Chapter 1
INTRODUCTION

INSTED®/CFD (2D) consists of three programs - INSTED Mesh Generator, Solver, and Post-processor. All three programs are integrated in such a way that a complete CFD analysis can be performed, from model building, solving, to visualization and result presentation. You can also use each of the programs separately. For instance, the mesh generator can produce the meshes for other programs, the solver can load previously meshed problems for analysis, while the post-processor can be used to display the results of previous calculations.

This manual consists of the description of all three programs. The second chapter provides a description of the mesh generator, the third and fourth chapters focus on the solver and post-processor, respectively.

1.1 Restart Features of INSTED Programs

All INSTED programs have an autosave feature that automatically saves the current analysis to a restart file on exiting the program. When you restart the program, the previous session's work is automatically loaded. In order to start a new project, the **New Project** button on the Main dialog box should be selected. This initializes all input data and prepares all dialog boxes to receive new data.

1.2 Saving a File

INSTED Mesh Generator allows sessions of INSTED to be saved for subsequent analysis at another time. To save a current session, follow these steps:

- Press the **Save As** button on the Main dialog box.
- Follow general Windows procedure for saving a file.

You can also periodically save your work, by clicking the **Save Project** button located on the Main dialog box. This causes the file for the current session to be loaded with the most recent data.

1.3 Loading a File

To load a previously saved file, follow these steps:

▪ Press the **Load Project** button on the Main dialog box.
→ An operating system 'Open' dialog box appears.
▪ Locate the directory in which the file you wish to load resides.
▪ Type *.* and press enter.
→ A list of files in the current directory is shown.
▪ Double click on the file you wish to load to select the file.

1.4 Files Generated by INSTED CFD Programs

The files associated with the INSTED/CFD (2D) Mesh Generator are as follows:

File	Purpose	Created by	Where Used
*.mes	Model data	User (Old Project)	Mesh Generator
Mesh.dat	Mesh data output	Program	CFD Analysis
Bound.dat	Boundary condition data	Program	CFD Analysis
Xnuslt.dat	Nusset number boundary data	Program	CFD Analysis
Generl.dat	General mesh info.	Program	Not used
Output.dat	Mesh data and output	Program	Not used

The files associated with the INSTED/CFD (2D) Solver program are as follows:

File	Purpose	Created by	Where Used
*.cfd	Solver project data	User (Old Project)	CFD Analysis

Solutn.out	Some results	Program	deactivated
Psimax.out	Stream function data	Program	FYI
Nusset.out	Nusset number data	Program	FYI
Numean.out	Mean nusset number	Program	FYI
lola.out	Main output data	Program	FYI
Minmax.out	Min., max. values	Program	FYI
Arbtr.dat	For initial conditions	User/Program	CFD Analysis
Plotf.sd	Main plot file	Program	Post-processor
Nusplt.dat	Nusset number plot file	Program	Post-processor
Htflxmat.dat	Heat flux data	User	CFD Analysis
Plt1.dat	Node data, 4-node elements	User	Post-processor
Plt2.dat	Element connectivity data, 4-node elements	User	Post-processor
Uhist*.out Vhist*.out Thist*.out	History plot (for sample points)	Program	Post-processor

Chapter 2
INSTED®/CFD (2D) MESH GENERATOR

2.1 Model Building with INSTED Mesh Generator

INSTED/CFD (2D) Mesh Generator provides the tools for building CFD models. Its automatic mesh generation feature ensures that models can be built quickly. It also supports multi-domain meshing, allowing the user to build models or multi-material models in blocks.

As with all model building programs, the engineer begins by planning ahead for the process. Some of the tasks involved in the planning stage include:

- Setting up the coordinate system, sketching the physical domain, and ensuring that the coordinates of points in the domain are known before hand.
- Examining the physical domain for symmetry and defining the computational domain accordingly.
- Identifying regions of high gradients, where mesh will be concentrated, as well as low gradient regions, where a coarse mesh might be more appropriate.
- Estimating the number of nodes and elements and ensuring that they are within limits set by the code.
- Identifying the types of boundary conditions that will be appropriate at all points on the boundary of the domain, including at flow inlet, flow exit, solid walls, stress-free boundaries, symmetry lines, sides for specified temperature, sides with adiabatic temperature conditions, and sides for specified heat flux. You should also be able to identify possible reference streamlines and calculate the distribution of stream function across the inlet.

2.1.1 Parts of a model

In INSTED, models consist of points, lines, boundaries and areas, in that hierarchical order. A line will be formed by two or more points. For instance, the basic line will be formed by two points, a circular arc may be formed by a start point, end point and a circle center, while a spline may be formed by three points on the line. In INSTED, all the created entities - points, lines, boundaries and areas - have a unique entity number.

Boundaries are a collection of lines. In all cases, a boundary should form a closed loop. As a result, all boundaries are composed of more than one line except if the line is a circle. Areas are a collection of boundaries.

INSTED provides the engineer with a simple and quick method for CFD modeling. The program features a powerful advancing front-technique that could reduce the need to create patches for models.
Mesh generation operations are only performed on areas. However, in order to mesh an area, the area must conform to certain simple rules. These rules are described in the next section.

2.2 Mesh Generation with INSTED

In order to mesh an object in INSTED, we need to have an area. A valid area must consist of an external boundary or a group of boundaries connected completely to form one continuous loop of lines. An external boundary will have a counter-clockwise orientation. **In order words, the lines that make up the collection of boundaries will be connected and counted in a counter-clockwise fashion**. This external boundary may contain one or several internal boundaries. **Internal boundaries must have a clockwise orientation.**

2.2.1 Steps involved in mesh generation

The steps for generating the mesh for your computational domain are as follows:

- Define the lower and upper limits of a rectangle containing your model (Section 2.4).
- Create the lines needed to form all the boundaries of your model (Section 2.5).
- Format the lines - give the number of elements required on the side or define the line as a boundary layer line (Section 2.6).
- Define the boundaries as a sequence of lines (Section 2.7).
- Define the areas as regions enclosed by boundaries (Section 2.8).
- Provide the boundary conditions for the model (Section 2.12).
- If you wish to create a boundary layer mesh, provide boundary layer data. That is, give depth of layer, number of elements in layer, and stretching factor (Section 2.14).
- Mesh the areas (Section 2.15).

▪ Perform post mesh generation activities such as assigning boundary conditions to nodes (Section 2.16). (Note: The specification of nodes for storage of history data is performed in the solver and is discussed in Section 4.6.)
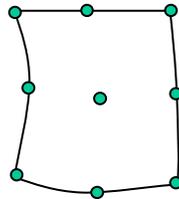
2.2.2   Problem size

For the PC version of INSTED/CFD (2D), the following limits are currently imposed on the problem size. These limits can be increased at customer's request.
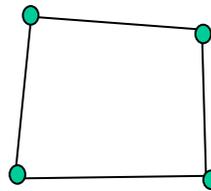
1. Number of nodal points - 8,000.
2. Number of 9-node elements - 3000.
3. Number of 4-node elements - 12,000.
4. Number of nodes for boundary condition for the variables u, v, T, Ψ, and scalars - 1000.
5. Number of nodes for heat flux boundary conditions - 500.
6. Number of boundary nodes for Nusselt number calculation - 500.

2.2.3   Element used in INSTED/CFD (2D)

The element used in the INSTED/CFD (2D) is the 9-node quadrilateral element Fig. 2.1(a). for the Navier-Stokes equations, studies have proved the superiority of this element to the standard 4-node element shown in Fig. 2.1(b) below. This element is shown below.

(a) Nine-node element used in INSTED/CFD   (b) Four-node element

Figure 2.1 Element used in INSTED/CFD (2D)

