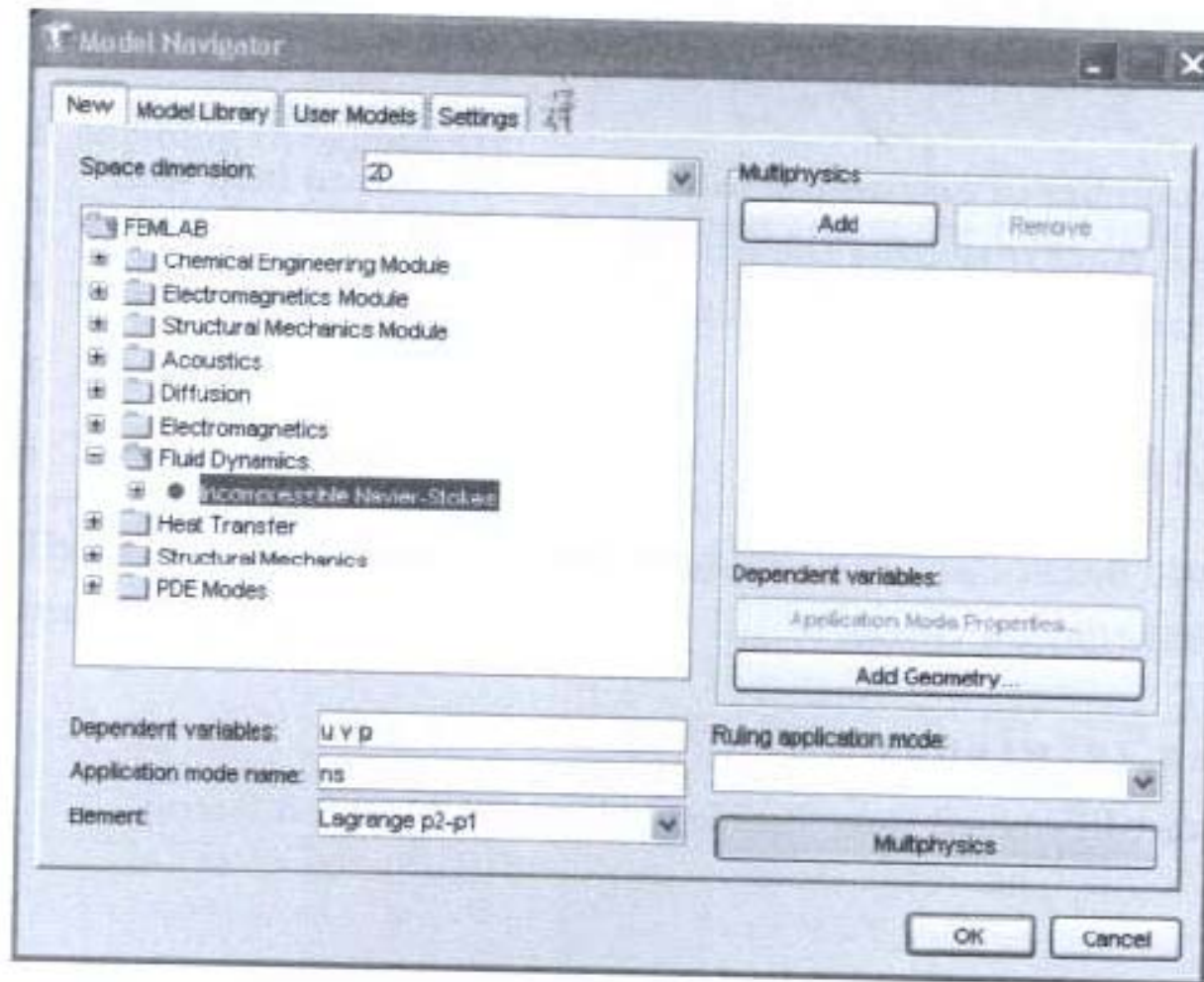


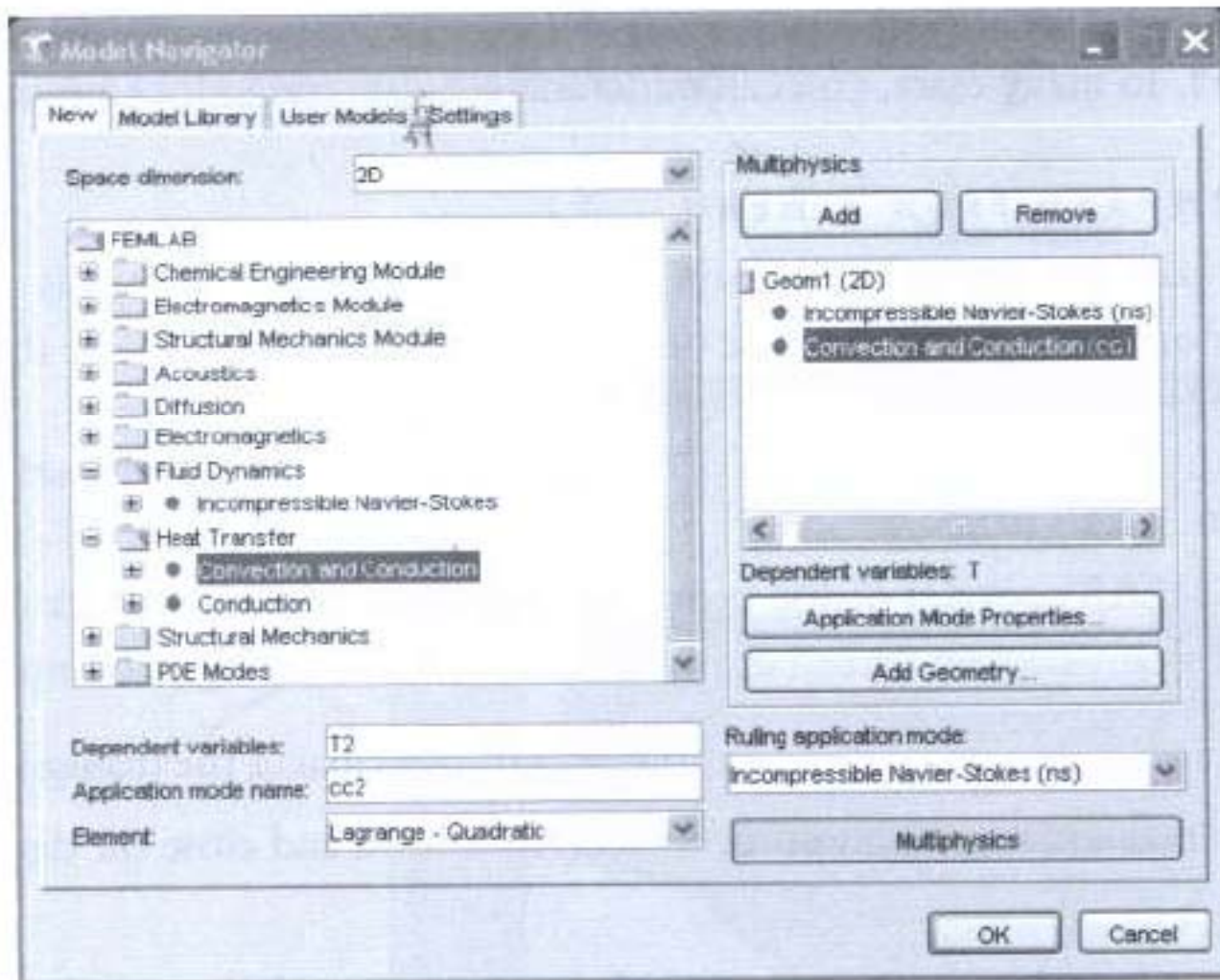
Starting FEMLAB

STARTING A NEW FEMLAB MODEL



- 1 In the **Model Navigator**, select the space dimension from the **Space dimension** list. Choose from 1D, 2D, and 3D geometries using Cartesian coordinates and, for most application modes, 1D and 2D axisymmetric geometries using cylindrical coordinates.
- 2 Select an application mode from the **Application modes** list.
Opening a folder at the top level shows all application modes within that application area. For some application modes you can select from different analysis types or solver types by clicking the plus sign at the application mode node.
- 3 Click **OK**.

CREATING A MULTIPHYSICS MODEL



- 1 Click the **Multiphysics** button.
- 2 Make an appropriate selection from the **Application modes** list.
- 3 Click **Add** to add it to the current model.
- 4 Continue adding application modes by selecting them and then clicking the **Add** button.
- 5 For extended multiphysics modeling using more than one geometry, click the **Add Geometry** button to specify a new geometry. Then proceed to add application modes to the new geometry.
- 6 Click **OK**.

Using commands and dialog box

UNDO AND REDO COMMANDS

To undo and redo many commands, choose **Undo** and **Redo** from the **Edit** menu or press **Ctrl+Z** or **Ctrl+Y**. In many cases, you can undo and redo several modeling steps.

