Maxwell® 2D
Student Version

Getting Started:

A 2D Electrostatic Problem

November 2002

Notice

Ansoft Corporation

Four Station Square
Suite 200
Pittsburgh, PA 15219
(412) 261 - 3200

Printing History

New editions of this manual will incorporate all material updated since the previous edition. The manual printing date, which indicates the manual's current edition, changes when a new edition is printed. Minor corrections and updates that are incorporated at reprint do not cause the date to change.

Update packages may be issued between editions and contain additional and/or replacement pages to be merged into the manual by the user. Pages that are rearranged due to changes on a previous page are not considered to be revised.

Edition	Date	Software Revision
1	June 1994	6.2
2	November 2002	9.0

Installation

Before you use Maxwell SV, you must:

1. Set up your system's graphical windowing system.
2. Install the Maxwell software, using the directions in the Ansoft *PC Installation Guide.*

If you have not yet done these steps, refer to the Ansoft *Installation* guides and the documentation that came with your computer system, or ask your system administrator for help.

Getting Started

If you are using Maxwell SV for the first time, the following two guides are available for the Student Version of Maxwell 2D:

- *Getting Started: A 2D Electrostatic Problem*
- *Getting Started: A 2D Magnetostatic Problem*

Additional Getting Started guides are available for the standard version of Maxwell 2D.

These short tutorials guide you through the process of setting up and solving simple problems in Maxwell SV, providing you with an overview of how to use the software.

Other References

To start Maxwell SV, you must first access the Maxwell Control Panel.

For information on all Maxwell Control Panel and Maxwell SV commands, refer to the following online documentation:

- *Maxwell Control Panel online help*. This online manual contains a detailed description of all of the commands in the Maxwell Control Panel and in the Utilities panel. The Maxwell Control Panel allows you to create and open projects, print screens, and translate files. The Utilities panel is accessible through the Maxwell Control Panel and enables you to view licensing information, adjust colors, open and create 2D models, open and create plots using parametric equations, and evaluate mathematical expressions.
- *Maxwell 2D online help.* This online manual contains a detailed description of the Maxwell 2D and the Parametric Analysis modules. Maxwell SV does not provide parametric capabilities.

Typeface Conventions

Field Names Bold type is used for on-screen prompts, field names, and messages.

Keyboard Entries Bold type is used for entries that must be entered as specified.

Example: Enter **0.005** in the **Nonlinear Residual** field.

Menu Commands Bold type is used to display menu commands selected to perform a specific task. Menu levels are separated by a carat.

Example 1: The instruction “Click **File>Open**” means to select the **Open** command on the **File** menu within an application.

Example 2: The instruction “Click **Define Model>Draw Model**” means to select the **Draw Model** command from the **Define Model** button on the Maxwell SV **Executive Commands** menu.

Variable Names Italic type is used for keyboard entries when a name or variable must be typed in place of the words in italics.

Example: The instruction “**copy** *filename*” means to type the word **copy**, to type a space, and then to type the name of a file, such as **file1**.

Emphasis and Titles Italic type is used for emphasis and for the titles of manuals and other publications.

Keyboard Keys Bold Arial type is used for labeled keys on the computer keyboard. For key combinations, such as shortcut keys, a plus sign is used.

Example 1: The instruction “Press **Return**” means to press the key on the computer that is labeled *Return*.

Example 2: The instructions “Press **Ctrl+D**” means to press and hold down the *Ctrl* key and then press the *D* key.

Table of Contents

1. Introduction 1-1

Sample Problem 1-2
General Procedure 1-3
Results to Expect 1-5

2. Creating the Microstrip Project 2-1

Access the Maxwell Control Panel 2-2
Access the Project Manager 2-2
Create a Project Directory 2-3
Create a Project 2-4
Access the Project Directory 2-4
Create the New Project 2-4
Save Project Notes 2-5

3. Accessing the Software 3-1

Open the New Project and Run the Simulator 3-2
Executive Commands Window 3-3
Executive Commands Menu 3-3
Display Area 3-3
Solution Monitoring Area 3-3

4. Creating the Model 4-1

Specify Solver Type 4-2
Specify Drawing Plane 4-2
Access the 2D Modeler 4-2

Screen Layout 4-3
The Microstrip Model 4-4
Set Up the Drawing Region 4-5
Create the Geometry 4-6
Keyboard Entry 4-6
Create the Substrate 4-6
Create the Left Microstrip 4-8
Create the Right Microstrip 4-9
Save the Geometry 4-10
Create the Ground Plane 4-11
Completed Geometry 4-13
Exit the 2D Modeler 4-13

5. Defining Materials and Boundaries 5-1

Set Up Materials 5-2
Access the Material Manager 5-2
Assign a Material to the Substrate 5-3
Assign a Material to the Microstrips and Ground Plane 5-4
Assigning Materials to the Background 5-4
Exit the Material Manager 5-5
Set Up Boundaries and Sources 5-5
Display the 2D Boundary/Source Manager 5-6
Boundary Manager Screen Layout 5-6
Types of Boundary Conditions and Sources 5-7
Set the Voltage on the Left Microstrip 5-8
Set the Voltage on the Right Microstrip 5-8
Set the Voltage on the Ground Plane 5-9
Assign a Balloon Boundary to the Background 5-9
Leave Substrate with a Natural Surface 5-10
Exit the Boundary Manager 5-10

6. Generating a Solution 6-1

Set Up a Matrix Calculation 6-2
Access the Setup Solution Menu 6-3
Modify Solution Criteria 6-4
Specify the Starting Mesh 6-4
Specify the Solver Residual 6-4
Specify the Solver Choice 6-5
Specify the “Solve for” Options 6-5
Specify the Adaptive Analysis Settings 6-5

Exit Setup Solution 6-5
Generate the Solution 6-6
Monitoring the Solution 6-6
Solution Criteria 6-6
Completed Solutions 6-7
Completing the Solution Process 6-7
Viewing Final Convergence Data 6-8
Plotting Convergence Data 6-9
Viewing Statistics 6-10

7. Analyzing the Solution 7-1

Access the Post Processor 7-2
Post Processor Screen Layout 7-2
General Areas 7-2
Menu Bar 7-3
Status Bar 7-3
Scientific Notation 7-3
Plot the Electric Field 7-4
Formatting and Manipulating Plots 7-6
Rotate a Plot 7-7
Hide or Show a Plot 7-7
Open Multiple Plots 7-7
The 2D Calculator 7-8
Examine Results of Capacitance Computation 7-9
Verify Capacitance Calculation 7-9
Reset the Conductor Values 7-9
Remove Mesh Refinement 7-9
Run the Solution Again 7-10
Calculate Capacitance 7-10
Enter the Post Processor Field Calculator Again 7-10
Compute the Energy 7-10
Compute the Capacitance Value 7-12
Exit the Calculator 7-13
Exit the Post Processor 7-13
Exit Maxwell 2D 7-13
Exit the Maxwell Software 7-13

1 Introduction

This guide is a tutorial for setting up an electrostatic problem using version 9.0 of Maxwell 2D Student Version (SV), a software package for analyzing electromagnetic fields in cross-sections of structures. Maxwell SV uses finite element analysis (FEA) to solve two-dimensional (2D) electromagnetic problems.

To analyze a problem, you need to specify the appropriate geometry, material properties, and excitations for a device or system of devices. The Maxwell software then does the following:

- Automatically creates the required finite element mesh.
- Iteratively calculates the desired electrostatic or magnetostatic field solution and special quantities of interest, including force, torque, inductance, capacitance, and power loss. You can select any of the following solution types: Electrostatic, Magnetostatic, Electrostatic, Eddy Current, DC Conduction, AC Conduction, Eddy Axial. The student version does not contain thermal, transient, or parametric capabilities.
- Provides the ability to analyze, manipulate, and display field solutions.

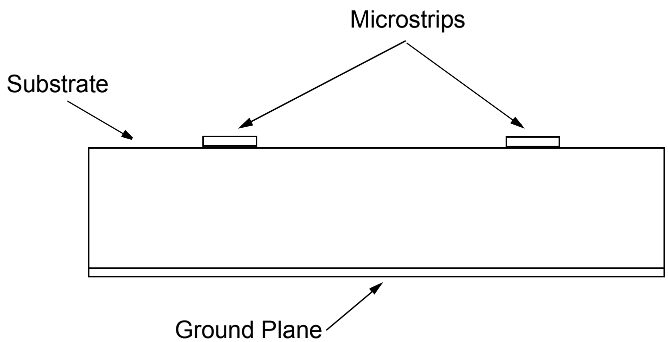
Many models are actually a collection of three-dimensional (3D) objects. Maxwell SV analyzes a 2D cross-section of the model, then generates a solution for that cross-section, using FEA to solve the problem. Dividing a structure into many smaller regions (finite elements) allows the system to compute the field solution separately in each element. The smaller the elements, the more accurate the final solution.

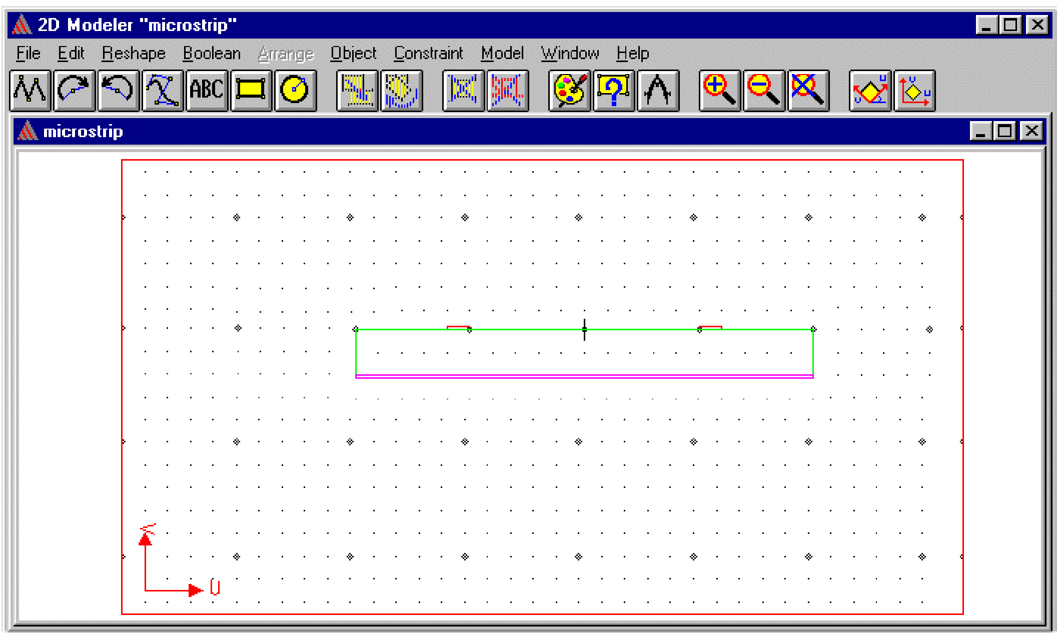
Sample Problem

After starting the software and introducing the **Executive Commands** window, this guide steps you through the setup, solution, and analysis of a simple electrostatic problem. The sample problem, shown below, is an electrically insulated structure that includes the following:

- Two microstrip lines, one set to +1 volt, and the other set to -1 volt.
- A substrate.
- A ground plane.

In this guide, you will draw, set up, and solve the sample microstrip problem shown below. Detailed dimensions and instructions for drawing this model are given in Chapter 4, “Creating the Model.”





General Procedure

Follows this general procedure when using the simulator to solve 2D problems:

1. Use the **Solver** command to specify which of the following electric or magnetic field quantities to compute:
   - **Electrostatic**
   - **Magnetostatic**
   - **Eddy Current**
   - **DC Conduction**
   - **AC Conduction**
   - **Eddy Axial**
2. Use the **Drawing** command to select one of the following model types:

   XY Plane Visualizes cartesian models as sweeping perpendicularly to the cross-section.

   RZ Plane Visualizes axisymmetric models as revolving around an axis of symmetry in the cross-section.
3. Use the **Define Model** command to access the following options:

   Draw Model Allows you to access the 2D Modeler and draw the objects that make up the geometric model.

   Group Objects Allows you to group discrete objects that are actually one electrical object. For instance, two terminations of a conductor that are drawn as separate objects in the cross-section can be grouped to represent one conductor.

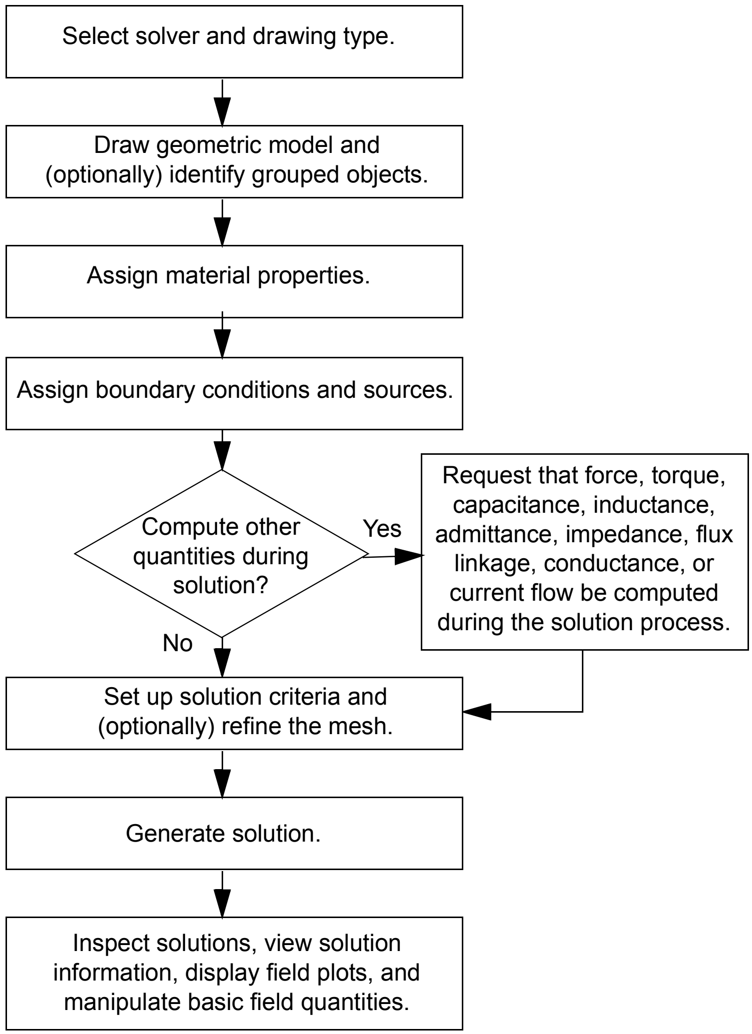
   Couple Model Allows you to define thermal coupling for a project.
4. Use the **Setup Materials** command to assign materials to all objects in the geometric model.
5. Use the **Setup Boundaries/Sources** command to define the boundaries and sources for the problem.
6. Use the **Setup Executive Parameters** command to instruct the simulator to compute the following special quantities:
   - Matrix (capacitance, inductance, admittance, impedance, or conductance matrix, depending on the selected solver)
   - Force
   - Torque
   - Flux Linkage
   - Current Flow
7. Use the **Setup Solution Options** command to specify how the solution is computed.
8. Use the **Solve** command to solve for the appropriate field quantities. For electrostatic problems, the simulator computes ϕ, the electric potential, from which it derives **E** and **D**.
9. Use the **Post Process** command to analyze the solution, as follows:
   - **Plot the field solution.** Common quantities (such as ϕ, **E**, and **D**) are directly accessible

from menus and can be plotted a number of ways. For instance, you can display a plot of equipotential contours or you can graph potential as a function of distance.

- **Use the calculators.** The post processor allows you to take curls, divergences, integrals, and cross and dot products to derive special quantities of interest.

The commands shown on the **Executive Commands** menu must be chosen in the sequence in which they appear. For example, you must first create a geometric model with the **Define Model** command before you specify material characteristics for objects with the **Setup Materials** command. A check mark appears on the menu next to the completed steps.

These steps are summarized in the following flowchart:



Results to Expect

After setting up the microstrip problem and generating a solution, you will:

- Plot and analyze voltage contours to view the voltage distribution around the conductors.
- Compute capacitance.