2.2.4   The backward-facing step illustration

In this manual, flow in a backward facing step will be used to illustrate the mesh generation procedure. The physical model is shown below.
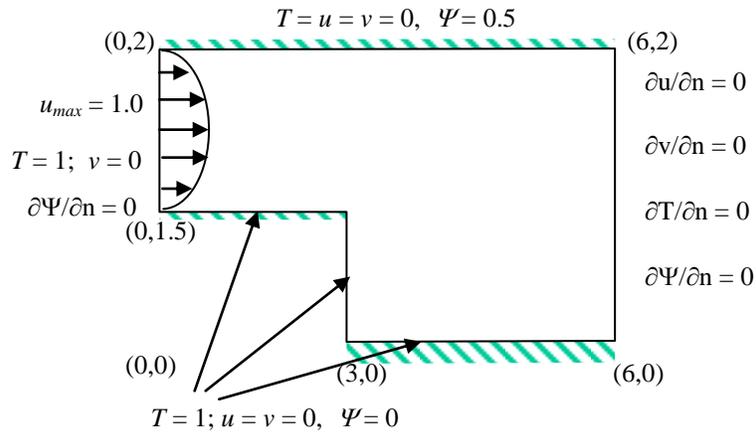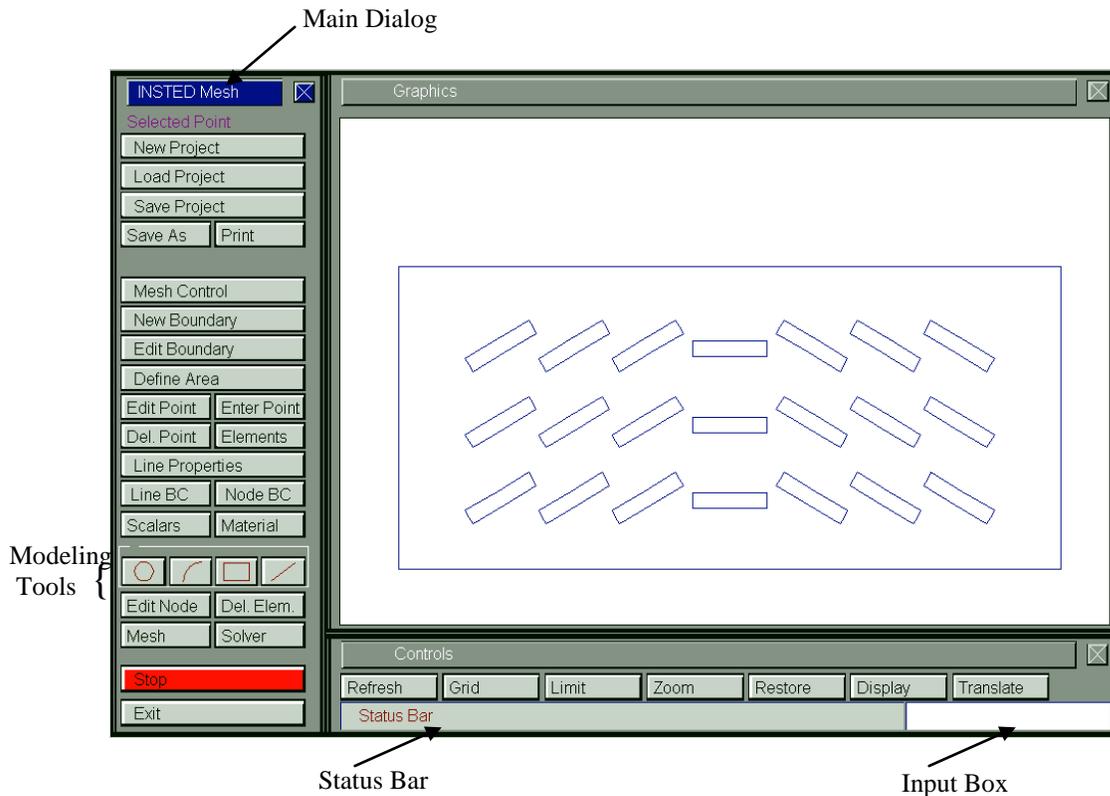

Figure 2.2 The Backward-Facing Step Problem

In Fig. 2.2, u, v, T, and Ψ are, respectively, the x-component of velocity, y-component of velocity, temperature, and stream function.

7/ 84

2.3 The INSTED Mesh Generation Interface

The elements of the graphics user interface (GUI) for INSTED/CFD (2D) are described briefly in this section.

The Status Bar

This is located at the bottom of the Controls dialog box. This bar displays interactive instructions for completing the tasks selected in the process of building a model.

The Input Box
This is the section of the screen for keyboard input of required numerical data. For instance, when creating new points, the coordinates of the points may be entered in this box with the keyboard instead of clicking the position of the points on the screen with a mouse.

2.3.1 Modeling tools

These include circle, circular arc, rectangle, and a line tool. On clicking any of the icons for these tools, instructions are provided on the status bar to prompt you for the required action. Each of the modeling tools creates a boundary. Each boundary is made up of one or more lines

Line Tool
You will be prompted to click (or enter the coordinate in the Input Box) the position of the start and end points of the line. The line tool creates a single line. It also creates a new boundary and assigns the lines to the boundary.

Rectangle Tool
You will be prompted to click (or enter) the position of two diagonally opposite corner points representing the limits of the rectangle. The rectangle tool creates four lines. It also creates a new boundary and assigns the lines to the boundary.

Arc Tool
You will be prompted to enter the center point of the circle and the start and end points of the circular arc. The arc tool creates a single line. It also creates a new boundary and assigns the line to the boundary.
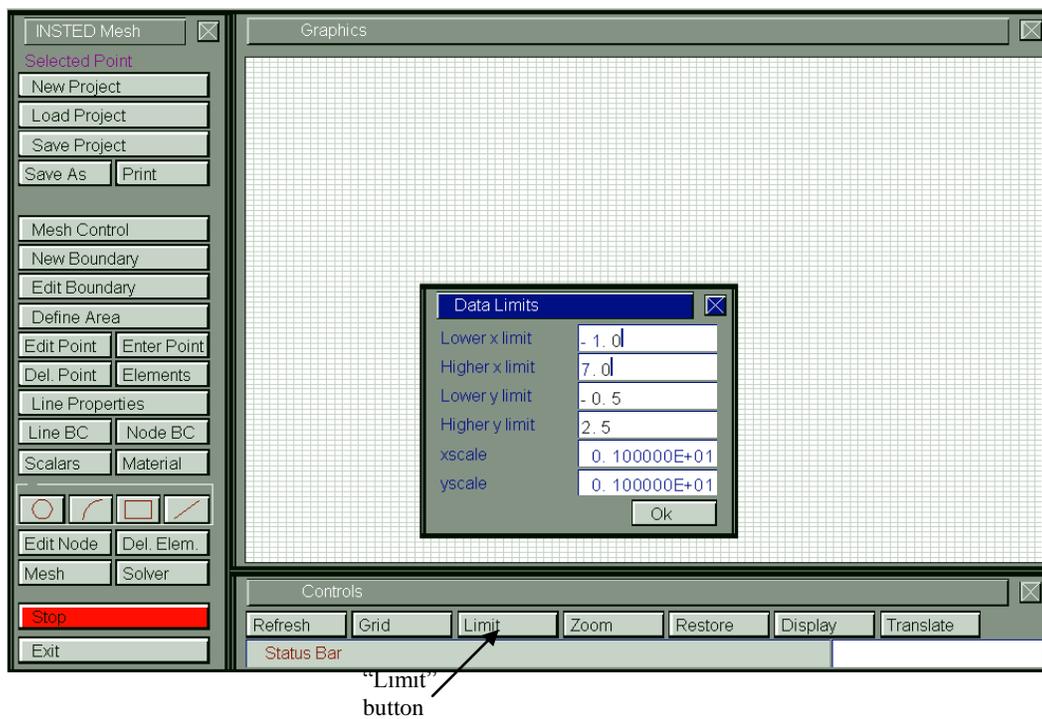
Circle Tool
You will be prompted for the center point of the circle and any point on the circumference. The circle tool creates a single line. It also creates a new boundary and assigns the line to the boundary.

2.4 Defining Data Limits for the Screen

The Data Limits dialog box allows you to define the extent of the data (i.e. the minimum and maximum in the *x*- and *y*- coordinate directions). This is required in order to format the graphic screen to contain the model. To access the Data Limit dialog, click **Limit** on the **Controls** dialog box.

The sample backward-facing step problem spans from (0, 0) to (6, 2) as shown in the physical model in section 2.2. Consequently, we will format the screen to receive data using the limit (-1, -0.5) to (7, 2.5). This is shown in the figure below. This limit allows an extra padding of unity at each boundary.

"Limit" button

2.5 Creating Lines for Mesh Generation

Lines are created from two points - a start point and an end point. For a circle, both the start and the end point are the same. All lines have a unique number, which is assigned according to the order in which the lines are generated. **The line number is usually indicated on the graphic display approximately at the center of the line.** Lines are created with the Modeling tools and the New Boundary tool. The New Boundary tool creates a closed or unclosed polygon. This tool also creates lines for the polygons and a single boundary to which the lines belong.

2.5.1 Creating a closed polygon with the 'New Boundary' tool

1. Click the **New Boundary** button on the Main dialog box.
→   You will be prompted to enter the first point.
2. Click on the position of the first point on the screen or enter its coordinate in the Input box.
→   You will be prompted to enter the second point.
3. Repeat step 2 for all the points in the polygon.
→   The process will automatically stop when you click or enter the coordinates of the starting point.

2.5.2 Creating an unclosed polygon with the New Boundary tool

1. Click the **New Boundary** button on the Main dialog box.
→   You will be prompted to enter the first point.
2. Click on the position of the first point on the screen or enter its coordinate in the Input box.
→   You will be prompted to enter the second point.
3. Repeat step 2 for all the points in the polygon.
4. Press the **Stop** button on the Main dialog to stop entering points.

