (2D) Solver program is launched. The mesh generated from the Mesh Generation program is automatically loaded.
- 43. Click the **Load Project** button on the Main dialog box of the Solver program.
- 44. Type \*.\* in the filename section of the operating system 'Open File' dialog box that appears, and press Enter.
  - A list of files in the current directory is displayed.
- 45. Locate the file 'devscala.cfd' from the 'Samples' directory.
- 46. Click this file and press Enter.
- 47. Click the **Prob. Description** button on the Main dialog box and inspect the loaded parameters.
- 48. Click the OK button to dismiss the 'Prob. Description' dialog box.
- 49. Click the **Material Properties** button on the Main dialog box and inspect the loaded material properties. Note particularly the heat flux parameters.
- 50. Click the OK button to dismiss the 'Material Properties' dialog box.

### Analyzing the Result

(Refer to the User's Manual for general details about INSTED/CFD (2D) Post-Processing.)

- 51. Click the **Post Processor** button on the INSTED/CFD (2D) Solver Main dialog box.
  - The INSTED/CFD (2D) Postprocessor program is launched. The results from the
- 52. Click the **Display** button on the 'Controls' dialog box.
  - The 'Display' dialog box appears.
- 53. Select the most recent time step for viewing from the 'Time Step' listbox.
- 54. Click the **Ok** button to dismiss the 'Display' dialog box.
- 55. View the Vector plot for velocity.
- 56. View the Contour plots for velocities, temperature, stream function, scalars 1 & 2, and pressure.
- 57. Locate and open the file 'lola.out' in the working directory (\cfd for the current example).
- 58. Inspect and compare the values at selected nodal points.

## Results

The contour maps for temperature, and the three scalars are shown in Figures 7.1 through 7.3. Note the similarity in the results for temperature and scalar 2. Since the flow is purely thermally driven, some differences, no matter how small should exist between scalar 2 and temperature because of the closer tie between temperature and the velocity. Finally, note that, because of the coarse grid used, unphysical solutions were observed. While scalar 1 have a high diffusivity to dissipate unphysical results, scalar 2 and the temperature do show the unphysical results because of low diffusivities.

## What's Next?

After reproducing the results for this test problem, go through the exercise again but this time, change the input data in the project file to study the effect on the solution. For example, during mesh generation, you could change the number of elements in each of the I and J directions. During analysis, you could locate the thermophysical properties, dimensionless parameters, solution parameters, etc. and change them to study the effect on the solutions.

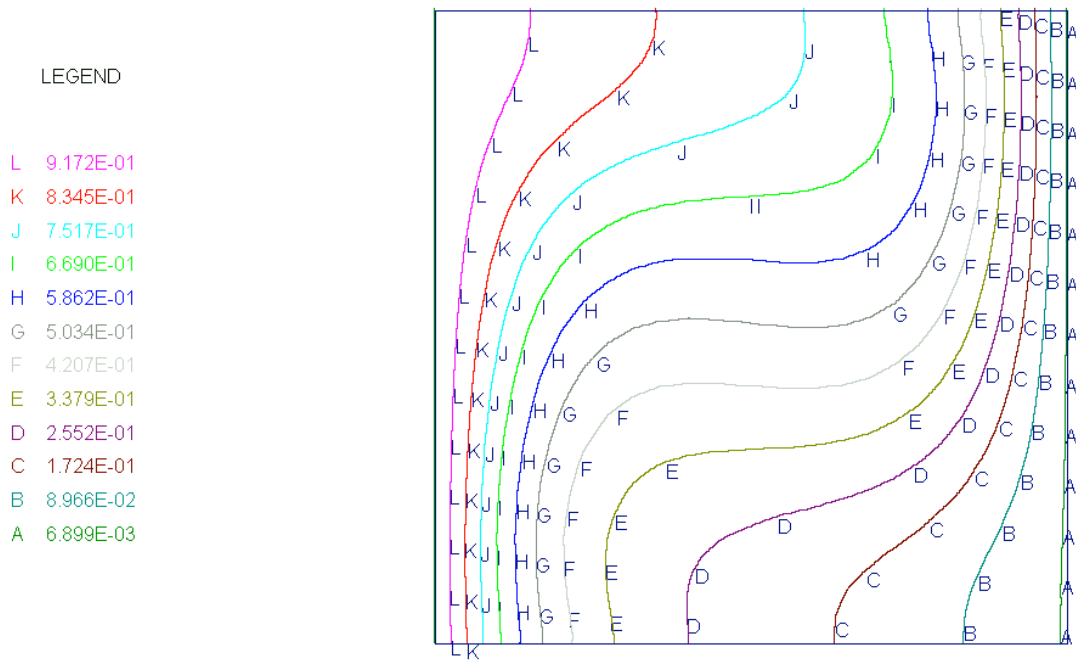


Figure 7.1 Contour Plot of Temperature for the Test Problem on the Transport of Temperature and Species By Natural Convection in a Cavity for  $Ra = 10^4$ .