Time This guide should take approximately 3 hours to work through.

2

Creating the Microstrip Project

This guide assumes that Maxwell SV has already been installed as described in the Ansoft *PC Installation Guide*.

Your goals in this chapter are as follows:

- Create a project directory in which to save sample problems.
- Create a new project in that directory in which to save the microstrip problem.

Time This chapter should take approximately 15 minutes to work through.

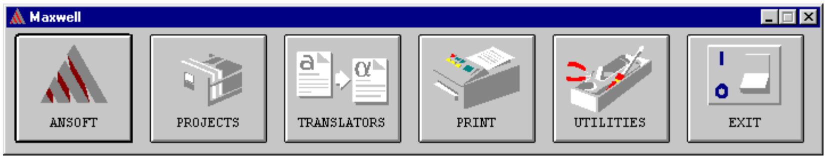
Access the Maxwell Control Panel

To access Maxwell SV, you must first open the Maxwell Control Panel, which allows you to create and open projects for all Ansoft projects.

To start the Maxwell Control Panel, do one of the following:

- Use the **Start** menu to select **Programs>Ansoft>Maxwell SV**.
- Double-click the **Maxwell SV** icon.

The Maxwell Control Panel appears. If not, refer to the Ansoft installation guides for assistance.

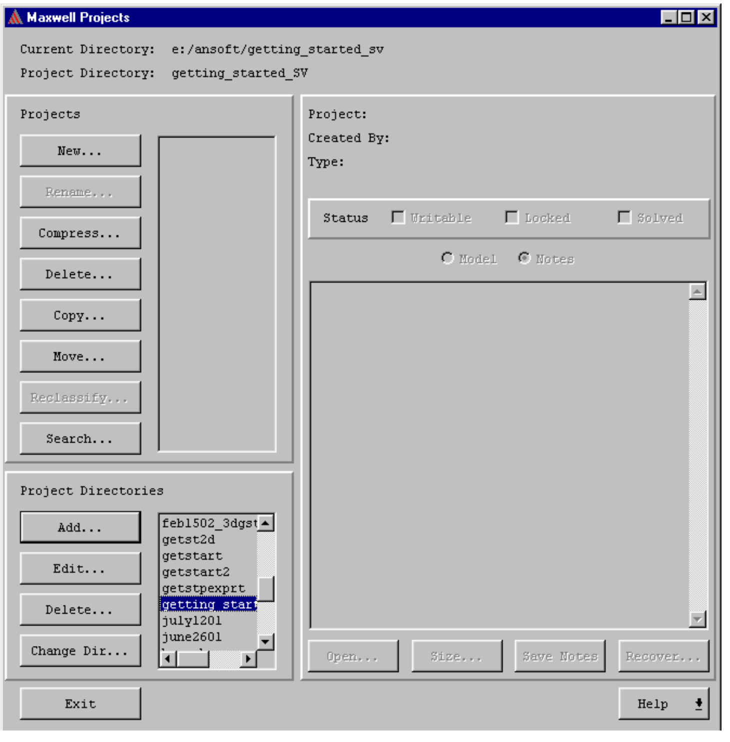


See the Maxwell Control Panel online help for a detailed description of the other options in the Maxwell Control Panel.

Access the Project Manager

The Project Manager enables you to create and manage Ansoft products. You can add new project directories, create projects in existing directories, and rename and copy projects.

To access the Project Manager, click **PROJECTS** from the Maxwell Control Panel. The Project Manager appears.



Create a Project Directory

The first step in using Maxwell SV to solve a problem is to create a project directory and a project in which to save all the data associated with the problem.

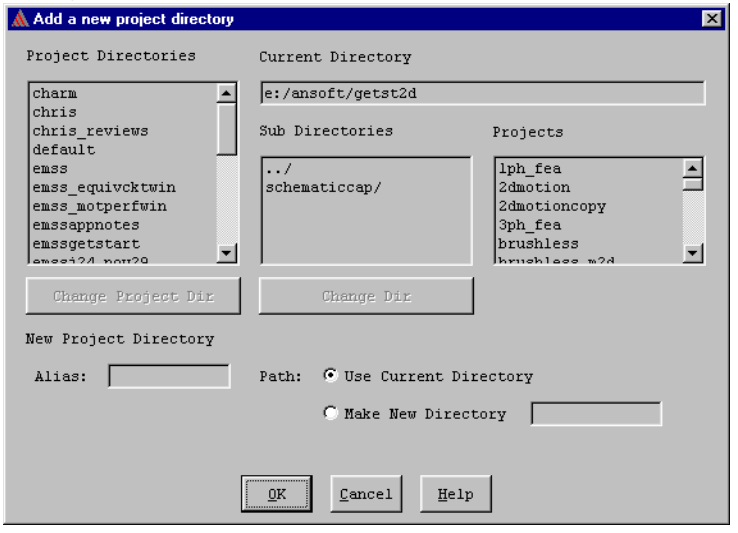
A project directory contains a specific set of projects created with the Ansoft software. You can use project directories to categorize projects any number of ways. For example, you might want to store all projects related to a particular facility or application in one project directory. You will now create a project directory.

The **Project Manager** should still be on the screen. You will add the **getting_started_SV** directory that will contain the Maxwell SV project you create using this *Getting Started* guide.

Note If you have already created a project directory while working through one of the other Ansoft *Getting Started* guides, skip to the "Create a Project" section.

To add a project directories, do the following in the Project Manager:

1. Click **Add** from the **Project Directories** list. The **Add a new project directory** window appears, listing the directories and subdirectories.



2. Type the following in the **Alias** field:

    `getting_started_SV`

    Maxwell SV projects are usually created in directories that have aliases — that is, directories that have been identified as project directories using the **Add** command.
3. Select **Make New Directory** near the bottom of the window. By default, **getting_started_SV** appears in this field.
4. Click **OK**.

The **getting_started_SV** directory is created under the current default project directory. You return to the **Project Manager**, and **getting_started_SV** now appears in the **Project Directories** list.

Create a Project

Now you are ready to create a new project named **microstrip** in the **getting_started_SV** project directory.

Access the Project Directory

Before you create the new project, access the **getting_started_SV** project directory.

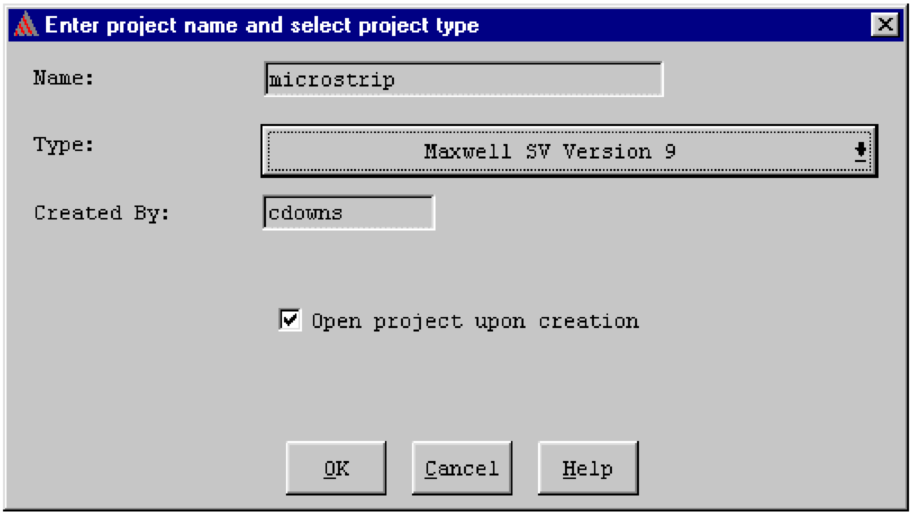
To access the project directory, click **getting_started_SV** in the **Project Directories** list of the Project Manager.

The current directory displayed at the top of the **Project Manager** menu changes to show the path name of the directory associated with the **getting_started_SV** alias. If you have previously created a model, it will be listed in the **Projects** list. Otherwise, the **Projects** list is empty — no projects have been created yet in this project directory.

Create the New Project

To create a new project:

1. In the Project Manager, click **New** in the **Projects** list. The **Enter project name and select project type** window appears.



2. Type **microstrip** in the **Name** field. Use the **Back Space** and **Delete** keys to correct typos.
3. If **Maxwell SV Version 9** does not appear in the **Type** field, do the following:
   a. Click the left mouse button on the software package listed in the **Type** field. A menu appears, listing all Ansoft software packages that have been installed.
   b. Click **Maxwell SV Version 9**.
4. Optionally, enter your name in the **Created By** field. The name of the person who logged onto the system appears by default.
5. Clear **Open project upon creation**. You do not want to automatically open the project at this point, so that you may enter project notes first.
6. Click **OK**.

The information that you just entered is now displayed in the corresponding fields in the **Projects** list. Because you created the project, **Writable** is selected, showing that you have access to the project.

Save Project Notes

It is a good idea to save notes about your new project so that the next time you use Maxwell SV, you can view information about a project without opening it.

To enter notes for the **microstrip** problem:

1. Leave **Notes** selected by default.
2. Click in the area under the **Notes** option. This places an I-beam cursor in the upper-left corner of the **Notes** area, indicating that you can begin typing text.

Note The **Model** option displays a picture of the selected model in the **Notes** area. It is disabled now because you are creating a new project. After you create the **microstrip** problem, its geometry will appear in this area by default when the **microstrip** project is selected. For a detailed description of the **Model** option, refer to the Maxwell Control Panel online help.

3. Enter your notes about the project, such as the following:

   ```
   This is the sample electrostatic problem created using
   Maxwell SV and the Electrostatic Getting Started guide.
   ```

   When you start entering project notes, the **Save Notes** button (located below the **Notes** area) becomes black, indicating that it is enabled. Before you began typing in the **Notes** area, **Save Notes** was grayed out, or disabled.
4. When you are done entering the description, click **Save Notes** to save it. After you do, the **Save Notes** button is disabled again.

Now you are ready to open the new Maxwell SV project and run the software.

3

Accessing the Software

In the last chapter, you created the **getting_started_SV** project directory and created the **microstrip** project within that directory.

This chapter describes:

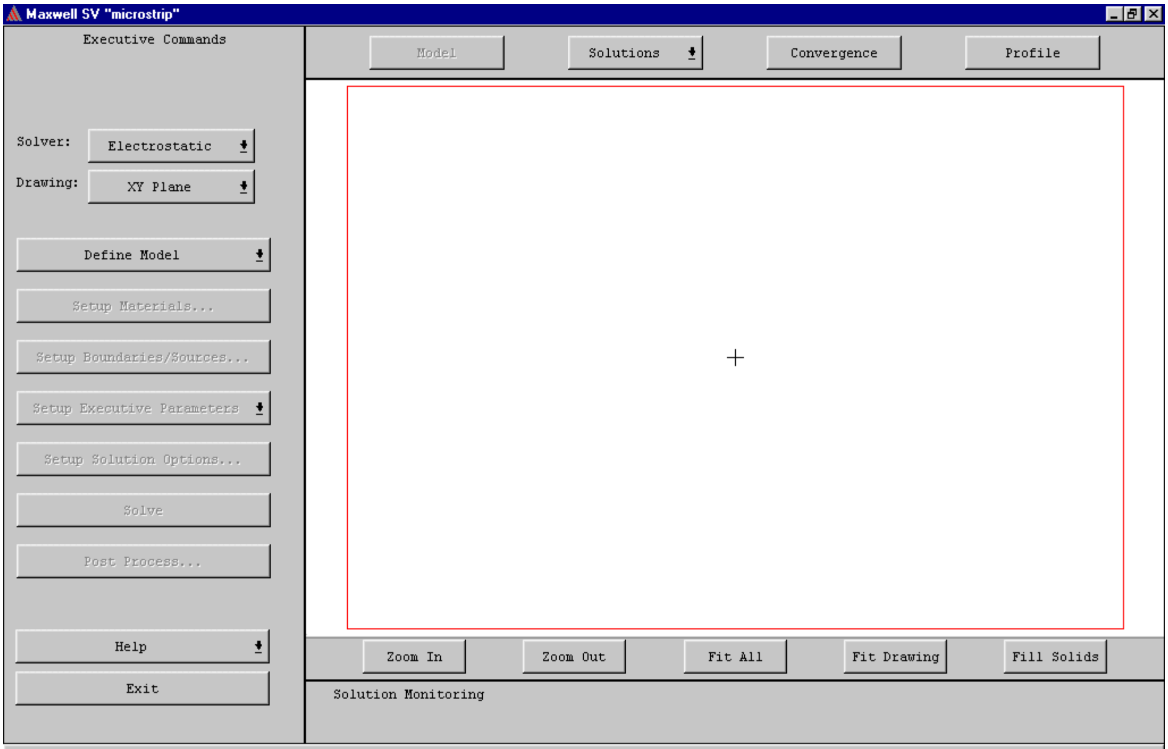
- How to open the project you just created and run Maxwell SV.
- The Maxwell SV **Executive Commands** window.
- The general procedure for creating an electrostatic problem in Maxwell SV.
- The sample problem and the procedures you will use to simulate its electric fields.

Time — This chapter should take approximately 10 minutes to work through.

Open the New Project and Run the Simulator

The newly created **microstrip** project should still be highlighted in the **Projects** list. (If it is not, click the left mouse button on it in the list.)

To run Maxwell SV, click **Open** in the project area. The Maxwell SV **Executive Commands** menu appears.



Executive Commands Window

The **Executive Commands** window is divided into three sections: the **Executive Commands** menu, display area, and the solution monitoring area.

Executive Commands Menu

The **Executive Commands** menu acts as a doorway to each step of creating and solving the model problem. You select each module through the **Executive Commands** menu, and the software returns you to this menu when you are finished. You also view the solution process through this menu.

Display Area

The display area shows either the project's geometry in a model window or the solution to the problem once a solution has been generated. Since you have not yet drawn the model, this area is blank. The commands along the bottom of the window allow you to change your view of the model:

Zoom In	Zooms in on an area of the window, magnifying the view.
Zoom Out	Zooms out of an area, shrinking the view.
Fit All	Changes the view to display all items in the window. Items will appear as large as possible without extending beyond the window.
Fit Drawing	Displays the entire drawing space.
Fill Solids	Displays objects as solids rather than outlined objects. Toggles with **Wire Frame**.
Wire Frame	Displays objects as wire-frame outlines. Toggles with **Fill Solids**.

The buttons along the top of the window are used when you are generating and analyzing a solution. These are described in more detail in Chapter 6, "Generating a Solution."

Solution Monitoring Area

This area displays solution profile and convergence information while the problem is solving, as described in Chapter 6, "Generating a Solution."

4

Creating the Model

In the last chapter, you opened the **microstrip** project, examined the **Executive Commands** window, and reviewed the procedure for creating a 2D model. Now you are ready to use Maxwell SV to solve an electrostatic problem. The first step is to create the geometry for the system being studied.

This chapter shows you how to create the geometry for the microstrip problem that was described in Chapter 1, “Introduction,” and in Chapter 3, “Accessing the Software.”

Your goals for this chapter are as follows:

- Set up the problem region.
- Create the objects that make up the geometric model.
- Save the geometric model to a disk file.

Time This chapter should take approximately 35 minutes to work through.

Specify Solver Type

The Maxwell SV **Executive Commands** window should still be on the screen. Before you start drawing your model, you need to specify which field quantities to compute. By default, **Electrostatic** appears as the **Solver** type. Because you will be solving an electrostatic problem, leave this type as it appears.