For the backward-facing step problem, we shall create a closed polygon with the following coordinates ((0, 1.5), (3, 1.5), (3, 0), (6, 0), (6, 2), (0, 2), (0, 1.5)). This will create 6 lines and a boundary. The six lines will be assigned to the boundary created.
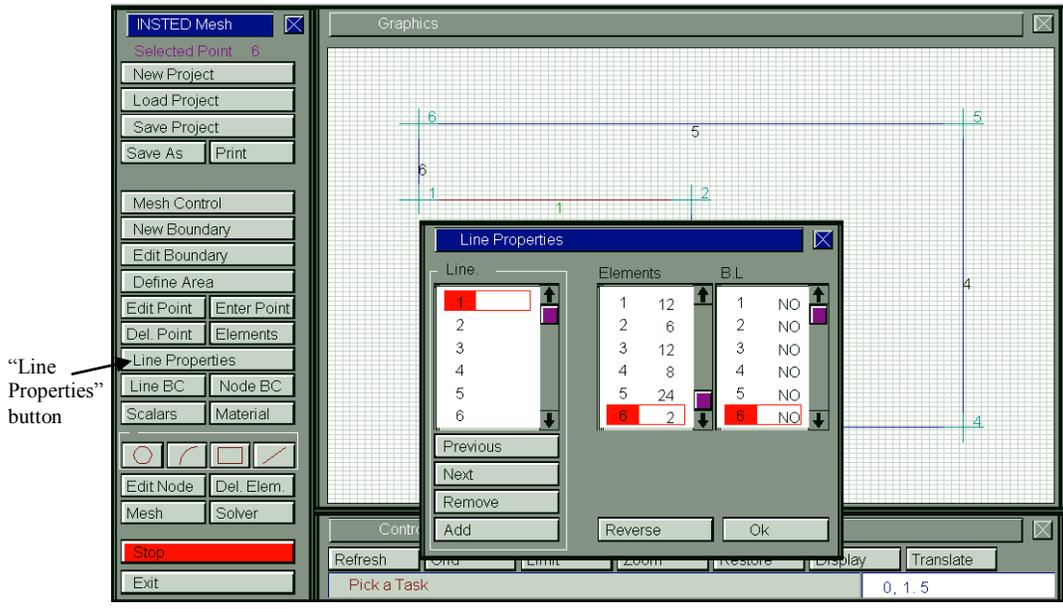
2.6 Formatting Lines for Mesh Generation

During mesh generation, it is necessary to define the number of elements required on every line. Initially, after they are created, all lines will contain only one (nine-node) element per side by default. Entering the number of elements needed per line is done in the 'Line Properties' dialog box.

2.6.1 The Lines Properties dialog box

The mesh properties of a line are defined in this dialog box. This includes the number of elements on the line as well as the boundary layer designation of the line. If boundary layer (B. L.) lines exist in a model, the global boundary layer behavior will need to be defined by clicking the **Mesh Control** button. This is described in Section 2.13.2. The Lines Properties dialog box is invoked by clicking the **Line Properties** button on the Main dialog box.

For the sample problem, we may choose to have 2 elements per 0.5 unit length of line. Therefore, the lines will be divided into 12, 6, 12, 8, 24, and 2, as shown in the figure below.

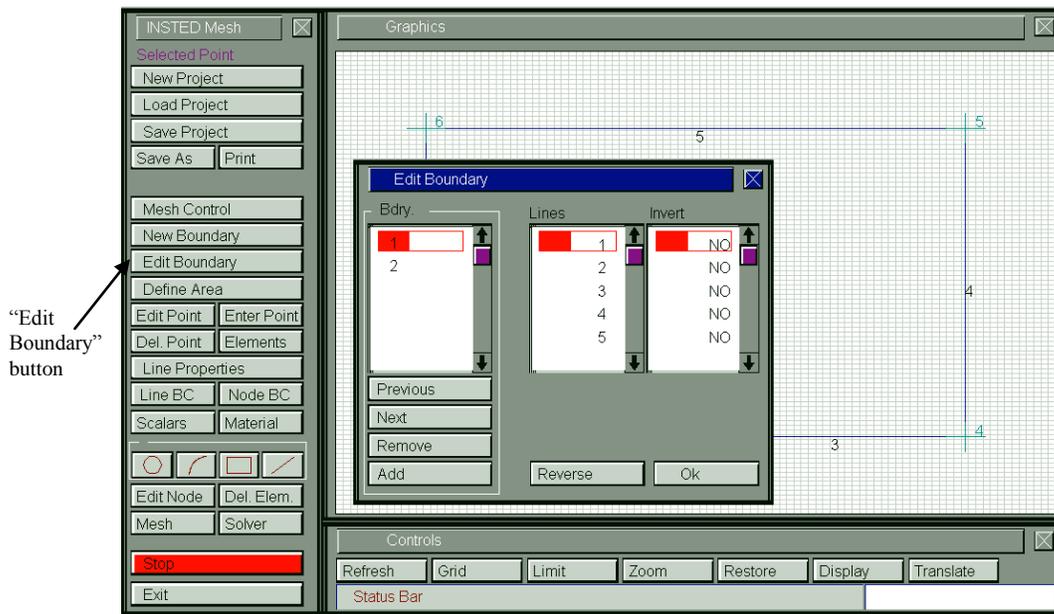"Line Properties" button

2.7 Defining and Editing Boundaries

Boundaries are created with the Modeling tools and the New Boundary tool. These boundaries can be viewed and edited in the 'Edit Boundary' dialog box shown and described below. Lines can be removed from or added to an existing boundary. In addition, a new boundary may be created in this dialog box using the Add button. Once a boundary is added, it necessary to include the lines belonging to the boundary in the 'Lines' listbox. A line may belong to more than one boundary.

2.7.1 Edit Boundary dialog box

This is the dialog box in which boundaries are defined. This dialog box is invoked by clicking the **Edit Boundary** button on the Main dialog box. Lines that are included in this boundary are entered in the 'Lines' listbox. The orientation of the lines is described in the 'Invert' listbox. In general, lines are oriented from their start point to the end point. Inverting a line reverses this orientation.

Previous and **Next** button
These two buttons navigates within the 'Bdry' listbox and are used to select or view the specification of a particular boundary. The **Previous** button moves the current selection of boundary to the previous boundary in the list while the **Next** button moves to the next boundary.

"Edit Boundary" button

Remove button
Deletes a boundary from the list of boundaries. A boundary can only be removed if it is not assigned to a currently defined area. Note that removing a boundary does not automatically remove lines that belong to it.

Add button
This button adds a new boundary to already defined boundaries. The lines contained in this boundary are then entered in the 'Lines' listbox.

In the sample problem, only one boundary was created (with the New Boundary tool) and this boundary consists of 6 lines as shown in the figure above. We will not need to edit this boundary.
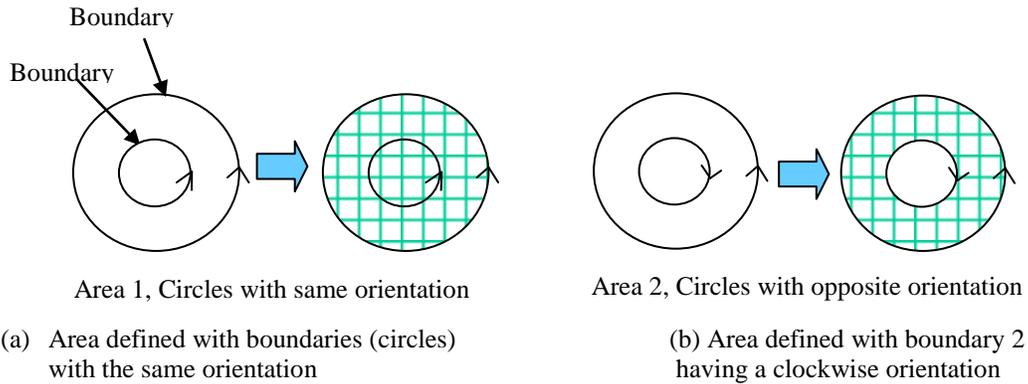
2.8 Defining Areas

Areas consist of boundaries. An area always consists of one external boundary such as the circle representing the shell of a shell-and-tube heat exchanger. The area may also consist of several internal boundaries such as the circles representing the tubes in a shell-and-tube heat exchanger. An external boundary will be made up of a group of lines joined continuously in a counterclockwise fashion. An internal boundary will be made up of one or more lines joined together in a clockwise sense.

The sense of a boundary may be reversed during definition. This is illustrated in the figure below.