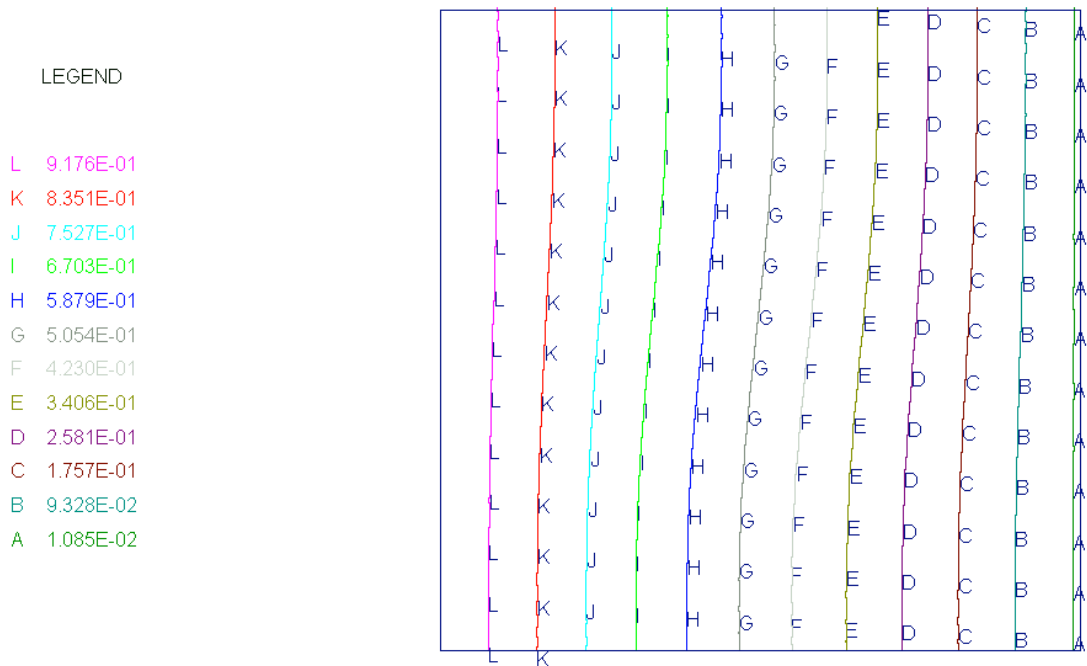


Figure 7.2 Contour Plot of Scalar 1 for the Test Problem on the Transport of Temperature and Species By Natural Convection in a Cavity for  $Ra = 10^4$ .

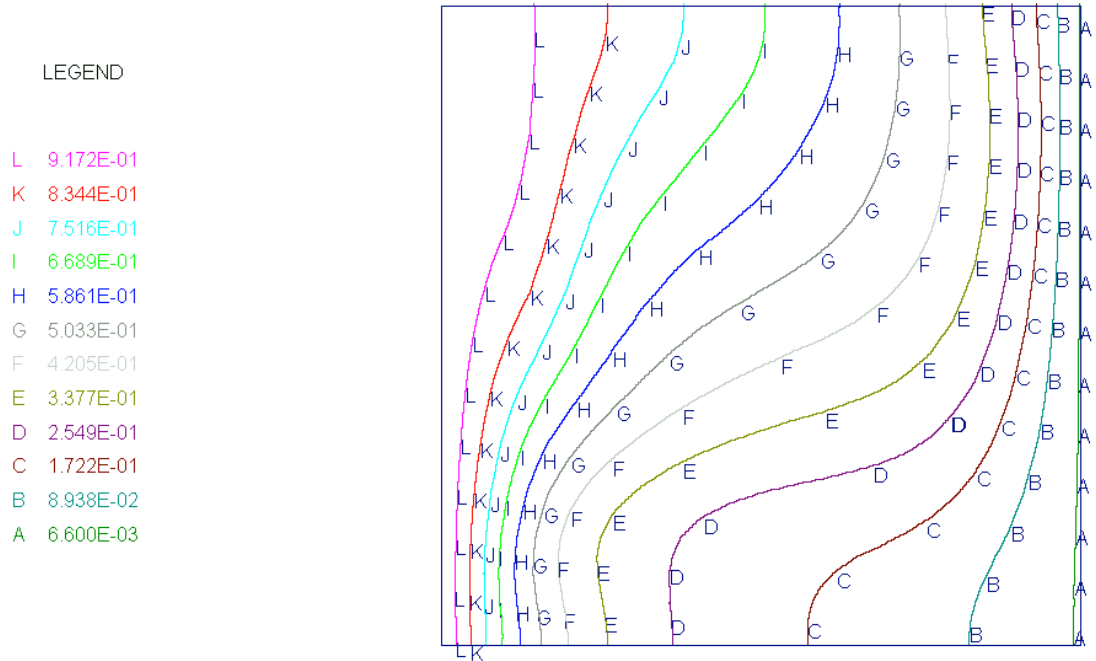


Figure 7.3 Contour Plot of Scalar 2 for the Test Problem on the Transport of Temperature and Species By Natural Convection in a Cavity for  $Ra = 10^4$ .

**Reference:**

Davis, G. de Vahl. (1983). Natural Convection in a Square Cavity. A comparison Exercise. Int. J. Numerical Methods in Fluids. Volume 3. 227-248.

Davis, G. de Vahl. (1983). Natural Convection in a Square Cavity. A Benchmark Numerical Solution. Int. J. Numerical Methods in Fluids. Volume 3. 249-264.