Specify Drawing Plane

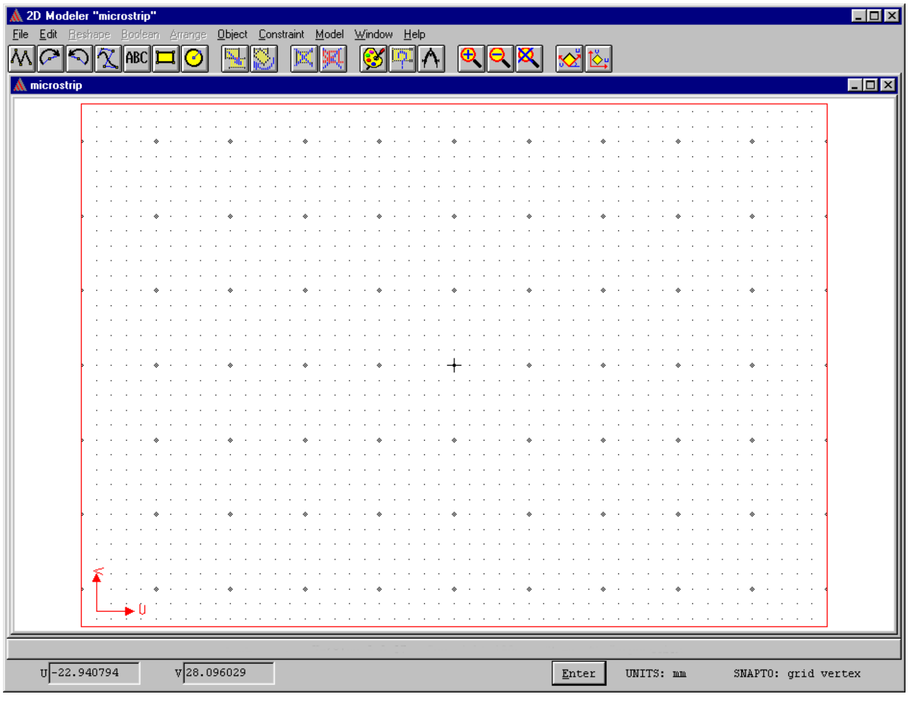
The microstrip model you will be drawing is actually the XY cross-section of a structure that extends into the z-direction. This is known as a cartesian or XY plane model. By default, **XY Plane** appears as the **Drawing** plane. Because the model you will be creating is in the XY plane, leave this type as it appears.

Now, you are ready to draw the model.

Access the 2D Modeler

To draw the geometric model, use the 2D Modeler, which allows you to create 2D structures.

To access the 2D Modeler, click **Define Model>Draw Model**. The 2D Modeler appears.



Screen Layout

The following sections provide a brief overview of the 2D Modeler.

Menu Bar

The 2D Modeler's menu bar appears at the top of the window. Each item in the menu bar has a menu of commands associated with it. If a command name has an arrow next to it, that command also has a menu of commands associated with it. If a command name has an ellipsis next to it, that command has a window or panel associated with it.

To display the menu associated with a command in the menu bar, do one of the following:

- Click on it.
- Press and hold down **Alt**, and then press the key of the underlined letter on the command name.

For example, to display the **Window** command's menu, do one of the following:

- Click it.
- Press **Alt**, and type a **W**.

The **Window** menu appears. Click outside the **Window** menu to make it disappear.

Project Window

The main 2D Modeler window contains the Drawing Region, the grid-covered area where you draw the objects that make up your model. This main window in the 2D Modeler is called the project window. A project window contains the geometry for a specific project and displays the project's name in its title bar. By default, one subwindow is contained within the project window.

Optionally, you can open additional projects in the 2D Modeler clicking the **File>Open** command. Opening several projects at once is useful if you want to copy objects between geometries. See the Maxwell 2D online help for more details.

Subwindows

Subwindows are the windows in which you create the objects that make up the geometric model. By default, this window:

- Has points specified in relation to a local uv coordinate system. The u-axis is horizontal, the v-axis is vertical, and the origin is marked by a cross in the middle of the screen.
- Uses millimeters (mm) as the default drawing unit.
- Has grid points 2 millimeters apart. The default window size is 100 millimeters by 70 millimeters.

Note If a geometry is complex, you may want to open additional subwindows for the same project so that you can alter your view of the geometry from one subwindow to the next. To do so, use the **Window>New** command as described in the Maxwell 2D online help. For this geometry, however, a single subwindow should be sufficient.

Status Bar

The status bar, which appears at the bottom of the 2D Modeler screen, displays the following:

U and **V** Displays the coordinates of the mouse's position on the screen and allows you to enter coordinates using the keyboard.

UNITS Drawing units in which the geometry is entered.

SNAPTO Which point — grid point or object vertex — is selected when you choose points using the mouse. By default, both grid and object points are selected so that the mouse snaps to whichever one is closest.

Message Bar

A message bar, which appears above the status bar, displays the functions of the left and right mouse buttons. When selecting or deselecting objects, the message bar displays the number of items that are currently selected. When you change the view in a subwindow, it displays the current magnification level.

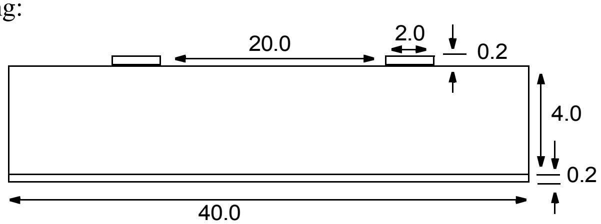
Toolbar

The toolbar is the vertical stack of icons that appears on the left side of the 2D Modeler screen. Icons give you easy access to the most frequently used commands that allow you to draw objects, change your view of the problem region, deselect objects, and so on. Click an icon and hold the button down to display a brief description of the command in the message bar at the bottom of the screen.

For more information on these areas of the 2D Modeler, refer to the Maxwell 2D online help.

The Microstrip Model

Use the following dimensions, which are given in mils, to create the geometric model of the microstrip you are modeling:



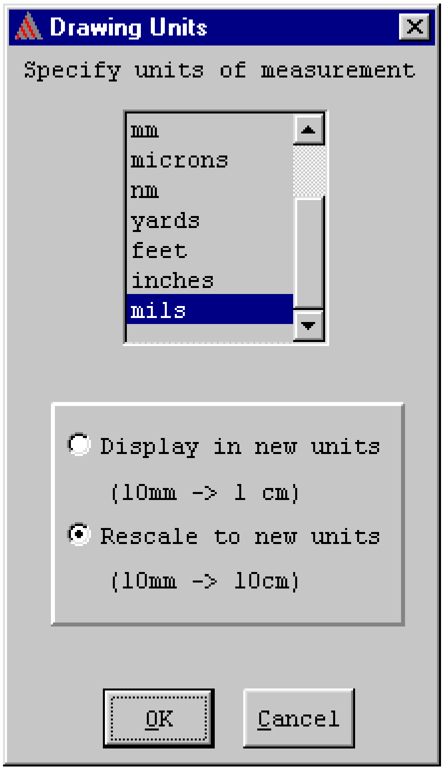
Note In this example, the default drawing size is appropriate. However, if your model were much larger or smaller, you would need to change the model size by clicking the **Model>Drawing Size** command. In general, the width of the drawing area should be three to five times longer than the width of the geometric model, and the height of the drawing area should be three to five times longer than the height of the geometric model.

Set Up the Drawing Region

The first step is to specify the drawing units to use in creating the model.

To set up the drawing region:

1. Click **Model>Drawing Units**. The **Drawing Units** window appears.



2. Select **mils** from the list, and then click the **Rescale to new units** radio button, if it is not already selected.
3. Click **OK**.

The units are changed from millimeters to mils, and **mils** now appears next to **UNITS** in the status bar at the bottom of the modeling window.

Create the Geometry

Now you are ready to draw the objects that make up the geometric model.

Keyboard Entry

In the following section, several of the points lie *between* grid points. You can position these points in one of two ways:

- Change the grid spacing so that the object's dimensions lie on grid points.
- Use "keyboard entry" — that is, enter the coordinates directly into the **U** and **V** fields in the status bar.

If you change the grid spacing, the screen may become cluttered with too many tightly-spaced grid points and make point selection difficult. Therefore, use keyboard entry to enter several of the dimensions of the sample geometry.

Note To change the grid spacing, click the **Window>Grid** command.

To change the size of the problem region, click the **Model>Drawing Size** command.

Create the Substrate

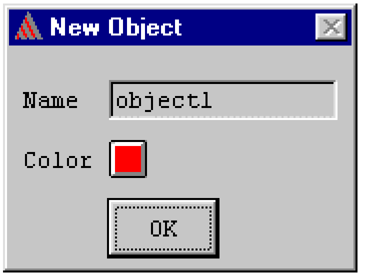
The substrate consists of a simple rectangle on which the two rectangular microstrips will rest.

Draw a Rectangle

To create the rectangle to represent the substrate:

1. Click **Object>Rectangle**. After you do, the pointer changes to crosshairs.
2. Select the first corner of the substrate, the upper-left corner, as follows:
   a. Move the crosshairs to the point on the grid where the u- and v- coordinates are (**-20**, **0**). Remember that the coordinates of the cursor's current location are displayed in the **U** and **V** fields in the status bar.
   b. Click the point to select it.
3. To select the second corner of the substrate, use keyboard entry because **-4**, the v-coordinate, lies between grid points. To specify the lower-right corner:
   a. Double-click in the **U** field in the status bar.
   b. Type **20**, using the **Backspace** and **Delete** keys to correct typos.
   c. Press **Tab** to move to the **V** field in the status bar. The value in the **dU** field changed. The **dU** and **dV** fields display the values of the offset from the previous point.
   d. Type –**4**.
4. To accept the point, press **Return** or click **Enter** in the status bar. After you do, the **New**

Object window appears.



Assign a Name and Color

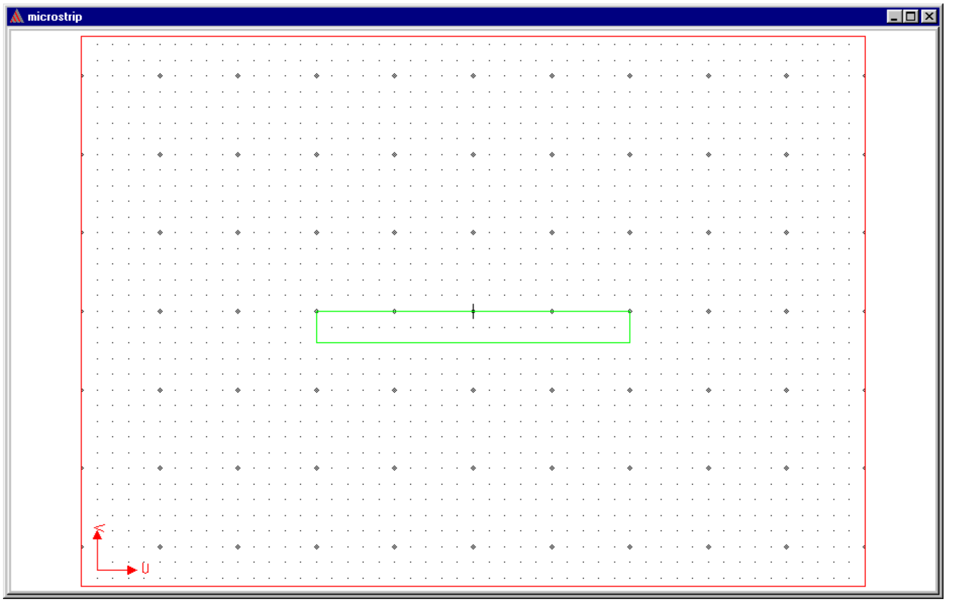
By default, the object that you are creating is assigned the name **object1** and the color red, and the **Name** field is selected.

Note Be sure to change the name of the object to indicate its function and to assign a different color to different objects. This will be important later when you assign boundary conditions, voltages, and so forth.

To define the name and color for the substrate:

1. Type **substrate** in the **Name** field. Do not press **Return**.
2. Click the solid red square next to **Color**. A palette of 16 colors appears.
3. Click one of the green boxes in the palette.
4. Click **OK**.

The object now appears in the drawing region as shown below. It is green and has the name **substrate**:



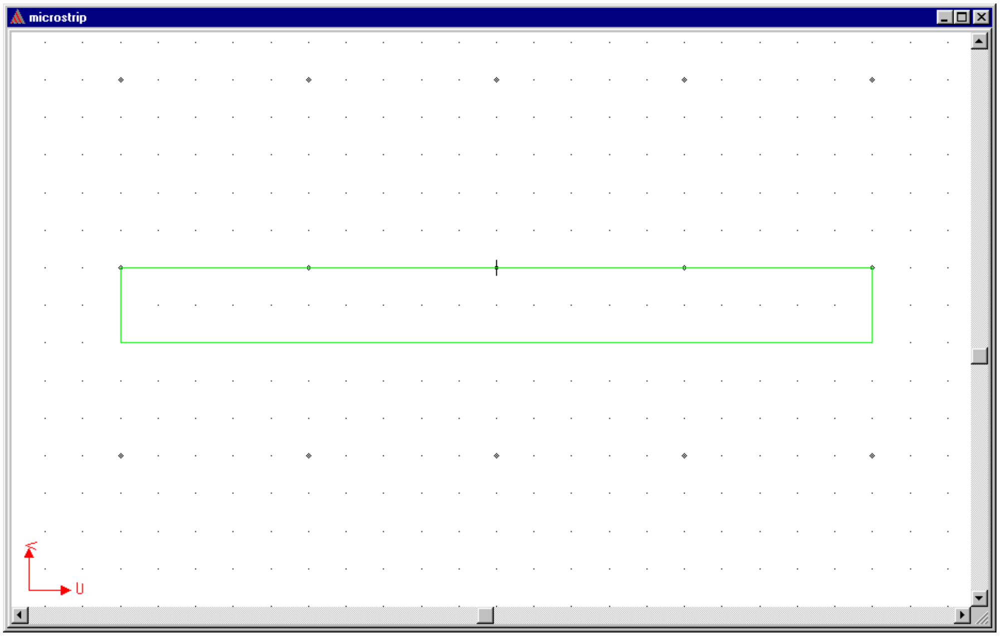
Create the Left Microstrip

Now that you have drawn the substrate, draw the left microstrip. Because the microstrip is relatively small compared to the substrate, you need to zoom in on the top part of the substrate and then use keyboard entry to draw it.

Zoom In on Top of the Substrate

To zoom in on the top of the substrate:

1. Click **Window>Change View>Zoom In**. The pointer changes to crosshairs.
2. Click on the point on the grid that is slightly to the left of the upper-left corner of the substrate and one grid point above it.
3. Move the crosshairs to the point on the grid that is slightly to the right of the upper-right corner of the substrate and one grid point below it. As you move the crosshairs, the system draws a box on the screen that encloses the selected area.
4. Click to select the point. The area you selected is enlarged to fill the **2D Modeler** window, as shown below.



Draw a Rectangle

Create a rectangle to represent the microstrip.

To create the rectangle:

1. Click **Object>Rectangle**.
2. Select the first corner of the microstrip, the upper-left corner, by entering the following coordinates using the keyboard:

   U **-12** Press **Tab**.

   V **0.2** Press **Return**.

3. Select the second corner of the microstrip, the lower-right corner, by entering the following coordinates as you did in the previous step:

   U **-10**

   V **0**

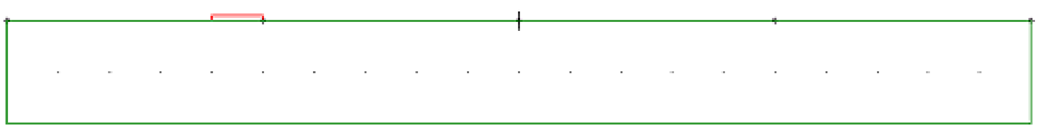
   The offset values **dU** and **dV** change as you enter the points. The **New Object** window appears.

Assign a Name and Color

To define the name and color for the left microstrip:

1. Change the object's **Name** to **left**.
2. Leave the object's **Color** the default color of red.
3. Click **OK**.

The substrate and the left microstrip appear as shown below:



Create the Right Microstrip

Because the left and right microstrip have the same dimensions, create the right microstrip by copying the left one.

Duplicate the Left Microstrip

To create the right microstrip:

1. Click the left microstrip to select it as the object to be copied. A double outline appears around it, indicating that it has been selected.

Note As an alternative to selecting an object by clicking on it, use the **Edit>Select** commands. After an object or objects are selected, they are the objects on which all other **Edit** commands are carried out.

2. Click **Edit>Duplicate>Along Line**. You must now select two points: first an "anchor" point,

and then a “target” point, which will be the new location for the anchor point.