In this figure, a model defined by an annulus, for instance, will be composed of two boundaries (lines) each of which is a circle. The outer circle is directed counterclockwise, the inner circle must be directed clockwise. If both circles go counterclockwise, the area defined is not an annulus but the union of two circles as illustrated below.
On generation, the default orientation of all boundaries is counter-clockwise because the constituent lines, when generated, have counterclockwise orientation by default. Invert boundary lines to change its default orientation

Boundary

Boundary

Area 1, Circles with same orientation

Area 2, Circles with opposite orientation

(a) Area defined with boundaries (circles) with the same orientation

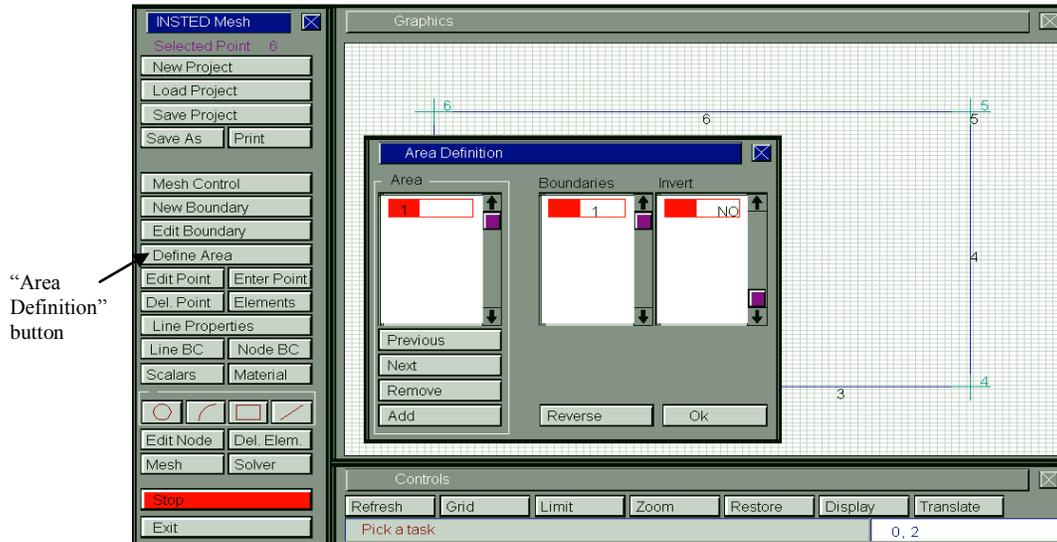(b) Area defined with boundary 2 having a clockwise orientation

In the figure above, Boundaries 1 and 2 belong to both areas in the figure. When boundary 2 has a counter-clockwise orientation, it has a designation of an external boundary. Consequently, a mesh of an area with two external boundaries as shown in (a) above will result in a union of the area enclosed by both circles. In (b), boundary 2 is given a clockwise orientation when defined with the areas. This gives it a designation of an internal boundary (Also see Section 2.2.). Section 2.19 provides a sample problem in which the mesh of the annulus problem is obtained.

To obtain an annulus, you will need to 'invert' the boundary line 2 when it is being included in the area. This is discussed later in this section.

2.8.1 Area Definition dialog box

Previous and **Next** button

"Area Definition" button

These two buttons navigate within the 'Area' listbox and are used to select a particular area for editing or viewing.

Remove button
Deletes an area from the list of areas that are defined.

Add button
This button adds a new area to the number of defined areas. The boundaries contained in this area are then entered in the 'Boundaries' listbox.

For the sample backward-facing step problem, we need to define a single area which is composed on one external boundary - the one created by the New Boundary Tool in section 2.4. To create the area:

1. Click the **Add** button on the dialog box.

→ An area is added in the 'Area' listbox.
2. Type in '1' in the 'Boundaries' listbox.
3. Click Ok to dismiss the dialog box.

2.9 Other Model Building Tools

2.9.1 Entering points

Lines are created by points. Using the modeling tools to create lines automatically creates the points necessary to define those lines. However, in some cases, it might be necessary to create points directly. Points are created by clicking the **Enter Point** button on the Main dialog. The status bar instructs you to click the graphic screen at the point where you want the point or to enter the coordinate of the point in the Input Box.

2.9.2 Editing points

Points may be edited or moved after they are created by clicking the **Edit Point** button on the Main dialog box. The status bar instructs you to click a point on the graphic screen to select the desired point. You are then prompted to click on the intended new position of the point or to enter its new coordinate in the Input Box. It may sometimes be necessary to Zoom or obtain a close-up view of the graphic screen before you can correctly select points. This is the case in densely packed regions.

2.9.3 Deleting points

Points may be deleted by clicking the **Del. Point** button on the Main dialog box. You are prompted to select the desired point. You are then requested to confirm deletion after which the point will be deleted. Points that belong to a line may not be deleted except the lines to which the point belongs are deleted first.

2.9.4 Defining the number of elements on a side

The number of elements on a side may be assigned as in Section 2.4. Alternatively, single lines may have the number of elements defined by clicking the **Elements** button on the main dialog. You are prompted to select the desired line. This is done by clicking the line on the graphic screen or by entering the line number in the Input Box. After the line is

selected, you will then be prompted to enter the number of elements desired on the line in the Input Box.

2.10 Entering the Number of Scalars

INSTED solves the governing equations for as many as 20 passive scalars convected by fluid flow. During the mesh generation process, the number of scalars required may be entered by clicking the **Scalars** button on the Main dialog box. You are then prompted to enter the number of scalars.

2.11 Entering the Number of Materials

IN INSTED, different areas will be assigned different material numbers. This is to allow the elements created to be flagged correctly for identification during the Solver phase. However, material properties are entered in the Solver. The number of materials required may be entered by clicking the **Material** button on the Main dialog box. You are then prompted to enter the number of materials required. If number of materials is not entered, the default number of materials is the same as the number of areas.
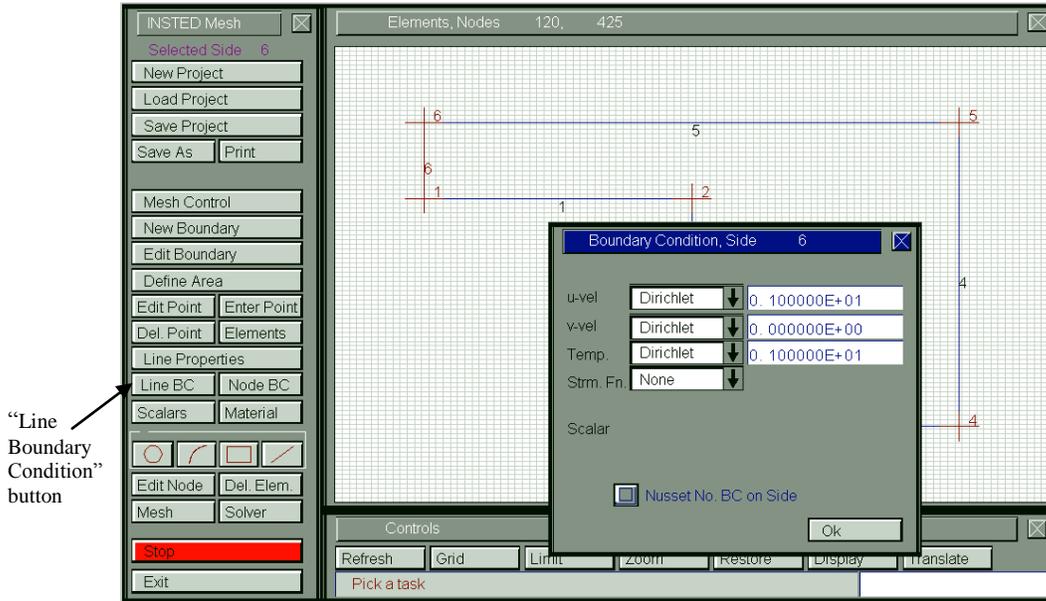
2.12 Defining Boundary Conditions in INSTED

In INSTED, boundary conditions are usually assigned to lines. However, boundary conditions may also be assigned to nodes after the mesh is completed. This is to allow you to prescribe a precise profile along the boundary. For instance, a boundary layer velocity profile may be assigned to a line by specifying an increasing (boundary layer) velocity profile along the nodes of the line.

2.12.1 Assigning boundary conditions to a line

Boundary conditions are assigned to a line by clicking the **Line BC** button on the Main dialog. You are prompted to select the line to which the boundary condition will be assigned. This is done by clicking the line on the graphic screen or by entering the line number in the Input Box. A 'Boundary Condition' dialog box appears. This dialog box is

formatted to receive input for the boundary conditions for *u*-velocity, *v*-velocity, temperature, stream function and as many scalars as are defined. The types of boundary conditions that may be specified include Dirichlet, Neuman, and flux. The boundary condition is entered by clicking the drop-down listbox and selecting the desired boundary condition type in the 'Boundary Condition' dialog box.

2.12.2 Assigning boundary conditions to a node

INSTED allows the specification of boundary conditions on a node-by-node basis **after the mesh has been generated**. Therefore, the instructions below for assigning boundary condition to a node should be carried out only after you have generated your mesh.