3. Click the lower-left corner of the left microstrip as the anchor point. After you do, two new fields appear in the status bar: **dU** and **dV**. These fields allow you to select the target point by specifying its offset from the anchor point rather than its u- and v- coordinates.
4. Enter **22** in the field **dU** to specify the offset between the anchor and target points.
5. Press **Return** or click **Enter**. The **Linear Duplicates** window appears.
6. Leave the default **2** in the **Total Number** field.
7. Click **OK** to accept the value and complete the command.

Now both microstrips have been created. By default, the new object — the right microstrip — is the selected object.

Assign a Name and Color

The 2D Modeler automatically assigns names to copied objects by appending a number to the end of the original object’s name. For instance, the right microstrip is assigned the name **left1** because it is the first copy of the object **left**. To assign meaningful names to the object, change the name of **left1** to **right**.

To rename the right microstrip:

1. Click **Edit>Attributes>Rename**. The **Rename Selected Objects** window appears, listing the names of all selected objects. Because **left1** is the only selected object, it appears in the field beneath the **Object** list.
2. Change the name **left1** that appears below the list box to **right**.
3. Click **Rename**. The new name now appears in the **Object** list at the top of the window.
4. Click **OK.**

Since you will be creating other objects, you should deselect the right microstrip.

To deselect the right microstrip:

- Click **Edit>Deselect All**. If a submenu appears, select either **Current Project** or **All Projects**.

The microstrip is deselected.

Save the Geometry

Maxwell SV does not automatically save your work. Therefore, periodically save the geometry to a set of disk files while you are working on it. If you have saved your files and a problem occurs that causes an unexpected abort, you will not have to re-create the model.

To save the microstrip model now:

- Click **File>Save**. The pointer changes to a watch while the geometry is written to files. When the pointer reappears, the geometry has been saved in a disk file in the **microstrip.pjt** directory.

Create the Ground Plane

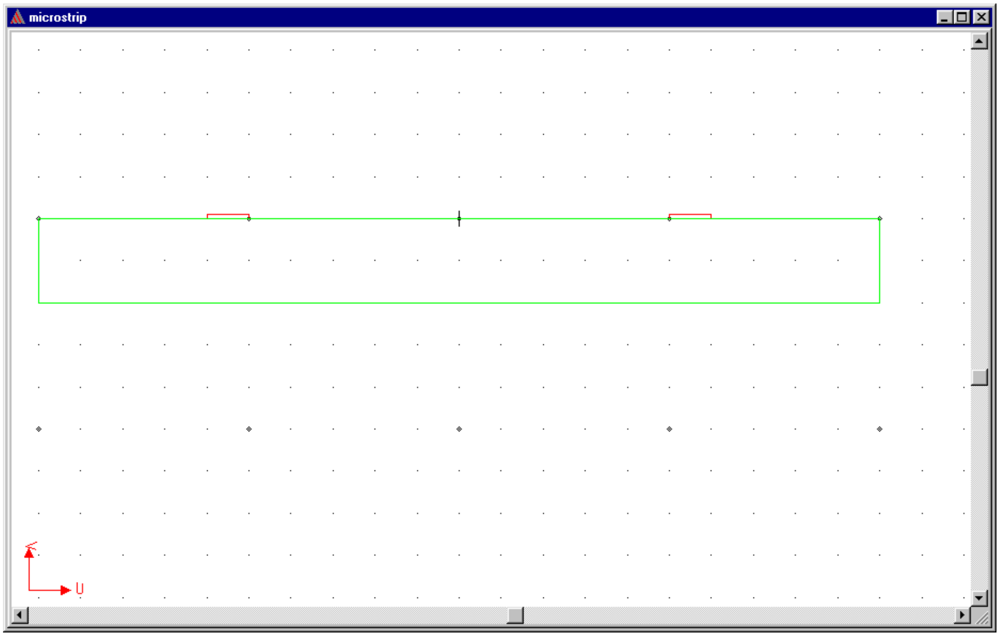
The last object that you need to create is the ground plane.

Zoom In on the Bottom of the Substrate

Before you zoom in on the bottom of the substrate, you must return to a pre-zoomed view of the drawing region as follows:

To zoom out of the drawing:

1. Click **Window>Change View>Fit Drawing**. After you do, all objects are displayed in the drawing region.
2. Click **Window>Change View>Zoom In**. You can also click the  toolbar icon.
3. Select a point slightly to the left and one grid point above the lower-left corner of the substrate.
4. Select a point slightly to the right and one grid point below the lower-right corner of the substrate. The selected area is enlarged, along with the rest of the model:



Displaying Zoomed Models

You can also use the scroll bars on the right side and bottom of the subwindow to change your view. Scroll bars appear only when the entire geometric model is not displayed in the window.

To change your view, do one of the following:

- Click the arrow buttons at the top and bottom of the scroll bar.
- To scroll through the model:
  1. Move the cursor to the off-colored bar, or “thumb scroll,” visible on the scroll bar.
  2. Drag the thumb scroll up, down, left, or right in the scroll bar to the portion of the model that you want to display.

For instance, to pan down a geometric model, drag the thumb scroll in the vertical scroll bar down. If the portion of the geometry in which you are interested does not appear, continue to manipulate the thumb scrolls until it does.

Draw a Rectangle

The **Object>Rectangle** command is generally used to create rectangular objects, such as the ground plane, while the **Object>Polyline** command is used to create lines and other simple closed shapes. However, to illustrate its use, **Object>Polyline** is used in this section to create a rectangle.

To create the ground plane:

1. Click **Object>Polyline**. To create a rectangle using this command, you will select four points representing the four corners of the rectangle.
2. Select the lower-left corner of the substrate (–**20**, –**4**) as the first point of the rectangle.
3. Select the lower-right corner of the substrate (**20**, –**4**) as the second point in the rectangle. The 2D Modeler draws a line connecting the points.
4. Use the **U** and **V** fields to enter the following coordinates for the third point on the rectangle:
   U 20

   V -4.2
5. Use the **U** and **V** fields to enter the following coordinates for the fourth point on the rectangle:
   U -20

   V -4.2

   Again, the offset distances change as you enter the coordinates of the second point.
6. Select the first point — the lower-left corner of the substrate (–**20**, –**4**) — twice, either by double-clicking or by entering the values and pressing **Enter** twice. The **New Objects** window appears.

Assign a Name and Color

To define the name and color for the left microstrip:

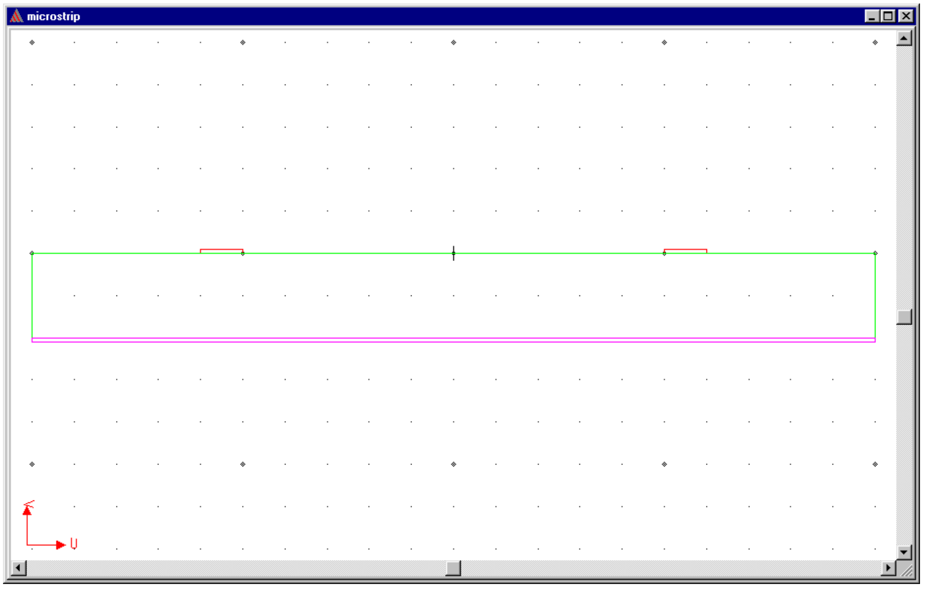
1. Change the object’s **Name** to **ground**.
2. Change the object’s **Color** to magenta.
3. Click **OK** to complete the command.

The object **ground** occupies a thin region along the bottom edge of the substrate.

Completed Geometry

The geometric model is now complete.

Click **Window>Change View>Fit All** to fit all objects on the screen and make them appear as large as possible. Your completed geometry should now resemble the one shown below.



Exit the 2D Modeler

To exit the 2D Modeler:

1. Click **File>Exit**. A window appears, prompting you to save the changes before exiting.
2. Click **Yes**. The geometry is saved to a disk file in the **microstrip.pjt** project directory, and the **Executive Commands** window appears. A check mark appears next to **Define Model**, indicating that this step has been completed.

Note Because none of the objects are electrically connected at any point in a 3D rendering of the model, you do not need to use the **Define>Model>Group Objects** command.

5

Defining Materials and Boundaries

Now that you have drawn the geometry for the microstrip problem and returned to the **Executive Commands** window, you are ready to set up the problem.

Your goals for this chapter are as follows:

- Assign material attributes to each object in the geometric model.
- Define any boundary conditions that need to be specified, such as the behavior of the electric field at the edge of the problem region, and potentials on the surfaces of the microstrips and ground plane.

Time This chapter should take approximately 35 minutes to work through.

Set Up Materials

To define the material properties for the objects in the geometric model, you must:

- Assign the properties of a perfect conductor to both microstrips and the ground plane.
- Assign FR4-epoxy, a dielectric commonly used in circuit boards, to the substrate.

In general, to assign materials to objects:

1. If necessary, add materials with the properties of the objects in your model to the material database.
2. Assign a material to each object in the geometric model as follows:
   a. Select the object(s) for which a specific material applies.
   b. Select the appropriate material.
   c. Click **Assign** to assign the selected material to the selected object(s).

In this sample problem, you do not have to add materials to the material database — all materials that you will need are already included in the global material database that the simulator makes available to every project.

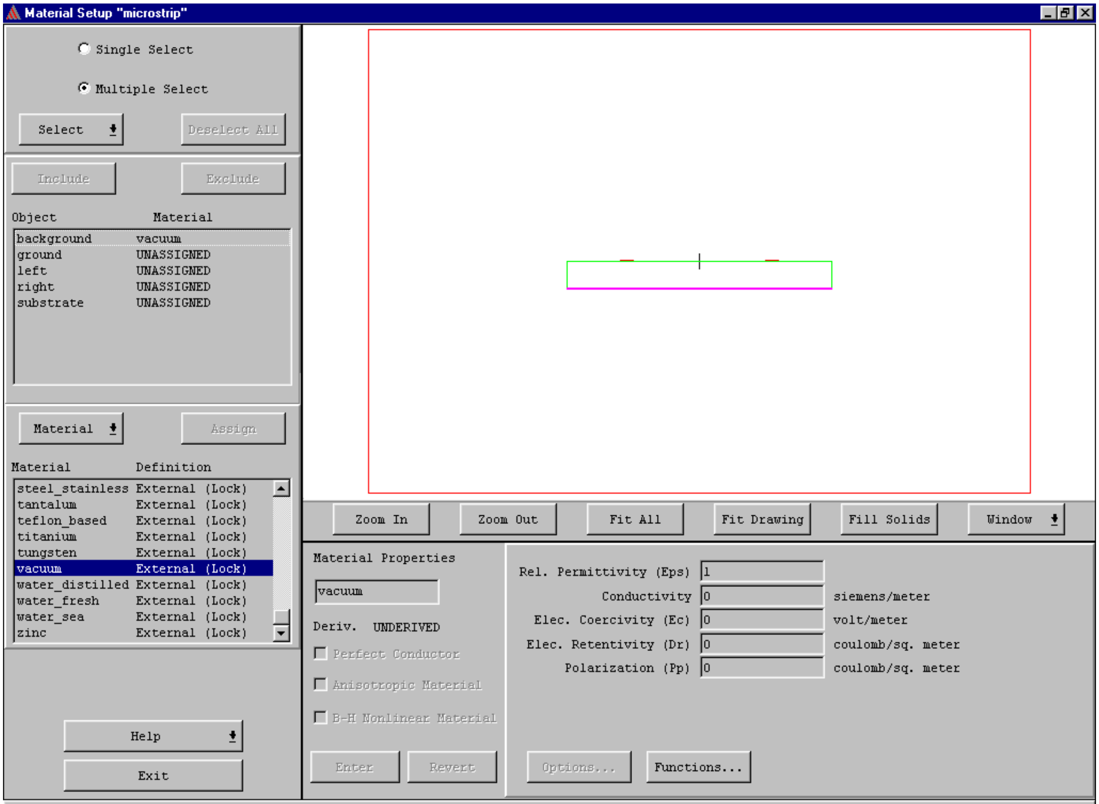
Note You must assign a material to each object in the model.

Access the Material Manager

To access the Material Manager:

1. Click **Setup Materials**. A warning window appears, explaining that materials with a conductivity greater than 10000 siemens per meter will be treated as perfect conductors and will be excluded from the solution region. Materials with lower conductivities will be included in the solution region; the conductivities of such materials will not effect the electrostatic simulation, since no current flow is modeled.

Note For highly conductive materials such as copper, the potential is the same across an object that is assigned the material. Because of this, Maxwell 2D’s electrostatic field solver does not generate a solution inside objects assigned these materials. Instead, it treats them as though they were perfectly conducting.



Assign a Material to the Substrate

Now assign a material to the substrate.

To assign a material to the substrate:

1. Click **substrate** in the **Object** list, or click on the substrate object in the geometric model.
2. Click **FR4_epoxy** in the **Material** list. If it does not appear in the list, use the scroll bars to scroll through the list as described in Chapter 4, "Creating the Model." Capitalized material names are listed first.
3. Click **Assign**.

FR4_epoxy now appears next to **substrate** in the **Object** list.

Assign a Material to the Microstrips and Ground Plane

Now you can assign materials to the microstrips and ground. which are the conductors.

To assign materials to the conductors:

1. Click **Multiple Select** at the top of the window, if it is not already enabled.
2. Do one of the following to select **left**, **right**, and **ground** from the **Object** list:
   - Press and hold down **Ctrl**, and then click each of the object names.
   - Press and hold down **Shift**, and then drag the pointer over the object names.

   To deselect an object, click it.
3. Click **perf_conductor** in the **Material** list.
4. Click **Assign**.

The microstrips and the ground plane have now been assigned the properties of a perfect conductor (a good approximation of which is copper). Also, **perf_conductor** appears next to those objects' names.

Note The potentials on the surfaces of conductors are specified with the **Setup Boundaries/ Sources** command that is described later in this chapter.

Assigning Materials to the Background

The **background** object is the only object that is assigned a material by default. Include it as part of the problem region in which to generate the solution. When a material name — such as **vacuum** — appears next to **background** in the **Objects** list, the **background** object is included as part of the solution region.

Because the model is assumed to be surrounded by a vacuum, accept the default material, **vacuum**, for the **background**.

Note In some cases, such as when all objects and electromagetic fields of interest are contained within an enclosure, including the background as part of the problem region wastes computing resources. It also prevents you from setting boundary conditions defining an external electric or magnetic field for the model.

In these cases, you can manually exclude the background from the solution.

Exit the Material Manager

Now that you ave assigned materials to the objects, exit the Material Manager and return to the **Executive Commands** window where you will continue setting up the project.

To exit the **Material Setup** window:

1. Click **Exit** at the bottom-left of the **Material Setup** window. A window with the following prompt appears:

   ```
   Save changes before closing?
   ```

2. Click **Yes**.

You are returned to the **Executive Commands** window. A check mark now appears next to **Setup Materials**, and **Setup Boundaries/Sources** is enabled.

Note If you exit the Material Manager before excluding or assigning a material to each object in the model, a check mark does not appear next to **Setup Materials** on the **Executive Commands** menu, and **Setup Boundaries/Sources** remains disabled.

Set Up Boundaries and Sources

After setting material properties, the next step in creating the microstrip model is to define boundary conditions and sources.

Initially, all object surfaces are defined as natural boundaries, which simply means that **E** is continuous across the surface. All outside edges are defined as Neumann boundaries, which means that the tangential components of **E** and the normal components of **D** are continuous across the surface.

To finish setting up the microstrip problem, you need to explicitly define the following:

- The voltages on the two microstrips and the ground plane.
- The behavior of the electric field on all surfaces exposed to the area beyond the problem region. Because you included the background as part of the problem region, this exposed surface is that of the **background** object.

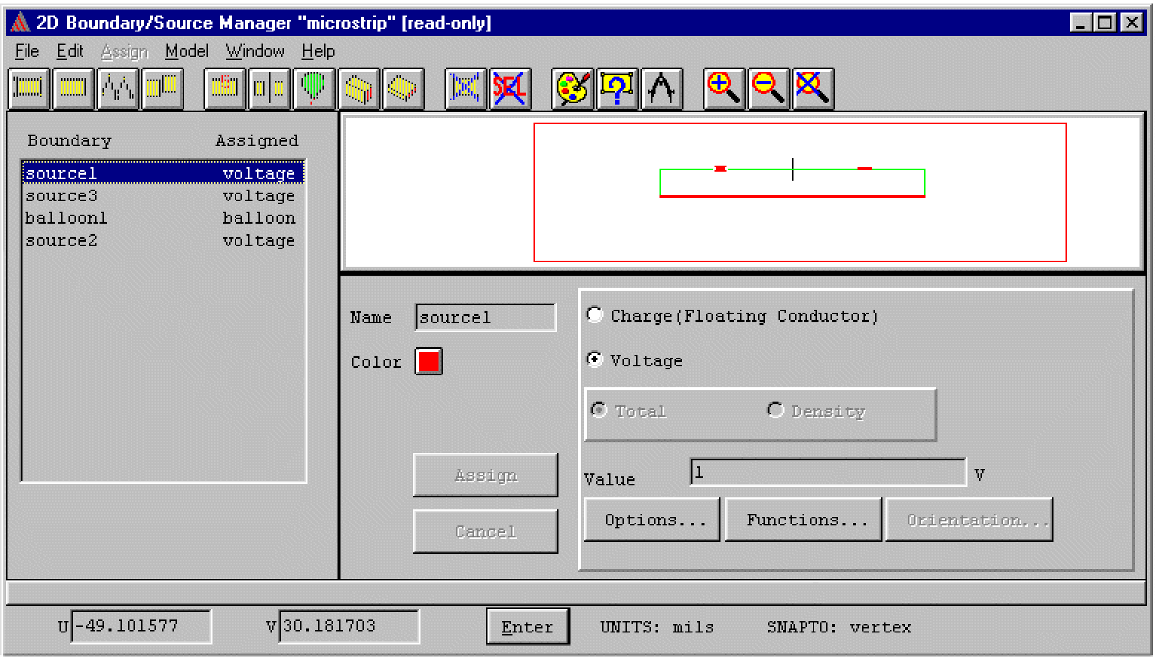
Maxwell SV is unable to compute a solution for the model unless you specify some source of electric field. In this model, the microstrips serve as sources of electric potential.

Note **Setup Boundaries/Sources** will not have a check mark, and the simulator will not attempt to solve the problem unless the potential on at least one object's surface has been explicitly defined using the 2D Boundary/Source Manager.

Display the 2D Boundary/Source Manager

To display the Boundary Manager:

- Click **Setup Boundaries/Sources**. The 2D Boundary/Source Manager appears.



Boundary Manager Screen Layout

The Boundary Manager is divided into several sections, each displaying information about a particular property of the model and its boundaries.

Boundary

Each time you choose one of the **Assign>Boundary** or **Assign>Source** commands, an entry is added to the **Boundary** list on the left side of the window. When you first access the 2D Boundary/ Source Manager, the **Boundary** list is empty.

Display Area

The geometric model is displayed so that you can select the objects or edges to be used as boundaries or sources, using the **Edit>Select** commands.

Boundary/Source Information

The area that appears below the geometric model allows you to assign boundaries and sources to objects and surfaces and display the parameters associated with the selected boundary or source.

Types of Boundary Conditions and Sources

There are two types of boundary conditions and sources that you will use in this problem:

Balloon boundary	Can only be applied to the outer boundary, and models the case in which the structure is infinitely far away from all other electromagnetic sources.
Voltage sources	Specifies the voltage on an object in the model. The electric scalar potential, ϕ, is set to a constant value, forcing the electric field to be perpendicular to the objects' surfaces.

You must assign boundary conditions and sources to the following objects in the microstrip geometry:

Left microstrip	This surface is to be set to 1 volt.
Right microstrip	This surface is to be set to −1 volt.
Ground plane	The grounded reference. This surface is to be set to zero volts.
Background	The outer boundary of the problem region. This surface is to be ballooned to simulate an electrically insulated system.

Before you identify a boundary condition or source, you must first identify the surface to which the condition is to be applied. You will select and then assign boundaries and sources for the following objects:

- **left**
- **right**
- **ground**
- **background**

There are several ways to select objects' surfaces, but in this sample problem you will select each object individually. As a result, the object's *surface* will be selected. There are also several ways to assign values to surfaces. The sample problem illustrates two ways to do so.

Set the Voltage on the Left Microstrip

Because the microstrips appear very small, you must first make the model appear larger before assigning the voltages. Then you need to select each microstrip and assign a voltage of 1 volt.

To select the left microstrip and assign the voltage:

1. Zoom in on the two microstrips:
   a. Click **Window>Change View>Zoom In**. The cursor changes to crosshairs.
   b. Select two corners of a rectangle that encloses both microstrips.
2. Click **Edit>Select>Object>By Clicking**. The menu bar commands are disabled, and the system expects you to select an item by clicking on it in the model.
3. Click on the left microstrip. After you do, it is highlighted.
4. Right-click anywhere in the display area to stop selecting objects. The commands in the menu bar are enabled again, and the left microstrip is the only highlighted object on the screen. Now you are ready to assign a voltage to the surface of the left microstrip.

> **Note** If the appropriate object is not highlighted, or if more than one object is highlighted, do the following:
>
> 1. Click **Edit>Deselect All**. After you do, no objects are highlighted.
> 2. Click **Edit>Select>Object>By Clicking**, and select the object.

5. Click **Assign>Source>Solid**. The name **source1** appears in the **Boundary** list, and **NEW** appears next to it, indicating that it has not yet been assigned to an object or surface.
6. In the properties section below the model diagram, verify that **Voltage** is selected.
7. Change the **Value** field to **1** V.
8. Click **Assign**. A value of one volt has now been specified for the left microstrip, and **voltage** replaces **NEW** next to **source1** in the **Boundary** list.

Set the Voltage on the Right Microstrip

Now set the voltage on the right microstrip to -1 volt.

To select the left microstrip and assign the voltage:

1. Click **Edit>Select>Object>By Clicking**.
2. Click on the right microstrip.
3. Right-click anywhere in the display area to stop selecting objects.
4. Click **Assign>Source>Solid**. The name **source2** appears in the **Boundary** list, and the source information appears below the model.
5. Verify that **Voltage** is selected.
6. Change the **Value** field to **−1** volt.
7. Click **Assign**. A value of **−1** volt has been specified for the right microstrip, and **voltage** replaces **NEW** next to **source2**.

Set the Voltage on the Ground Plane

Now set the voltage on the ground plane to 0 volts.

To select the ground and assign a voltage:

1. Click **Window>Change View>Fit All** to make all objects — including the object representing the ground plane — appear as large as possible in the subwindow.
2. Click **Edit>Select>Object>By Name**. A prompt with the following message appears:

   ```
   Enter item name/regular expression
   ```
3. Enter **ground**, and click **OK**. The ground appears highlighted in the model.
4. Click **Assign>Source>Solid**. The name **source3** appears in the **Boundary** list, and the source information appears below the model.
5. Verify that **Voltage** is selected.
6. Verify that the **Value** field is set to **0** volts.
7. Click **Assign**. Now the voltage has been specified for the ground plane, and **voltage** replaces **NEW** next to **source3**.

Assign a Balloon Boundary to the Background

The balloon boundary extends the object to which it is assigned infinitely far away from all other sources in all directions. Since the structure of the microstrip problem is an electrically insulated system, the background should be ballooned.

Note For this sample problem, all surfaces of the background are ballooned. Thus, you select **background** to pick its *entire* surface before ballooning it. If you create only part of an electromagnetically symmetrical model, at least one surface — the one representing the symmetry plane — would not be ballooned. In such a case, do not select the object's entire surface using the **Edit>Select>Object** commands. Instead, use the **Edit>Select>Edge** command, described in the Maxwell 2D online help, to select the three edges to balloon separately from the edge representing the symmetry plane.

To select the background and assign a balloon boundary:

1. Click **Window>Change View>Fit Drawing** so that the limits of the drawing region are displayed.
2. Click **Edit>Select>Object>By Clicking**.
3. Click anywhere on the background so that the boundary of the drawing region is highlighted.
4. Right-click to stop selecting. The background is the only object selected in the drawing region. Now you are ready to assign a balloon boundary to the background.

5. Click **Assign>Boundary>Balloon**. The **balloon1** boundary appears in the **Boundary** list, and the following types of balloon boundaries appear at the bottom of the window:

   Charge Models an electrically insulated system. That is, the charge at infinity balances the charge in the problem region, forcing the net charge to be zero.

   Voltage Models an electrically grounded system (voltage at infinity is zero). However, the charge at infinity may not exactly balance the charge in the problem region.

6. Verify that **Charge** is selected, and then click **Assign** to define the balloon boundary. The background is ballooned, and **balloon** replaces **NEW** next to **balloon1** in the **Boundary** list.

Leave Substrate with a Natural Surface

The substrate is to remain with a natural surface. Therefore, its boundary does not need to be explicitly defined.

Exit the Boundary Manager

Once the boundaries and sources have been defined, you can exit the Boundary Manager.

To exit the Boundary Manager:

1. Click **File>Exit**. A window appears, displaying the following prompt:

   `Save changes to “microstrip” before closing?`

2. Click **Yes**.

You are returned to the **Executive Commands** window. A check mark now appears next to **Setup Boundaries/Sources**, and **Setup Solution Options** and **Solve** are now enabled.

6

Generating a Solution

Now that you have created the geometry and set up the problem, you are ready to specify solution parameters and generate a field solution.

Your goals for this chapter are as follows:

- Set up an automatic capacitance calculation.
- Modify the criteria that affect how Maxwell 2D computes the solution.
- Generate the electrostatic solution. The electrostatic solver calculates electric potential at all points in the problem region.
- View information about how the solution converged and what computing resources were used.

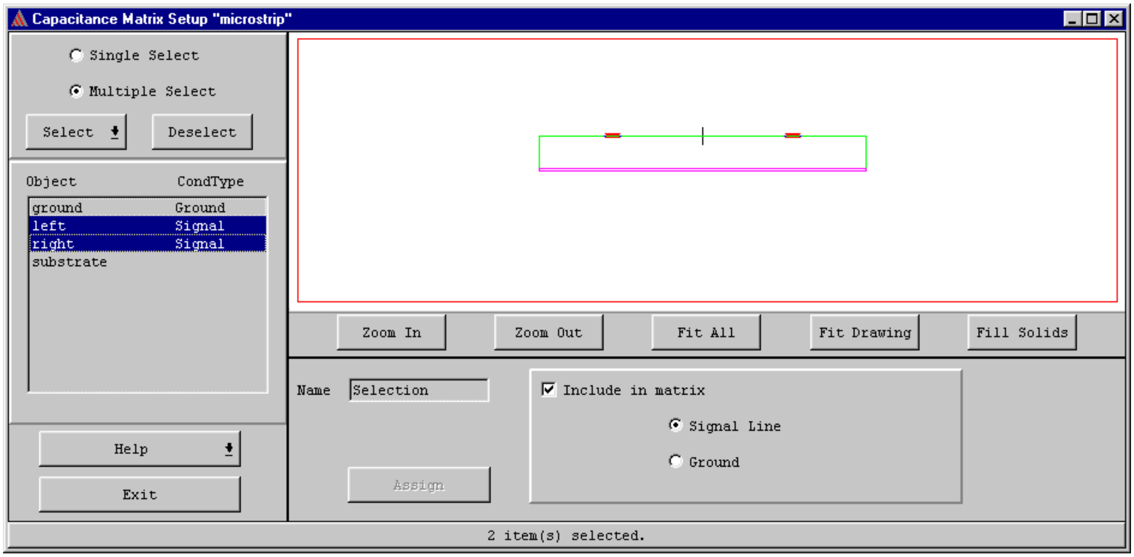
Time	This chapter should take approximately 35 minutes to work through.

Set Up a Matrix Calculation

After solving the problem, you will use a capacitance matrix to calculate the capacitance. The matrix calculation is defined using the **Setup Executive Parameters** command.

To set up the capacitance matrix calculation:

1. Click **Setup Executive Parameters>Matrix** from the **Executive Commands** menu. The **Capacitance Matrix Setup** window appears.



2. Assign the two microstrips as signal lines:
   a. Click **left** and **right** in the **Object** list.
   b. Select the **Include in matrix** check box.
   c. Select **Signal Line**, and click **Assign**.
3. Assign the ground as ground:
   a. Click **ground** in the **Object** list.
   b. Select the **Include in matrix** check box.
   c. Select **Ground**, and click **Assign**.
4. Click **Exit** to close the **Capacitance Matrix Setup** window. A message appears, asking if you want to save your changes.
5. Click **Yes**.

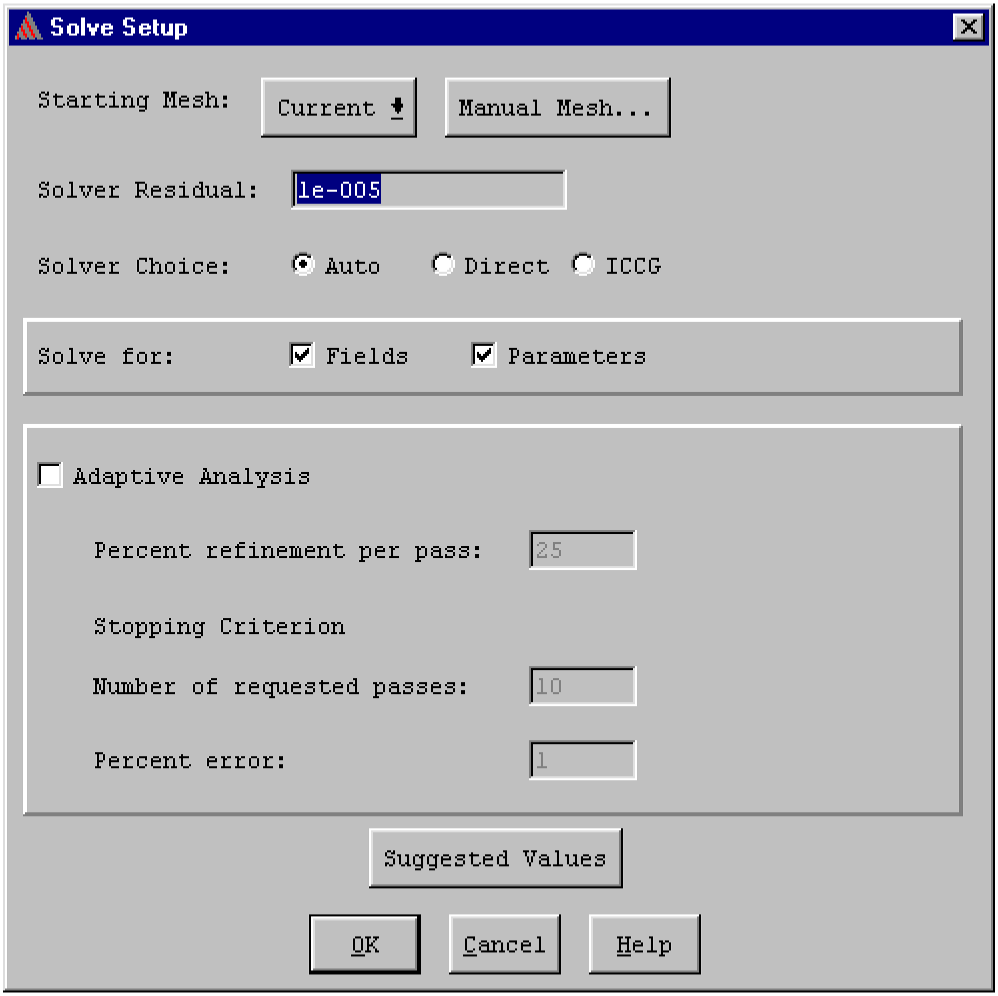
You return to the **Executive Commands** window. A check mark now appears next to the **Setup Executive Parameters** and **Setup Executive Parameters>Matrix** commands.