Boundary conditions are assigned to a node by clicking the **Node BC** button on the Main dialog. You are prompted to select the target node.

This may be done by clicking the node on the graphic screen or entering the node number in the Input Box. The 'Boundary Condition' dialog is presented. It is formatted to receive input in the same way as described above except that the boundary conditions defined will be assigned only to the selected node.

2.13 Graphics Related Commands

<u>Refresh</u>
The **Refresh** button causes the screen to be redrawn. This button is located in the 'Controls' dialog box.
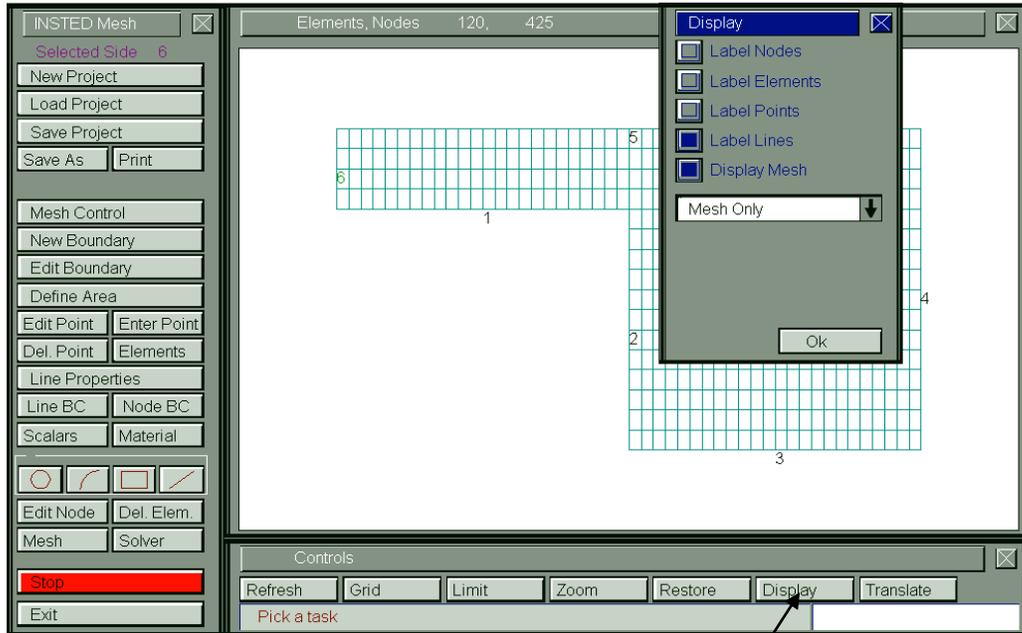
<u>Grid</u>
The **Grid** button turns the printing of grid lines on or off. This button is located in the 'Controls' dialog box.

2.13.1 The Display dialog box

This dialog box is obtained by clicking the **Display** button on the 'Controls' dialog box. The display dialog formats the way graphics are displayed on the screen. This dialog box is shown in the figure below. As can be observed on the screen, the printing of line labels, point, node or element labels may be turned on or off from this dialog box. In the graphics displayed in the picture, only line labeling are currently turned on.

The listbox on the 'Display' dialog box allows you to view the boundary conditions that you have specified for the mesh. For instance, you may choose to view nodes with Dirichlet boundary conditions. This option may be selected from this dialog box.

"Display"
button

Boundary Condition view options in the View Listbox

Listbox Item	Function
Mesh Only	Only a mesh will be displayed. This is the default view.
All Variables, Dirichlet	Display mesh and the nodes with Dirichlet boundary conditions; when it exists; for the *u*-

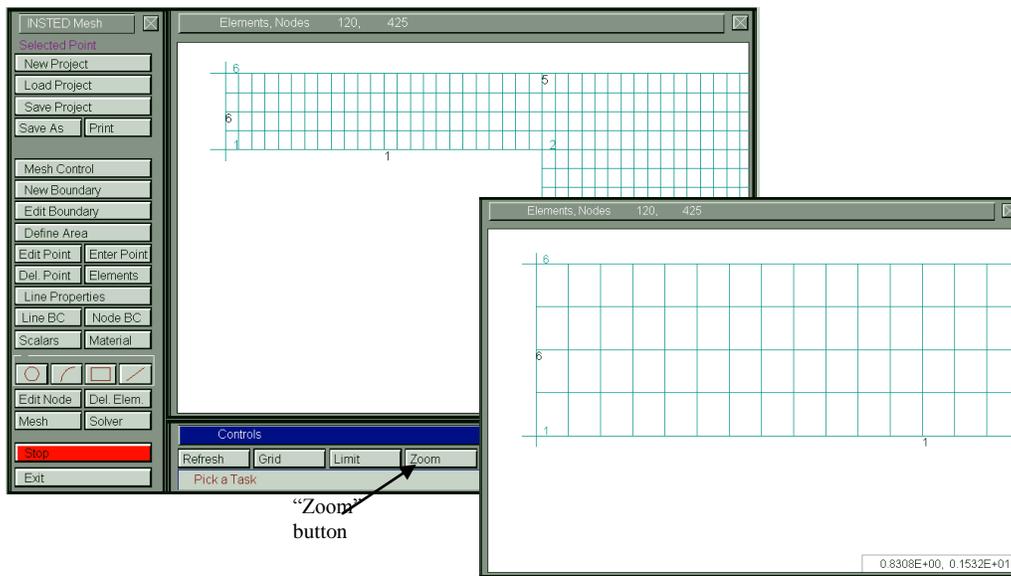|	
	velocity, *v*-velocity, temperature and stream function.
All Variables, Flux	Display mesh and the nodes with flux boundary conditions; when it exists; for *u*-velocity, *v*-velocity, temperature and stream function.
u-velocity, Dirichlet	Displays mesh and the nodes with Dirichlet boundary conditions for *u*-velocity.
u-velocity, Flux	Displays mesh and the nodes with Flux boundary conditions for *u*-velocity.
v-velocity, Dirichlet	Displays and the nodes with Dirichlet boundary conditions for *v*-velocity.
v-velocity, Flux	Displays mesh and the nodes with Flux boundary conditions for *v*-velocity.
Temperature, Dirichlet	Displays mesh and the nodes with Dirichlet boundary conditions for temperature.
Temperature, Flux	Displays mesh and the nodes with Flux boundary conditions for temperature.
Stream Fn., Dirichlet	Displays mesh and the nodes with Dirichlet boundary conditions for stream function.
Stream Fn., Flux	Displays mesh and the nodes with Flux boundary conditions for stream function.
Nusset No.	Displays mesh and nodes for Nusset No. calculation.
Scalar(s), Dirichlet	Displays mesh and the nodes with Dirichlet boundary conditions for scalars.
Scalar(s), Flux	Displays mesh and the nodes with Flux boundary conditions for scalars.

2.13.2 Zooming within the graphics screen

INSTED Mesh Generator allows you to obtain a close-up view of parts of a model. This may be useful for inspecting the mesh or in selecting points, lines, or other objects in a densely packed portion of the graphics screen.

Zooming

Initiate zooming by clicking on the **Zoom** button in the 'Controls' dialog box. You will then be prompted for the top left and bottom right corner of the zoom area. You may zoom further within a zoom screen.

Restore

You may return to the previous configuration (before a zoom) by clicking the **Restore** button, also located in the 'Controls' dialog box. Note that restore can only be done within two zoom levels. You may however return to the initial view by re-entering the original data in the 'Limits' dialog box as described in Section 2.4.
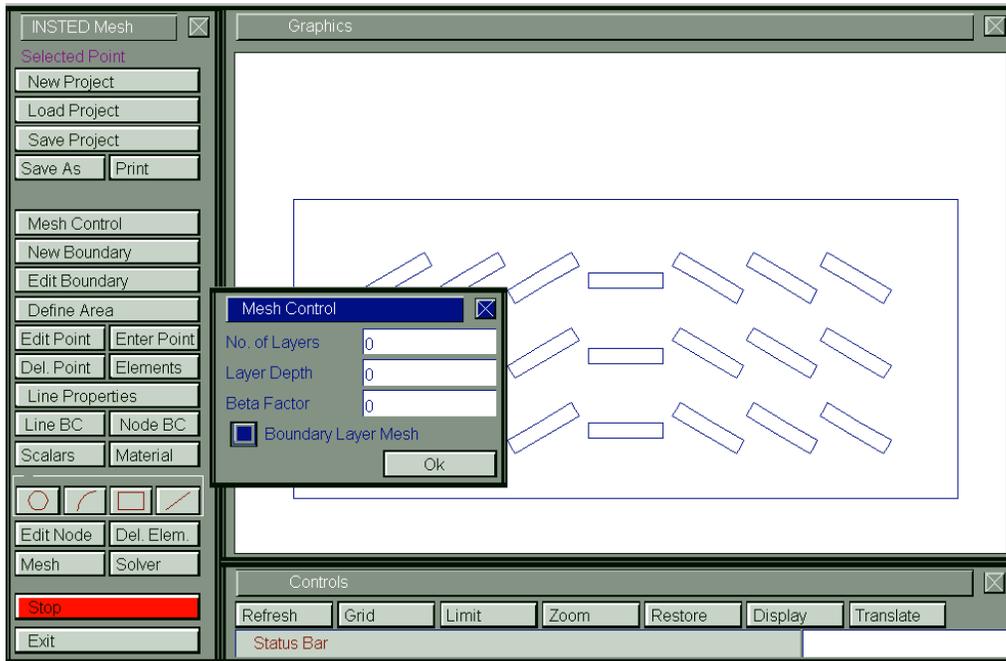

"Zoom" button

2.14 Specifying Boundary Layer Parameters

The procedures for generating boundary layer mesh are defined in the Mesh Control dialog box. This includes the boundary layer designation and boundary layer parameters, including the number of layers, depth, and stretch factor.