Access the Setup Solution Menu

Maxwell 2D automatically assigns a set of default solution criteria after you assign boundaries and sources. As a result, a check mark automatically appears next to the **Setup Solution Options** button on the **Executive Commands** menu after you use the **Setup Boundaries/Sources** command.

You can generate a solution using the default criteria. In this problem, however, you will change two of the criteria to make the solution converge more quickly.

To access and set up the solution options, click **Setup Solution Options**. The **Solve Setup** window appears.



Modify Solution Criteria

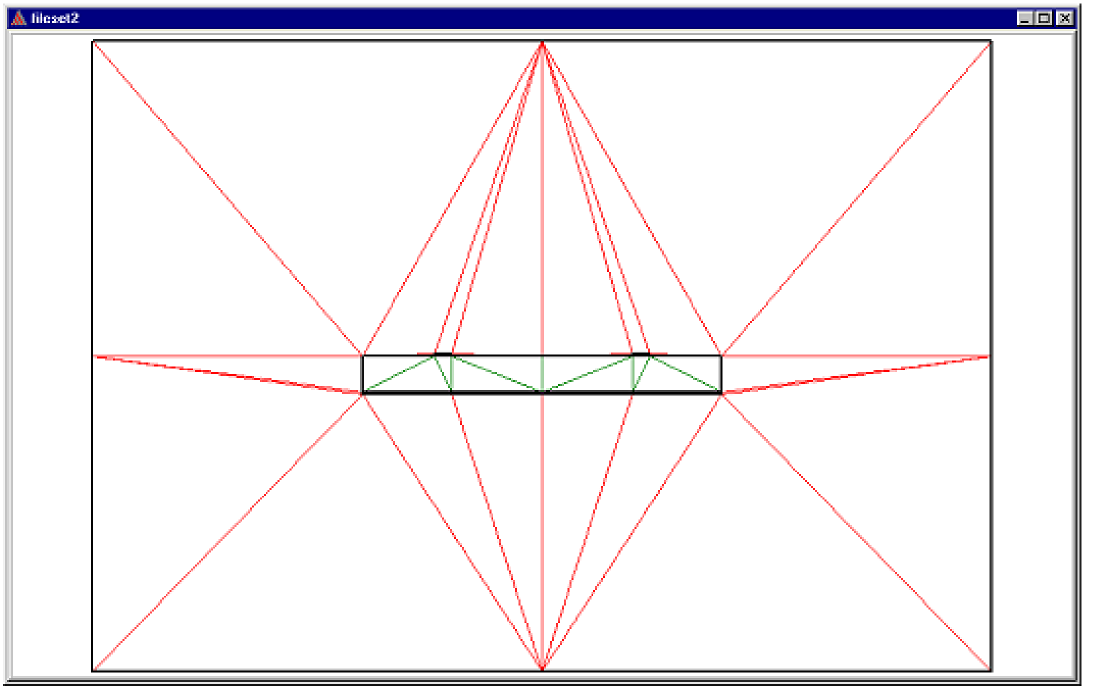
When the simulator generates a solution, it explicitly calculates the potential values at each node in the finite element mesh and interpolates the values at all other points in the problem region.

Specify the Starting Mesh

For this problem, you will use the coarse mesh that is first generated when you begin the solution process. This is referred to as the initial mesh.

Leave the **Starting Mesh** option set to **Initial**.

The coarse initial mesh for the microstrip problem is shown below:



Note If you had already generated a solution and wanted to take advantage of the most recently refined mesh, you would click **Current**.

Specify the Solver Residual

The solver residual specifies how close each solution must come to satisfying the equations that are used to generate the solution. For this model, the default setting is sufficient.

Leave the **Solver Residual** field set to the default.

Note Some solution criteria are given in scientific notation shorthand. For instance, **1e-05** is equal to $1\text{x}10^{-5}$, or 0.00001. When entering numeric values, you can use either notation.

Specify the Solver Choice

You can specify which type of matrix solver to use to solve the problem. In the default **Auto** position, the software makes the choice. The **ICCG** solver is faster for large matrices, but occasionally fails to converge (usually on magnetic problems with high permeabilities and small air-gaps). The **Direct** solver will always converge, but is much slower for large matrices. In the **Auto** position, the software evaluates the matrix before attempting to solve; if it appears to be ill-conditioned, the **Direct** solver is used, otherwise the **ICCG** solver is used. If the **ICCG** solver fails to converge while the solver choice is in the **Auto** position, the software will fall back to the **Direct** solver automatically.

Leave the **Solver Choice** option set to **Auto**.

Specify the "Solve for" Options

The **Solve for** options tell the system what types of solutions to generate.

Fields	A field solution is generated.
Parameters	Any special quantities that you set up using the **Setup Executive Parameters** command are computed.

Leave both the **Fields** and **Parameters** check boxes selected, to solve for both fields and parameters in this example.

Specify the Adaptive Analysis Settings

Set the adaptive refinement settings.

To adaptively refine the mesh and solution:

- Leave **Adaptive Analysis** selected.
  This allows the simulator to solve the problem iteratively, refining the regions of the mesh in which the largest error exists. Refining the mesh makes it more dense in the areas of highest error, resulting in a more accurate field solution.
- Change **Percent refinement per pass** to **25**.
  This causes 25 percent of the mesh with the highest error energy to be refined during each adaptive solution (that is, each solve-refine cycle).
- Leave **Number of requested passes** and **Percent error** set to their defaults of **10** and **1**.
  After each iteration, the simulator calculates the total energy of the system and the percent of this energy that is caused by solution error. It then checks to see if the number of requested passes has been completed, or if the percent error *and* the change in percent error between the last two passes match the requested values. If either of the criteria have been met, the solution process is complete and no more iterations are done.

Exit Setup Solution

When the solution criteria have been defined, you can exit the **Solve Setup** window. The solution criteria are saved automatically upon exiting.

To save your changes and exit the **Solve Setup** window, click **OK**. You return to the **Executive Commands** menu.

Generate the Solution

Now that you have set up the solution parameters, the problem is ready to be solved. In the sample, the default number of 10 passes is more than enough to ensure the solution converges.

To execute the solution, click **Solve**. The solution process begins, and the following actions occur:

- The system creates the initial finite element mesh for the microstrip structure. A bar labeled **Making Initial Mesh** appears in the **Solution Monitoring** box at the bottom of the screen. It shows the system's progress as it generates the mesh.
- A button labeled **Abort** appears next to the progress bar and remains there throughout the entire solution process. You can click it to stop the solution process.
- A bar labeled **Setting up solution files** appears.

After the system makes the initial mesh, the electrostatic field solution process begins.

Monitoring the Solution

The following two monitoring bars alternate in the **Solution Monitoring** area at the bottom.

Solving Fields	Displayed as the simulator computes the field solution. After computing a solution, identifies the triangles with the highest energy error.
Refining Mesh	Displayed as the simulator refines the regions of the finite element mesh with the highest error energy. Since you specified 25% as the portion mesh to refine, the simulator refines triangles with the top 25% error.

Periodically, the solver also displays messages beneath these progress bars.

To monitor the solution after a few adaptive passes are completed, click the **Convergence** button.

Solution Criteria

Information about the solution criteria is displayed on the left side of the convergence display, as shown in the previous figure.

Number of passes	Displays how many adaptive passes have been completed and still remain.
Target Error	Displays the percent error value that was entered using the **Setup Solution Options** command — in this case, one percent.
Energy Error	Displays the percent error from the last completed solution — in this case, **1.61** percent. Allows you to see at a glance whether the solution is close to the desired error energy.
Delta Energy	Displays the change in the percent error between the last two solutions — in this case, **5.94** percent.

Completed Solutions

Information about each completed solution is displayed on the right side of the screen.

Pass	Displays the number of the completed solutions.
Triangles	Displays the number of triangles in the mesh for a solution.
Total Energy (J)	Displays the total energy of a solution.
Energy Error (%)	Displays the percent error of the completed solutions.

Completing the Solution Process

When the solution is complete, a window with the following message appears:

```
Solution Process is complete.
```

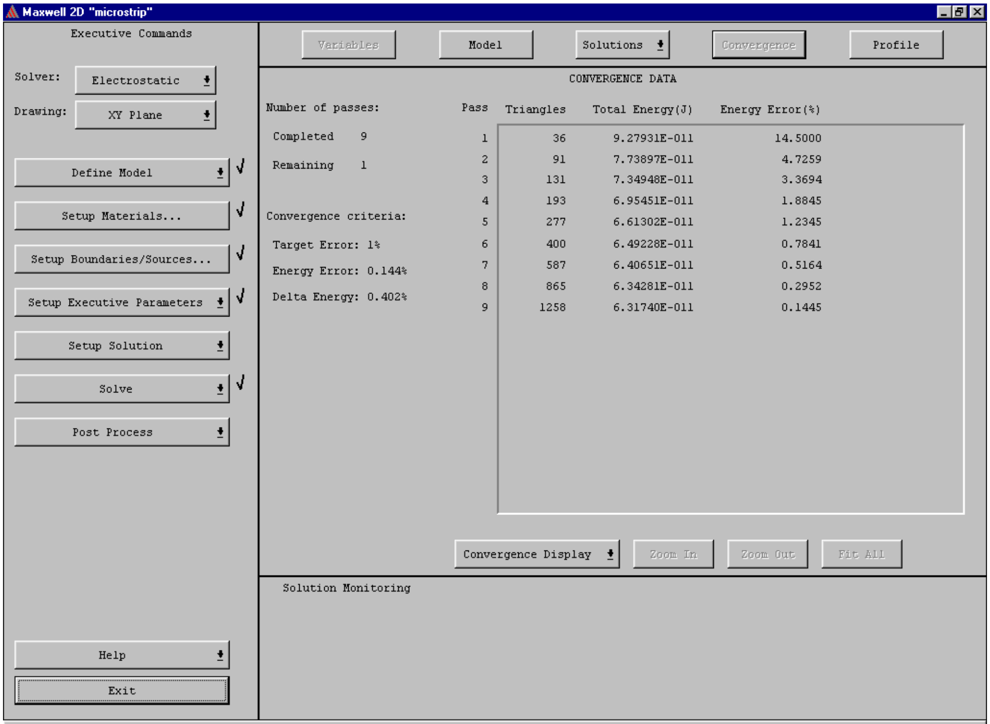
Click **OK** to continue.

You are now ready to view the final convergence for the completed solution.

Note After a solution is generated, the system does not allow you to change the geometry, material properties, or boundary conditions of the model unless you first delete the solution. Therefore, you must generate a new solution if you change the model after generating a solution.

Viewing Final Convergence Data

If you have selected another button from the top of the display area, click the **Convergence** button above the viewing window to view the convergence data for the problem. Convergence data for the completed solution appears as shown below:



Because the **Energy Error** (**0.144**) and the **Delta Energy** (**0.402**) for the seventh pass were both less than the specified stopping criterion of one percent, the solution process stopped before all 10 requested passes were completed. The relatively small changes in the total energy between the last few adaptive passes indicate that the solution has converged to the defined criteria.

Note Generally, the energy error decreases after each adaptive pass as the simulator converges on an accurate solution. If a problem does not begin to converge after several adaptive passes, the problem is probably ill-defined; for instance, boundary conditions may not have been specified correctly.

If this ever happens, do the following:

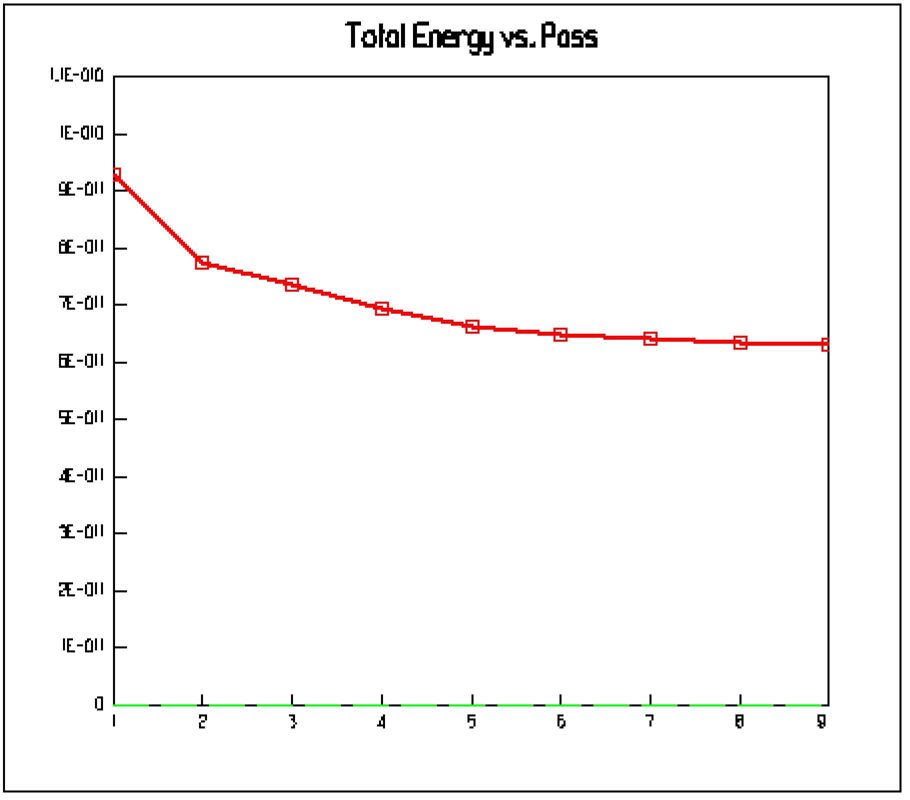
1. Interrupt the simulator by clicking the **Abort** button in the **Solution Monitoring** box.
2. Check the problem definition.
3. Generate a new solution for the problem.

Plotting Convergence Data

By default, convergence data is displayed in table format. This data can also be displayed graphically.

To plot the total energy associated with each pass:

- Click **Convergence Display>Plot Total Energy** from the bottom of the display area. The following plot appears:



Displaying the data graphically often makes it easier to see how the solution is converging. For instance, in the previous plot, the curve begins to level off after the fourth pass as the total energy starts to stabilize.

Optionally, use the other commands under **Convergence Display** to plot the number of triangles or percent error for all adaptive passes.

Viewing Statistics

Click **Profile** at the top of the **Executive Commands** window to see what computing resources were used during the solution process.

PROFILE INFORMATION

Command/Info	Real Time	CPU Time	Mem Size	Num Elements
Pass 7				
mesh_adapt	00:00:00	00:00:00	21592K	655 triangles
Solver RSS	00:00:01	00:00:00	19336K	1438 matrix
Disk I/O				125 K
es2d_solve	00:00:00	00:00:00	18264K	587 triangles
Pass 8				
mesh_adapt	00:00:00	00:00:00	21592K	947 triangles
Solver RSS	00:00:00	00:00:00	19336K	2004 matrix
Disk I/O				177 K
es2d_solve	00:00:01	00:00:00	19288K	865 triangles
Pass 9				
mesh_adapt	00:00:00	00:00:00	21592K	1377 triangles
Solver RSS	00:00:01	00:00:00	21384K	2831 matrix
Disk I/O				249 K
es2d_solve	00:00:00	00:00:01	19288K	1258 triangles
Matrix solution				
Solver RSS	00:00:00	00:00:00	21384K	2831 matrix
Disk I/O				249 K
Solver RSS	00:00:01	00:00:00	21384K	2831 matrix
Disk I/O				249 K
es2d_c_mat	00:00:01	00:00:01	20312K	1258 triangles
Total	00:00:06	00:00:02	21592K	1377 triangles

Finished microstrip on ALT02 in 00:00:59 at 11/26/2002 11:26:36

The time the solution process began is displayed at the top of the window. Beneath it, the following information is displayed for each adaptive field solution and mesh refinement step completed:

Command/Info Displays the name of the simulator command that was used.

Real Time Displays the time taken to complete the step.

CPU Time Displays the amount of time taken by the PC's CPU (central processing unit) to complete the step.