A value of stretch factor between 1.01 and 1.05 produces the packed cell look that is obtained for meshes intended for boundary layer computation. The boundary layer depth is the physical thickness of the boundary layer. The number of layers is simply the number of elements desired within the boundary layer.

2.14.1 Mesh Control dialog box

This dialog box provides the means for specifying the data required for generating boundary layers, including the number of layers, depth of layer (in model units) and the stretch factor. The stretch factor must be a real number between 1 and 1000. The mesh control dialog box is obtained by clicking the "Mesh Control" button on the Main dialog box.

Section 2.20 provides a complete example of a boundary layer mesh and illustrates the use of the boundary layer mesh parameters.
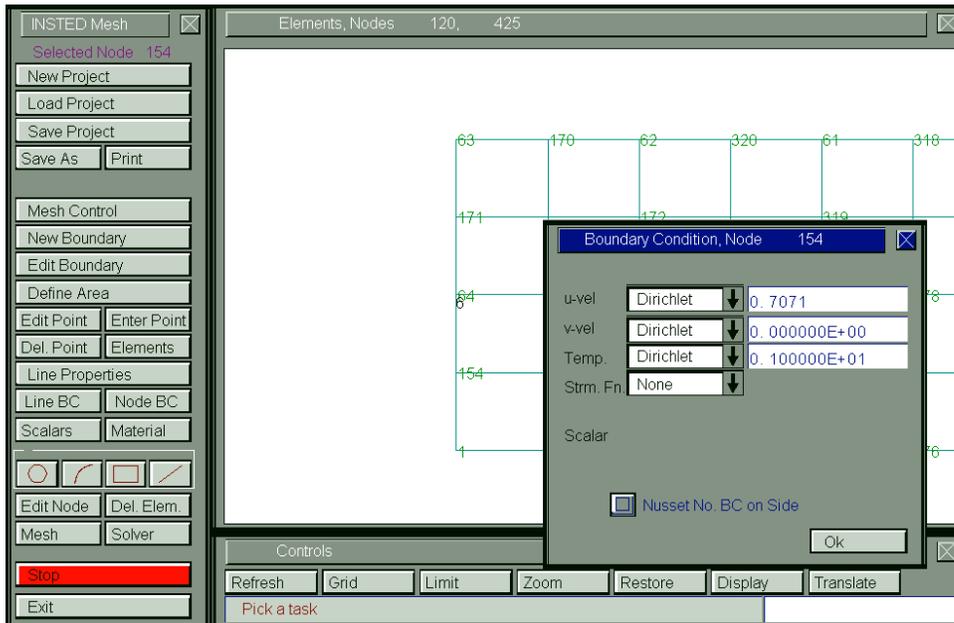
2.15 Meshing the Area

To mesh the part after providing the various data, click the **Mesh** button located in the Main dialog box.

2.16 Post Mesh Generation Activities

After the mesh is generated, it may be necessary to modify the boundary conditions for some specific nodes, view the boundary conditions on nodes or label the elements and nodes for mesh inspection purposes. Commands that help you to label nodes and elements are located on the 'Display' dialog box and are described in Section 2.13.1. Commands that assist you to focus (or zooming) on specific areas of the mesh are described in Section 2.13.2.

For the backward-facing step sample problem, we wish to specify a parabolic profile for the velocity profile at the inlet. Therefore, we need to individually select the nodes along Line 6 (inlet) and assign them specific values. This is illustrated in the figure below. Initially, after mesh generation, the nodes along Line 6 will have a u-Dirichlet value of 1.0, which is the value assigned to the line. To obtain the parabolic profile required for the model, we need to assign a value of 0.7071 to the nodes 171 and 154.

2.17 Saving the Project

Note that created meshes are automatically stored in the file mesh.dat, bound.dat and xnuslt.dat. In addition, there is an automatic restart file that saves your project if you exit the program normally. (You exit normally by clicking the **Exit** button on the Main dialog box of INSTED/CFD (2D) Mesh Generator.) However, for project management, you may wish to save your work in other directories with user-defined names. To save your project in a special file for later retrieval, follow the procedure below:

Click the **Save As** button on the Main dialog box.
→ An operating system 'Save File' dialog box appears.
Follow operating system procedures for saving a file.

2.18 Launching INSTED/CFD (2D) Solver from the Mesh Generator

You may wish to proceed directly to the Solver to generate the CFD solutions to your meshed models or simply save the file and exit the program. To carry out a CFD solution, simply click the **Solver** button located on the Main dialog box. INSTED/CFD Solver is launched. The current mesh data is transferred to the solver including all the boundary condition information. Instructions for using the solver are presented in the next two chapters.

2.19 The Annulus Problem

The annulus problem was discussed in section 2.8. Follow the steps below to generate the mesh for a cylindrical annulus having an internal diameter of 1 unit and an external diameter of 2, with a center at (0,0):

1.   Click the **New Project** button on the Main dialog box.
2.   Click the **Limit** button on the 'Controls' dialog box.
→   The 'Limit' dialog box appears.
3.   Enter -4.4, 4.4, -3.0, and 3.0 for the lower x limit, higher x limit, lower y limit, and higher y limit, respectively.
4.   Click the **Ok** button.

Creating the Lines
5.   Click the Circle icon from the Modeling tools on the Main dialog box.
→   The status bar prompts you for the center point of the circle.
6.   Enter '0, 0' in the Input Box and press the Enter key.
→   The status bar prompts you for a point on the circle.
7.   Enter '0.5, 0.5' in the Input Box and press the Enter key.
→   A circle is created and displayed in the 'Graphics' dialog box.
8.   Repeat step (4) to (6) for the outer circle with center at (0, 0) and point (1, 1).

Formatting the Lines for Mesh Generation
9.   Click the **Line Properties** button on the Main dialog box.
→   The 'Line Properties' dialog box appears
10. Enter the values of 20 and 30 in the 'Elements' listbox for the number of elements for the inner and outer circles, respectively.
11. Click the **Ok** button to dismiss the 'Line Properties' dialog box.

Defining and Editing Boundaries
12. Click the **Edit Boundary** button on the Main dialog box.
→   The 'Edit Boundary' dialog box appears. The first boundary is highlighted in the 'Bdry.' Listbox, and by default consists of only one uninverted line (Line 1), as shown in the 'Lines' listbox and the 'Invert' listbox. **Note that the default orientation of boundaries created by the Modeling tools is anticlockwise.**
13. Click the Next button to view the default properties of the second boundary.
→   The second boundary is highlighted. It is shown as consisting of Line 2 and is uninverted.

14. Click the **Ok** button to dismiss the 'Edit Boundary' dialog box. (Note that for this sample mesh, we did not have to make any changes to the default boundaries defined by the Modeling tool).

Defining an Area Containing only the Inner Circle

In this case, the area is defined only by the inner boundary of the annulus.

15. Click the **Define Area** button on the Main dialog box.
→ The 'Define Area' dialog box appears. The dialog box shows that there is no area defined yet.
16. Click the **Add** button on this dialog box.
→ Area '1' is added to the 'Area' listbox.
17. Click on the first line of the 'Boundaries' listbox and type the value '1' and press the Enter key. (This step implies that area 1 is defined completely by boundary 1, and is the area inside the inner circle.)
18. Click the **Ok** button to dismiss the 'Area Definition' dialog box
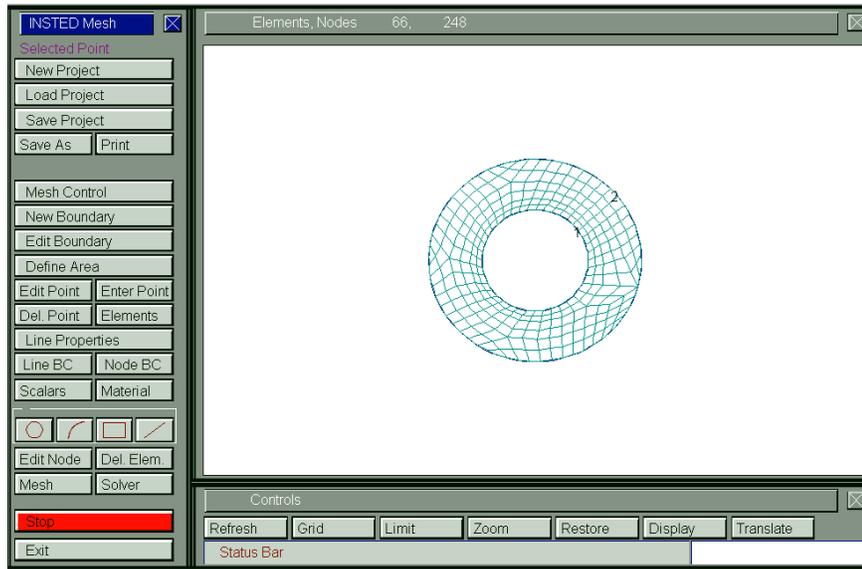
Meshing the Area Inside the Inner Circle

19. Click the Mesh button on the Main dialog box.
→ A mesh of the inner circle is obtained.

Defining the Area for the Annulus

(Note that boundary 1 represents the inner circle, which we created first, and boundary 2 represents the outer circle).
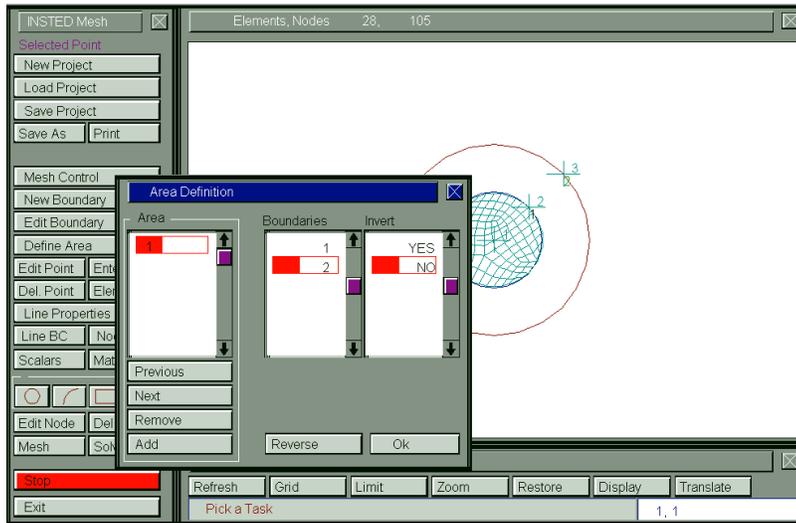
20. Click the **Define Area** button on the Main dialog box.
→ The 'Define Area' button appears. The dialog box shows that only one area - Area 1 - is defined and it contains boundary 1 in an uninverted form. This is the default of the program.
21. Click on the second line of the 'Boundaries' listbox and type the value '2' and press the Enter key. (Note that this serves to include boundary 2 as one of the boundaries enclosing Area 1).
22. Click the 'NO' text of the first line of the 'Invert' listbox to invert the boundary 1. (Clicking on an item in this listbox toggles the value between 'YES' and 'NO').
→ The dialog box should now appear as shown in the figure below:

23. Click the **Ok** button to dismiss the 'Area Definition' dialog box.

Meshing the Annulus
24. Click the **Mesh** button on the Main dialog box
25. You should obtain a mesh as shown in the figure below.

What's Next?

26. Click the Save As button on the Main dialog box. Follow operating system procedure for saving the file. Use a filename such as 'annulus.mes' and place the file in a desired directory.

27. Go back and alter the number of the elements on each of two the lines to investigate the effect this would have on your mesh.

2.20 Creating a Sample Boundary Layer Mesh

In this section, we will generate a boundary layer mesh of the annulus problem just meshed to illustrate the use of the boundary layer parameters - depth, number of layers, and the stretch factor. Since the radius of the inner circle is 0.5 and the outer radius is 1.0, a boundary layer of depth 0.4 will be used for illustration purposes. In an actual flow calculation, the boundary layer thickness will be considerably smaller, dependent on the Reynolds number, Re, and will have a value of the order $\sim 5/\sqrt{Re}$. We will initially pack 7 layers of elements within the boundary layer using a stretch factor of 1.04. Boundary layer mesh would be generated on both annulus boundaries.
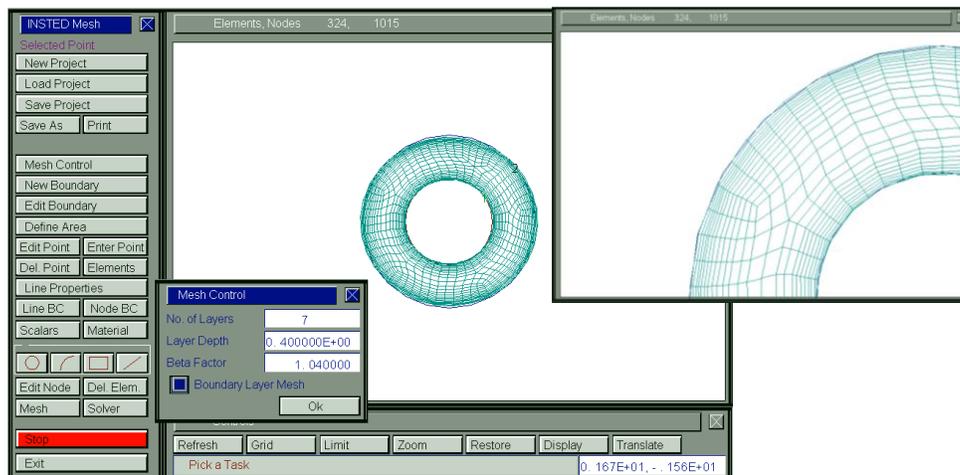
1. Create the annulus model described above section or load the file containing the annulus mesh problem. Follow the procedure in the next two lines to load the file:
   1.1. Click the **Load Project** button on the Main dialog box.
   1.2. Follow operating system procedure to locate and load files.

Formatting the Lines
2. Click the **Line Properties** button on the Main dialog box,
→ The 'Line Properties' dialog box appears.
3. Click the 'B.L' listbox for the two lines that form the boundary of the object.
→ The listbox should now have the text 'YES' on both lines.
4. Click the OK button to dismiss this dialog box.

Specifying Boundary Layer Parameters
5. Click the **Mesh Control** button on the Main dialog box.
→ The 'Mesh Control' dialog box appears.
1. Click the radio button 'Boundary Layer Mesh' to indicate that you wish to generate a boundary layer mesh.
→ Input boxes to receive data for number of layers, layer depth and beta factor appears on the 'Mesh Control' dialog box.
6. Enter the number 7 for the number of layers, 0.4 for depth and 1.04 for the beta or stretch factor. This is illustrated in the figure below.
7. Click the **Ok** button to dismiss the dialog box.

Generating the Boundary Layer Mesh

8. Click the **Mesh** button to mesh the model.

Viewing the Details of the Generated Boundary Layer Mesh

9. Click the **Zoom** button on the 'Controls' dialog box.

→ You are prompted to click the top left corner of the desired view area.

10. Click on an area of the screen close to the top left corner of the annulus

→ You are prompted to click the bottom right corner of the desired view area.

11. Click on an area close to the center of the annulus

→ A close-up view of the zoomed region will be presented.

12. Click the **Restore** button on the Main dialog box to undo the zoom operation. (Note you may restore multiple zoom operations by clicking the Restore button as many times as you performed the zoom operation.)

What's Next?

13. Modify the number of layers (use a value of 12, for instance) to examine the effect on the mesh. Also alter the beta factor, β, using a value between 1.02 and 1.5 (Note that $1.0 < \beta < 1000.0$).

2.21 Creating a Sample Shell and Tube Mesh

In this section, a sample shell-and-tube system consisting of a shell of radius 1 and four tubes with radius 0.2 each will be meshed. The center of the shell is at (0, 0) and the tubes will be located at (0.5, 0.5), (-0.5, 0.5), (0.5, -0.5) and (-0.5, 0.5) respectively. The steps for generating the mesh is as follows:

1. Click the **New Project** button on the Main dialog box.
2. Click the **Limit** button on the 'Controls' dialog box.
→ The 'Limit' dialog box appears.
3. Enter -4.4, 4.4, -3.0, and 3.0 for the lower *x* limit, higher *x* limit, lower *y* limit, and higher *y* limit, respectively.
4. Click the **Ok** button.