Mem Size Displays the amount of memory used.

Num Elements Displays the of number of triangles in the finite element mesh.

Note If more data is available than can fit on a single screen, scroll bars appear.

7

Analyzing the Solution

Now that you have generated an electrostatic solution for the microstrip problem, you can analyze it using the post processing features of Maxwell 2D.

Your goals for this chapter are as follows:

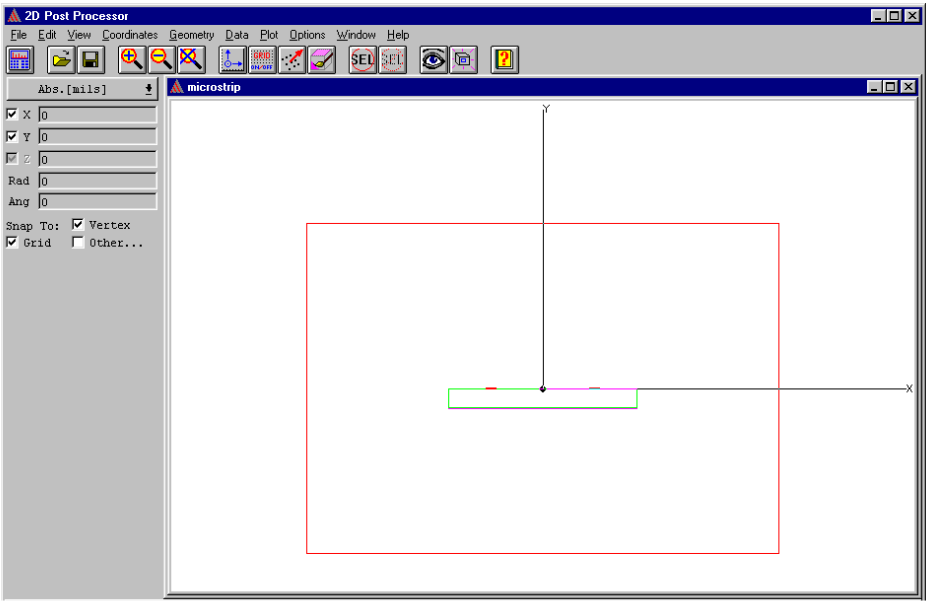
- Plot the voltage.
- Print a hardcopy of the plot.
- Calculate the capacitance between the two microstrips in the structure.

Time This chapter should take approximately 35 minutes to work through.

Access the Post Processor

Use the Post Processor to plot quantities and access the calculator menus.

To access the Post Processor, click **Post Process**. The 2D Post Processor appears.



Post Processor Screen Layout

The following sections provide a brief overview of the Post Processor in Maxwell 2D. For more details, refer to the Maxwell 2D online help.

With version 9, the Post Processor is now common to both the Maxwell 2D and the Maxwell 3D solvers. Most visualization techniques available in 3D can also be used in 2D.

General Areas

The Post Processor is divided into the following general areas:

Menu Bar Appears at the top of the **2D Post Processor** window. Generally, it is set up the same way as the 2D Modeler's menu bar.

Toolbar Appears at the top of the **2D Post Processor** window as a row of icons. Includes icons with easy access to frequently used commands.

Viewing Window Contains the geometry, allowing you to view the model and plots. Displays **2D Post Processor** and project name in title bar.

Side Window Displays the vertex and snap behavior, and shows the coordinates, type of coordinate system, and units of length.

Status Area Displays the version number of the Maxwell 2D Post Processor, and any status help on the menu command.

Menu Bar

The post processor's menu bar appears at the top of the window. Generally, it is set up the same way as the 2D Modeler's menu bar.

Status Bar

The status bar, which appears on the right side of the post processor screen, displays the following:

Mouse Mode Displays the "snap-to-point" behavior of the mouse when points are picked from the screen. For instance, when the following is displayed:

```
Mouse Mode
Object          Yes
Grid            Yes
Keyboard        No
```

the simulator will snap to the closest grid point or object vertex. See the description of the command **Global/Defaults** in *Maxwell 2D Field Simulator User's Reference* for more details.

Maximums and Minimums Display the maximum and minimum values of x- and y-coordinates associated with objects that are being displayed. Essentially, these coordinates define the diagonal corners of the problem region.

Mouse position Displays the coordinates of the cursor.

Units Displays the current unit of length.

Mouse functions Display the effect of clicking the left and right mouse buttons. For instance, when the simulator is waiting for you to choose a command, the following is displayed:

```
Mouse Left      MENU PICK
Mouse Right
```

Generally, the left button allows you to perform some function while the right button allows you to abort the command.

Scientific Notation

The simulator displays most values in scientific notation. For example, the value 234 is displayed as:

```
+2.3400e+02
```

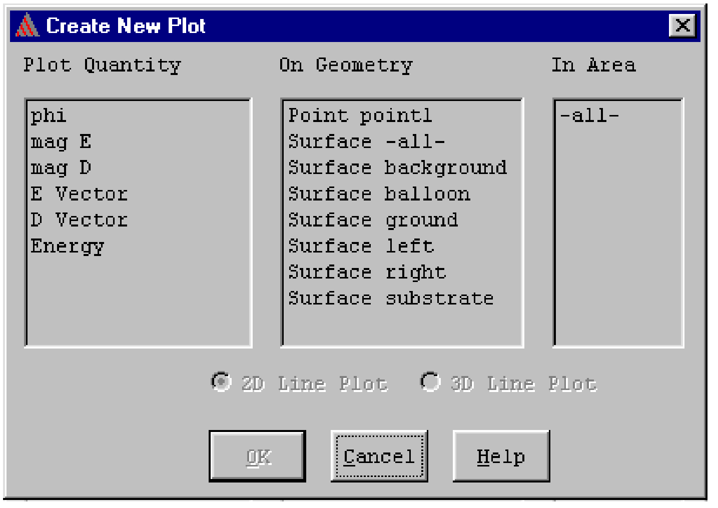
which is equivalent to 2.34×10^{2}. When entering values, you can use scientific notation or regular notation. For example, you can type the value .00021 as **.00021** or **2.1e-4**.

Plot the Electric Field

Now you are ready to plot the electric field. Use the **Plot>Field** command as follows to plot the field throughout the problem region.

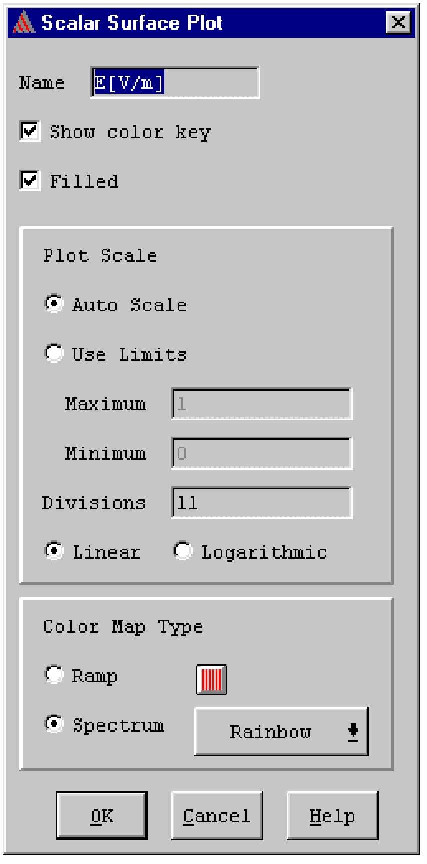
To plot the voltage:

1. Click **Plot>Field**. The **Create New Plot** window appears.



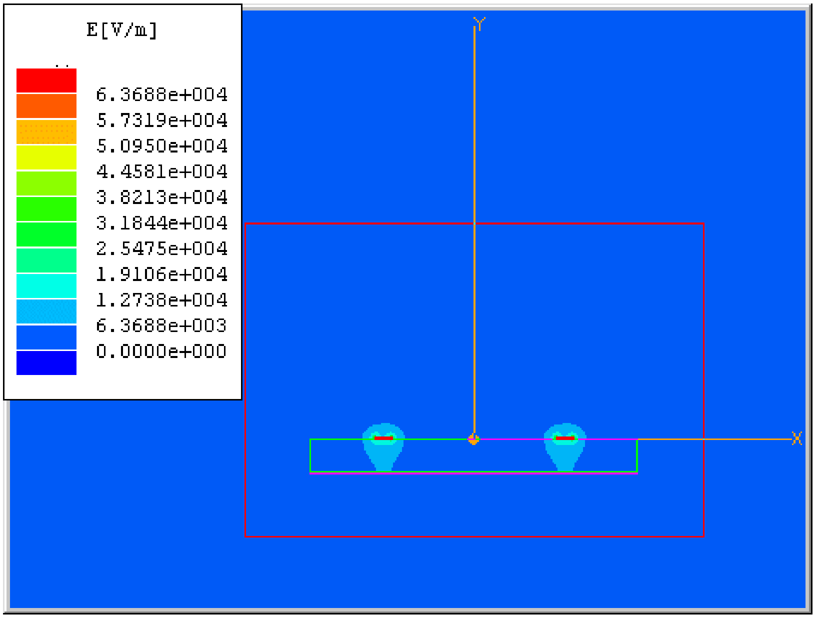
2. Click **mag E** in the **Plot Quantity** list. This is magnitude of the electric field.
3. Click **Surface -all-** in the **On Geometry** list. This instructs the software to plot the field over the surfaces of all the objects in the model.
4. Click **-all-** in the **In Area** list. This option plots the field over the entire model. You can define other areas using the **Geometry>Create** commands. Once an area is defined, it appears in this list.

5. Click **OK**. The **Scalar Surface Plot** window appears.



6. Verify that the **Show color key** and **Filled** check boxes are selected. This allows you to observe both the color key and the shading of the created plot.

7. Accept the remaining default settings, and then click **OK**. The field plot appears.



The plotted field range appears in the plot key at the upper-left corner of the screen. By default, this range has ten divisions. Maxwell 2D set this range up automatically based on the E-field solution.

Note You can change the number of divisions and the range for this contour plot by double-clicking on the plot key. For more details, refer to the Maxwell 2D online help.

Formatting and Manipulating Plots

Use the Post Processor **Plot** menu to edit plot attributes.

To make a plot easier to read, you can move the legend by clicking the right mouse button on it, then dragging it into position. You can also adjust the axes to make the plot more readable.You may also need to move plots, hide plots, or manipulate other plot characteristics.

Rotate a Plot

You may want to rotate a plot to view different angles.

To rotate a plot, use the hot keys in the following manner:

1. Press and hold down **Ctrl**, and then click.
2. Move the pointer to the right to rotate the plot.

Note After you have rotated the plot, to return to XY view, use the following hot key command:

```
Ctrl+Double-click in upper center of view
```

When you do this, you return to the original position, rotated by 90 degrees.

Hide or Show a Plot

You may want to hide plots to make the remaining plots more readable.

To set a plot's visibility:

1. Click **Plot>Visibility**. The **Plot Visibility** window appears.
2. In the list, click on the plot you want to hide. The **Visible** column changes to **No**, and the plot disappears.
3. Click on the plot again to show it. The **Visible** column changes back to **Yes**, and the plot reappears.
4. Click **OK** to close the **Plot Visibility** window.

Open Multiple Plots

You can create or open multiple plots and display them individually or tile your windows so that you see all of the plots at once.

To open another plot:

1. Click **Plot>Open>2D Plot**. The **Load 2D plot file** window appears.
2. Find and select the plot you want to open.
3. Click **OK**.

To view multiple plots at once, do one of the following:

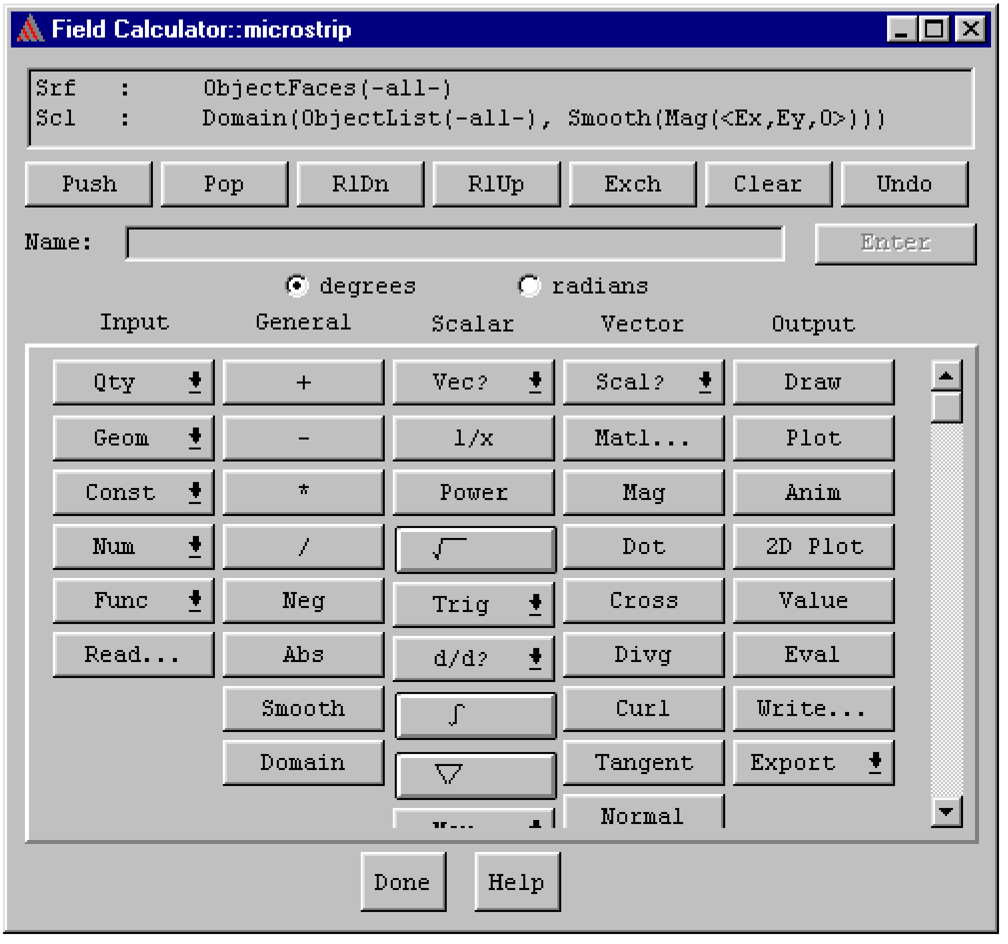
- Click **Window>Tile**. Plot windows are arranged so that all are displayed on the screen at the same time.
- Click **Window>Cascade**. All plot windows are stacked.

The 2D Calculator

To calculate capacitance, use the 2D field calculator. The field calculator allows you to manipulate the field quantities. All functions of the 2D field calculator are described in the Maxwell 2D online help.

To access the 2D Field Calculator:

1. Click **Data>Calculator**. The Field Calculator appears.



The calculator is divided into two parts: the top portion displays the contents of the register stack, and the bottom portion displays the functions of the calculator. The calculator already contains the results of the previous field plot, which is the magnitude of the E-field plotted on all the object faces.

2. Click **Done** to close the Field Calculator.

Examine Results of Capacitance Computation

Before using the capacitance matrix results to calculate the capacitance in the Field Calculator, first exit the Post Processor, and then examine the results of the capacitance matrix calculation.

To exit the Post Processor and examine the resultant capacitance matrix:

1. Click **File>Exit** to exit the Post Processor.
2. In the **Executive Commands** window, click **Solutions>Matrix**. The capacitance in the C11 entry should be approximately 6.2 e-11 or 6.3e-11 F/m.

The electrostatic solver calculates the capacitance matrix by solving the final mesh again, this time applying a 1 volt excitation to the test conductor and grounding all other signal conductors included in the matrix. In this example, the final mesh is solved twice after the final pass: once for the left conductor, and once for the right conductor.

The energy for the two 1 volt solutions is used to compute the capacitance for the test electrode, using the following equation:

$$U = \frac{1}{2}CV^2$$

Since the voltage was set to 1 volt, the capacitance is equal to 2 x the energy:

$$C = \frac{2U}{V^2}$$

In the next section, you will verify this calculation.

Verify Capacitance Calculation

You can verify this calculation by returning to the Boundary/Source Manager, resetting the left conductor to 1 volt, and the right conductor to ground.

Reset the Conductor Values

Reset the left microstrip to 1 volt and the right microstrip to 0 volts.

To reset the conductor values:

1. From the **Executive Commands** menu, click **Setup Boundaries/Sources**, and click **Modify**. The Boundary/Source Manager appears.
2. Keep the left microstrip **Value** set to 1 V.
3. Select the right microstrip, reset the **Value** to 0 V, and then click **Assign**.
4. Click **File>Exit** to exit the Boundary/Source Manager, and save changes when prompted.

Remove Mesh Refinement

Set the project up so that it will not refine the mesh this time. Doing so decreases the number of passes needed to solve the project.

To remove mesh refinement:

1. Click **Setup Solution Options**. The **Solve Setup** window appears.
2. Clear the **Adaptive Analysis** check box.
3. Click **OK**.

Run the Solution Again

To run the solution again:

1. Click **Solve>Nominal Problem**.
2. When the solution is complete, click **Solutions>Matrix** to view the capacitance calculation again. It should still be between 6.2 e-11 and 6.3e-11 F/m.

Calculate Capacitance

The energy for the two 1 volt solutions is used to compute the capacitance for the test electrode, using the following equations:

$$U = \frac{1}{2}\int_{vol} E \bullet D dv \quad \text{and} \quad U = \frac{1}{2}CV^2$$

Since the voltage was set to 1 volt, the capacitance is equal to 2 x the energy:

$$C = \frac{2U}{V^2}$$

Note Regardless of the drawing units used in the Maxwell 2D Modeler, the field calculator expresses all output quantities in MKS (SI) units.

Enter the Post Processor Field Calculator Again

Compute the capacitance manually from the energy, and compare this result to the total energy in the convergence table.

To open the Post Processor and the Field Calculator:

1. Click **Post Process**. The **2D Post Processor** window appears.
2. Click **Data>Calculator**. The 2D Field Calculator appears.

Compute the Energy

The first step in computing capacitance is to load the E-field and the D-field into the register stack.

To load the E-field and the D-field:

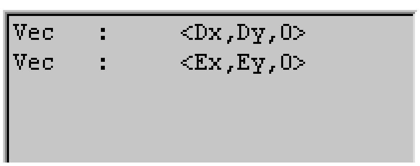
1. If any entries remain on the calculator stack, click **Clear** to remove them. A message appears, asking you to confirm your command. Click **Yes**. The calculator stack is cleared of any existing values.
2. Click **Qty/E** from the **Input** column to load the electric field vector **E** into the top register of the calculator first. After **E** is loaded, the top register appears as follows:

```
Vec: <Ex,Ey>
```

3. Click **Qty/D** to load the electric flux density vector **D** into the top register of the calculator. The top register appears as follows:

```
Vec: <Dx,Dy>
```

The register stack now contains the following:



Calculate the Dot Product

Calculate the dot product of the two vector fields.

To calculate the dot product:

1. Click **Dot** from the **Vector** column of calculator commands. After the dot product has been calculated, the top register of the calculator appears as follows:

   ```
   Scl: Dot(<Ex,Ey>, <Dx,Dy>)
   ```

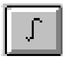
2. Click **Geom>Surface** from the **Input** column. The **Select Surface** window appears.
3. Select **all** from the list, and click **OK**.
   The register stack now contains the following:

   ```
   Srf   :     ObjectFaces(-all-)
   Scl   :     Dot(<Ex,Ey,0>, <Dx,Dy,0>)
   ```

Integrate the Values

Integrate the value in the top register of the calculator, and save it in the top register of the calculator stack.

To integrate the values:

1. Click the integrate button  from the **Scalar** column.
   The register stack now contains the following:

   ```
   Scl   :     Integrate(ObjectFaces(-all-), Dot(<Ex,Ey,0>, <Dx,Dy,0>)
   ```

2. Click **Eval** from the **Output** column.
   The register stack now contains the following:

   ```
   Scl   :     6.27760908599585E-011
   Scl   :     Integrate(ObjectFaces(-all-), Dot(<Ex,Ey,0>, <Dx,Dy,0>)
   ```

   The value in the top register is integrated and summed for all surfaces. The result of that integration — 6.277609e^{-11} — is displayed in the top register of the stack. That sum represents:

   $$\int_{vol} E \bullet D dv$$

3. Enter a constant value of 2 as follows:
   a. Click **Num>Scalar** from the **Input** column. The **Scalar Constant** window appears.

b. Enter **2** in the **Scalar Value** field.
c. Click **OK**.
The register stack now contains the following:

```
Scl   :     2
Scl   :     6.27760908599585E-011
Scl   :     Integrate(ObjectFaces(-all-), Dot(<Ex,Ey,0>, <Dx,Dy,0>)
```

4. Click / from the **General** column to divide the top two quantities in the calculator stack. The register now contains the total energy of the system, which is $3.1388e^{-11}$ joules per meter. The register stack now contains the following:

```
Scl   :     3.13880454299792E-011
Scl   :     Integrate(ObjectFaces(-all-), Dot(<Ex,Ey,0>, <Dx,Dy,0>)
```

Compute the Capacitance Value

You are now ready to compute the value of the mutual capacitance between the electrodes. The energy was calculated in the previous section to be approximately $3.1388e^{-11}$. This value should still appear in the top register of the calculator stack.

To obtain the capacitance, you need to multiply the energy by 2.

To compute the capacitance:

1. Click **Num>Scalar** from the **Input** column. The **Scalar Constant** window appears.
2. Enter **2** in the **Scalar Value** field.
3. Click **OK**.

```
Scl   :     2
Scl   :     3.13880454299792E-011
Scl   :     Integrate(ObjectFaces(-all-), Dot(<Ex,Ey,0>, <Dx,Dy,0>)
```

4. Click ***** from the **General** column to multiply the first two quantities in the calculator stack. The register now contains the capacitance, which is approximately $6.2776e^{-11}$ farads per meter.

```
Scl   :     6.27760908599585E-011
Scl   :     Integrate(ObjectFaces(-all-), Dot(<Ex,Ey,0>, <Dx,Dy,0>)
```

Compare these total energy and capacitance values (3.1388e^{-11} and 6.2776e^{-11}) to the total energy and capacitance listed in the convergence table in the **Executive Commands** window (which are 3.1365e^{-11} and 6.2732e^{-11}).

Note To view the total energy, click the **Convergence** button in the **Executive Commands** window. To view the capacitance matrix, click **Solutions>Matrix**.

Exit the Calculator

Once the calculations are complete, you can exit the calculator and return to the Post Processor.

To exit the calculator:

- Click **Done** from the bottom of the calculator.

You return to the 2D Post Processor.

Exit the Post Processor

To exit Maxwell 2D:

1. Click **File>Exit** from the post processor. A window with the following prompt appears:

    `Exit Post Processor?`

2. Click **Yes**. The **Executive Commands** window appears.

Exit Maxwell 2D

Once you have finished the post processing functions, you are ready to exit the software.

To exit Maxwell 2D:

1. Click **Exit** from the bottom of the **Executive Commands** menu. The following prompt appears:

    `Exit Maxwell 2D?`

2. Click **Yes**. The **Executive Commands** window closes, and the Project Manager reappears.

Exit the Maxwell Software

To exit the Project Manager, click **Exit**. The Project Manager closes, and the Control Panel reappears.

To exit the Maxwell Control Panel:

1. Click **EXIT**. A window with the following prompt appears:

    `Exit Maxwell?`

2. Click **Yes**. You return to Microsoft Windows.

You have completed *Getting Started: A 2D Electrostatic Problem*, the first Getting Started guide for Maxwell 2D SV. You can now continue with the second guide in the SV Getting Started series, *Getting Started: A 2D Magnetostatic Problem.*

Index

Numerics

2D Modeler
  accessing 4-2
  exiting 4-13
2D Post Processor 7-2

A

adaptive analysis 6-5
adaptive refinement 6-5
aliases 2-3
Ansoft software, installing 1-iv

B

background
  and boundary conditions 5-5
  ballooning 5-9
  default material for 5-4
  excluding from solution region 5-4
  extending 5-9
  including in solution region 5-4
boundary conditions 5-5
  displaying 5-6
  for sample problem 5-7
  screen layout 5-6
  substrate 5-10
  use of 5-5
Boundary Manager 5-6
Boundary Manager, exiting 5-10
buttons, disabled 2-5

C

calculators
  display of functions 7-8
  display of registers 7-8
  menu 7-8
  overview 7-8
  quitting 7-13
capacitance
  calculating with solution calculators 7-8
  equations for calculating 7-10
colors, changing for objects 4-7, 4-9, 4-12
commands
  accessing from menus 4-3
  accessing with the keyboard 4-3
  accessing with the mouse 4-3
computing resources
  and background 5-4
  viewing 6-10
Control Panel
  icon 2-2

convergence data
    plotting 6-9
    viewing 6-6, 6-8, 6-9
coordinate system, uv (local) 4-3
coordinates
    entering via keyboard 4-4
    maximums and minimums 7-3
    of cursor 4-4
CPU time 6-10

D

Define Model 1-3, 4-2
D-field 7-10
direct solver 6-5
directories
    project 4-1
    selecting project 2-4
dot product calculations 7-11
Draw Model 1-3
Drawing 1-3
drawing plane 4-2
drawing region
    adjusting view of 4-8, 4-11, 4-13
    enlarging objects 4-11
    setting up 4-5

E

Edit>Duplicate>Along Line command 4-9
Edit>Select commands 4-9
E-field 7-10
electric fields 5-7
electric scalar potential
    and electric fields 5-7
Electrostatic 4-2
electrostatic solver 4-2
energy
    error 6-5, 6-6, 6-7
    plotting total energy 6-9
    total 6-7
error
    energy 6-6, 6-7
    of solution 6-5
Executive Commands menu
    Define Model 4-2
    Post Process 7-2, 7-10
    Setup Boundaries/Sources 5-5
    Setup Materials 5-2
    Setup Solution 6-1
    Solve 6-6
    Solver 4-2
executive commands menu 3-3
executive commands window 3-3

F

farads 7-12
field of view
    fitting all objects in 4-13
    zooming in 4-8, 4-11
field quantities
    dot products of 7-11
    loading into registers 7-10
    manipulating with solution calculators 7-8
File commands
    Open 4-3
File commands
    Exit 4-13
    Save 4-10
files, saving modeler files 4-10
Fit All 4-13
Fit Drawing 4-11

G

geometric models
    general strategy for creating 4-2
    saving 4-10
geometry, sample problem model 1-2
grid
    in subwindows 4-3
    snapping to 4-4
grid points, snapping to 4-4
grids, changing point spacing 4-6
ground plane
    applying voltage to 5-9
    assigning a material to 5-4

`Group Objects` 1-3, 4-13

I

ICCG solver 6-5
inductance
  plotting 7-7
installing Ansoft software 1-iv
integrating, over a plane 7-11

K

keyboard
  accessing commands with 4-3
  selecting points with 4-4, 4-6

M

material database
  adding materials to 5-2
Material Manager, accessing 5-2
materials
  adding to database 5-2
  assigning to microstrips 5-4
  assigning to objects 5-2
  assigning to the background 5-4
  assigning to the ground plane 5-4
  assigning to the substrate 5-3
Maxwell 2D Field Simulator
  executive commands 1-3
  exiting 7-13
  general procedure 1-3
  model window 3-3
  starting 3-2
Maxwell 2D Modeler 4-2
Maxwell Control Panel
  accessing 2-2
  exiting 7-13
Maxwell software, installing 1-iv
memory size 6-10
menu bar
  for 2D Modeler 4-3
  for post processor 7-3
mesh
  and solution accuracy 6-4
  default 6-4
  example of 1-1
  initial 6-6
  number of triangles in 6-7
  starting 6-4
  use of 1-1
mesh refinement
  adaptive 6-5
  and error energy 6-5
  percentage refined 6-5
message bar 6-6
  in 2D modeler 4-4
microstrip model 4-4
microstrips
  applying voltages to 5-8
  assigning a material to 5-4
  creating 4-9
MKS units, farads 7-12
`Model>Drawing Units` command 4-5
modeling
  creating the geometry 4-6
  drawing microstrips 4-8
  using the keyboard 4-6
mouse
  accessing commands with 4-3
  snap-to behavior of 4-4, 7-3
mouse buttons 7-3
  functions of 7-3
mouse buttons functions 4-4
mouse coordinates
  displaying 7-3
mouse coordinates, displaying 4-4

N

names
  changing for objects 4-7, 4-9, 4-12
  defining object names 4-7, 4-9, 4-12

O

Object commands
    Rectangle 4-6
objects
    appearing larger 4-11
    assigning materials to 5-2
    changing the colors of 4-7, 4-9, 4-12
    changing the names of 4-7, 4-9, 4-12
    copying 4-9
    creating 4-6
    rectangles 4-6

P

plots
    convergence 6-9
    from calculator registers 7-8
plotting (signals)
    multiple variables 7-7
points
    entering from the keyboard 4-4, 4-6, 4-9
    selecting with the mouse 4-6
points, grid
    snapping to 7-3
points, vertex
    snapping to 7-3
Post Process 1-3, 7-2, 7-10
post processing 7-2
    exiting 7-13
    menu bar 7-3
post processing, screen layout 7-2
    accessing commands 7-3
    status bar 7-3
post processor, screen layout 7-2
project alias 2-3
project directory
    creating 2-3
    definition of 2-3
Project Manager commands
    Projects 2-3
Project Manager options
    New 2-4
    Open 3-2
    Save Notes 2-5
Projects 2-3
projects
    creating 2-4
    entering descriptions of 2-5
    naming 2-4
    opening 3-2
    project notes 2-5
    selecting and accessing 2-4
    selecting project type 2-4

R

Rectangle 4-6
registers
    loading values into the plane 7-10
residual 6-4

S

saving geometry files 4-10, 4-13
scientific notation 6-5, 7-3
scroll bars 6-10
Setup Boundaries/Sources 1-3, 5-5
Setup Executive Parameters 1-3
Setup Materials 1-3, 5-2
Setup Solution Options 1-3, 6-1
size
    of the drawing region 4-3
snap-to behavior of mouse 4-4, 7-3
solution monitoring area 3-3
solutions
    adaptive analysis of 6-5
    analysis 7-1
    and adaptive mesh refinement 6-5
    calculated values 6-4
    changing boundary conditions 6-8
    changing materials 6-8
    changing sources 6-8
    changing the geometry 6-8
    completed solutions 6-7
    computing 6-6
    convergence 6-6, 6-8

CPU time 6-10
criteria for computing 6-4
effect of mesh 6-4
generating 6-6
interpolated values 6-4
interrupting 6-6, 6-8
memory used 6-10
monitoring 6-6
percent error 6-8
plotting 6-9
real time used 6-10
refinement of 6-5
satisfying equations 6-4
total energy 6-7
viewing convergence data for 6-8
viewing profile data for 6-8
`Solve` 1-3, 6-6
`Solver` 1-3, 4-2
solver choice 6-5
solver residual 6-4
solver type 4-2
solvers
auto 6-5
direct 6-5
ICCG 6-5
sources 5-5
displaying 5-6
for sample problem 5-7
screen layout 5-6
specifying drawing planes 4-2
starting mesh 6-4
status information
cursor position 4-4, 7-3
maximums and minimums 7-3
mouse buttons 7-3
snap-to behavior of mouse 4-4, 7-3
unit of length 4-4, 7-3
substrate
assigning a material to 5-3
creating 4-6
subwindows
and complex geometries 4-3
and project windows 4-3
coordinate system 4-3
drawing grid 4-3
size 4-3

T

2D Modeler
screen layout
accessing commands 4-3
menu bar 4-3
message bar 4-4
project window 4-3
status bar 4-4
subwindows 4-3

U

units
changing 4-5
default 4-3
displaying 4-5
of length 7-3
rescaling geometric models 4-5

V

variables
plotting multiple 7-7
vertex points, snapping to 4-4
voltages
applying to ground plane 5-9
applying to microstrips 5-8

W

`Window>Change View` commands
`Fit Drawing` 4-11
`Zoom In` 4-8, 4-11
windows, project 4-3

Z

`Zoom In` 4-8, 4-11