Creating the Lines

5. Click the Circle icon from the Modeling tools on the Main dialog box.
→ The status bar prompts you for the center point of the circle.
6. Enter "0, 0" in the Input Box and press the Enter key.
→ The status bar prompts you for a point on the circle.
7. Enter "1, 1" in the Input Box and press the Enter key.
→ A circle is created and displayed in the 'Graphics' dialog box.
8. Repeat step (4) through (6) for the center (0.5, 0.5) and point (0.7, 0.7).
9. Repeat step (4) through (6) for the center (-0.5, 0.5) and point (-0.7, 0.7).
10. Repeat step (4) through (6) for the center (0.5, -0.5) and point (0.7, -0.7).
11. Repeat step (4) through (6) for the center (-0.5, -0.5) and point (-0.7, -0.7).
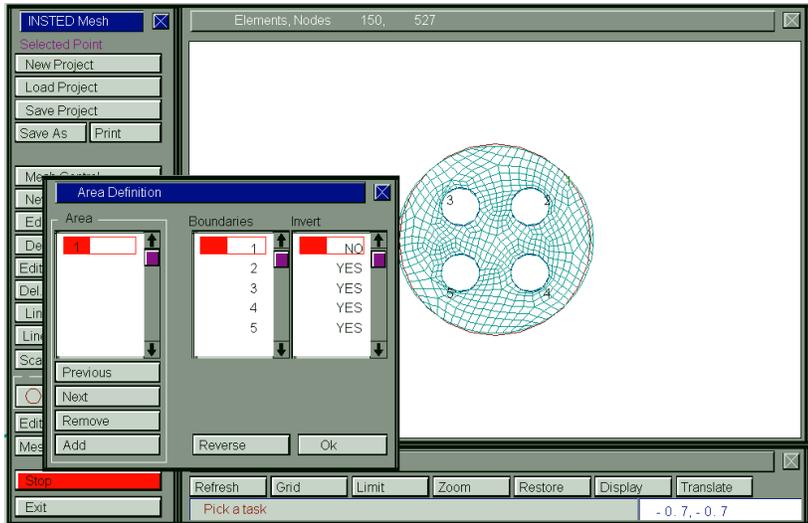
Formatting the Lines for Mesh Generation

12. Click the **Line Properties** button on the Main dialog box.
→ The 'Line Properties' dialog box appears
13. Enter 40 for the number of elements to use for the outer circle and 10 for each of the inner circles, respectively, in the 'Elements' listbox.
14. Click the **Ok** button to dismiss the 'Line Properties' dialog box

Defining the Area

15. Click the **Define Area** button on the Main dialog box
→ The 'Define Area' dialog box appears. The dialog box shows that no area has been defined
16. Click the **Add** button on this dialog box
→ Area '1' is added to the 'Area' listbox

17. Click on the first line of the 'Boundaries' listbox and type '1' and press the Enter key. Note that this serves to include boundary 1 in the Area 1. Add the boundaries 2, 3, 4, and 5 respectively repeating the above procedure. Note that boundary 1 is the shell whereas boundaries 2 through 5 are the tubes.
18. Click the second line of the 'Invert' listbox to invert the orientation of the first inner circle. Invert the orientation of all the inner circles using the same procedure. Your dialog box should look like that in the figure below, without the mesh.
19. Click the **Ok** button to dismiss the 'Area Definition' dialog box.

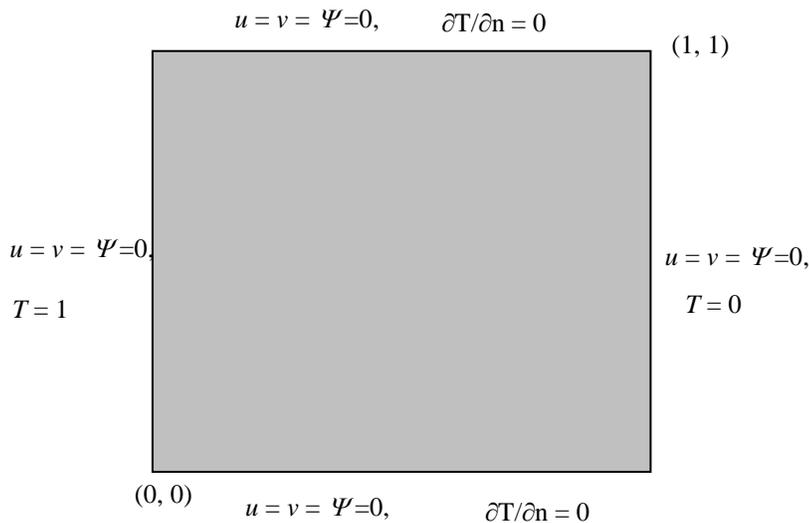
Meshing the Area Between the Shell and the Tubes
20. Click the Mesh button on the Main dialog box.
→ A mesh of the region between the shell and the tubes will be generated. Your mesh should be similar to that in the above figure. Note that the entire region inside the shell would have been meshed had we not inverted the orientation of the boundaries for the tubes.

2.22 Natural Convection in a Square Cavity

The problem being considered is that of the two-dimensional flow of Boussinesq fluid of Prandtl number 0.71 inside an upright square cavity of side L ($L = 1$ in non-dimensional form). 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.

This problem will be used to illustrate the procedure for obtaining the solutino to a CFD problem whose mesh has been generated by the procedures discussed below.

The physical domain is shown below; where (u, v, T, Ψ) are the x-component of velocity, y-component of velocity, temperature, and stream function, respectively.

$u = v = \Psi = 0,$ $\partial T/\partial n = 0$

(1, 1)

$u = v = \Psi = 0,$

$T = 1$

$u = v = \Psi = 0,$

$T = 0$

(0, 0) $u = v = \Psi = 0,$ $\partial T/\partial n = 0$

The model for this sample problem will be composed of a square with a counter-clockwise orientation. The procedure for generating the mesh for this problem is as follows:

2.   Start INSTED Mesh Generator.
3.   Click the **New Project** button on the Main dialog box.
4.   Click the **Limit** button on the 'Controls' dialog box.
→   The 'Limits' dialog box appears.
5.   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.
6.   Click the OK button to dismiss this dialog box.

To draw the square

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

Defining number of elements per line

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

Defining the area to mesh

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

Applying boundary conditions

18. Click the **Line BC** button on the Main dialog box,
→   Status bar displays message: 'Pick Side'.
19. Click 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" (You should confirm from the title of the dialog box that data is being received for Line 1. This may not be the case if you accidentally selected some other line.).

20. 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.
21. Click the drop-down list box for *v*-velocity and select a Dirichlet boundary condition from the list and enter '0.0' in the accompanying input box that appears as soon as Dirichlet condition is selected.
22. Click the drop-down list box for stream function and select a Dirichlet boundary condition from the list and enter '0.0' in the accompanying input box that appears as soon as Dirichlet condition is selected.
    Note: The boundary condition for temperature for line 1 and line 3 is that these sides are adiabatic. That is, the gradient of temperature is zero on those lines. This is the default boundary condition in INSTED. Therefore, you do not need to specify any boundary condition for temperature.
23. Click the **OK** button to dismiss the dialog box.
24. Repeat procedures 17 through 21 for Line 3.
25. Click the **Line BC** button on the Main dialog box,
→ Status bar displays message: 'Enter the Line Number'.
26. 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 for Line, 4" (confirm from the title of the dialog box that data is being received for Line 4. This may not be the case if you accidentally selected some other line.).
27. Click the drop-down list box for *u*-velocity and select a Dirichlet boundary condition from the list and enter '0.0' in the accompanying input box that appears as soon as Dirichlet condition is selected.
28. Click the drop-down list box for *v*-velocity and select a Dirichlet boundary condition from the list and enter '0.0' in the accompanying input box that appears as soon as Dirichlet condition is selected.
29. Click the drop-down list box for stream function and select a Dirichlet boundary condition from the list and enter '0.0' in the accompanying input box that appears as soon as Dirichlet condition is selected.
30. Click the drop-down list box for temperature and select a Dirichlet boundary condition from the list and enter '1.0' in the accompanying input box that appears as soon as Dirichlet condition is selected.
31. Click on the radio button 'Nusset No. B.C. on Side' to select calculation of Nusset No. for Line 4.
32. Click the **OK** button to dismiss the dialog box.

33. Repeat the procedure (24-29) for Line 2 using zero Dirichlet condition for *u*, *v*, and *Ψ*, and Dirichlet condition for temperature, *T*, with *T* = 0.0.
34. Click the **OK** button to dismiss the dialog box.

Mesh and Confirm Mesh Results

35. Click the **Mesh** button on the Main dialog box,
→ A mesh of the region defined by the lines is displayed on the 'Graphics' dialog box.
36. Click the **Display** button on the 'Controls' dialog box.
→ The 'Display'dialog box appears.
37. Click the Views listbox arrow on this dialog box, select 'Temperature, Dirichlet '
38. Click the **Ok** button on this dialog box to view nodes with Dirichlet temperature boundary conditions.
39. Click the **Refresh** button on the 'Controls' dialog box to return the screen to it original condition.