TO CHOOSE A COMMAND FROM A MENU

Choose commands from the menus in the main menu bar. For many commands shortcut keys appear on the menu to the right of the command. Learning to use these shortcut keys can help you work faster.

TO USE A DIALOG BOX

For most inputs, FEMLAB provides dialog boxes that stay open while you work on your model. They typically contain three buttons for using or discarding your input:

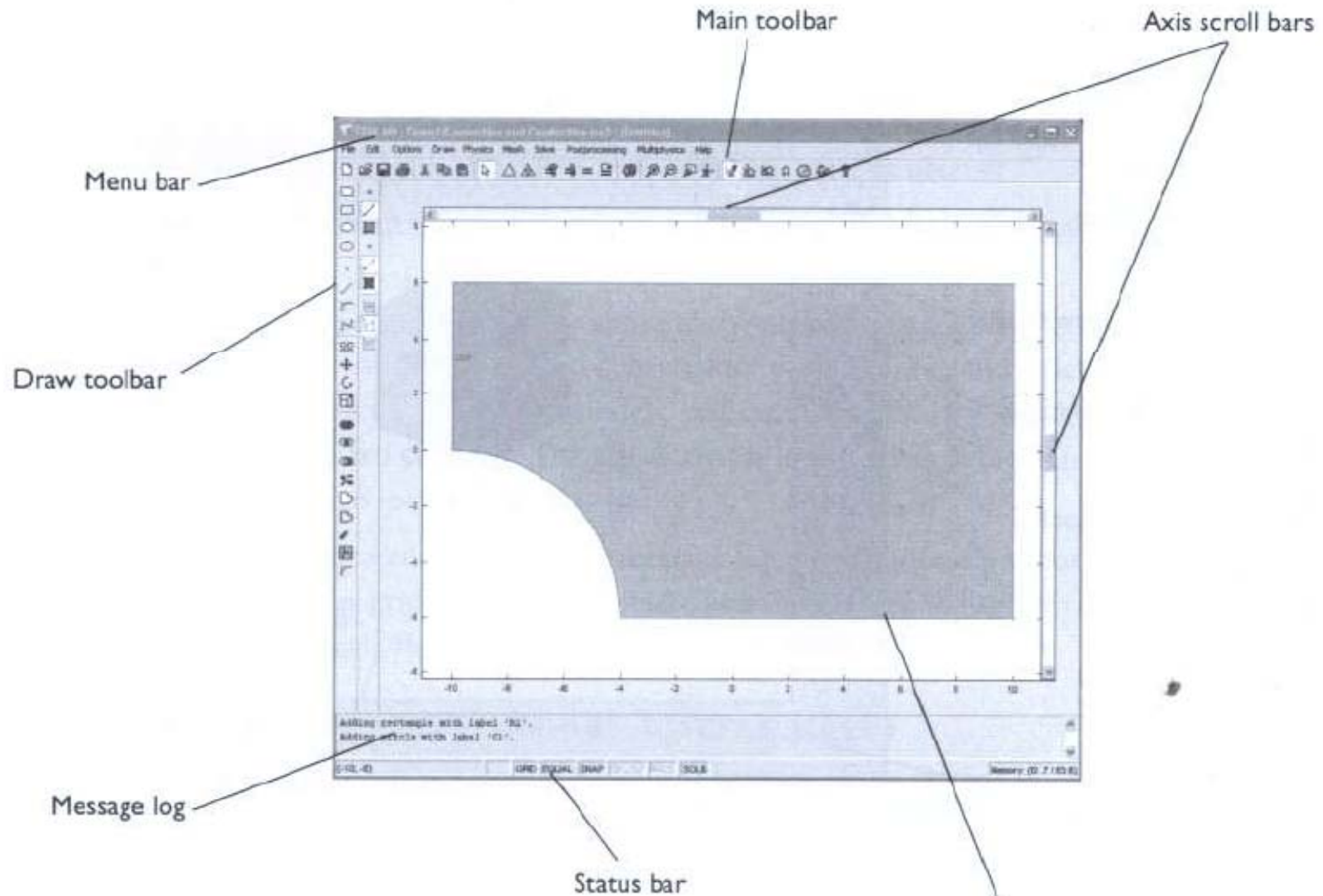
- Click **Apply** to apply all inputs to the current model without closing the dialog box.
- Click **OK** or press **Enter** to apply all inputs to the current model and close the dialog box.
- Click **Cancel** or press **Esc** to discard all inputs and close the dialog box.

Modeling in FEMLAB

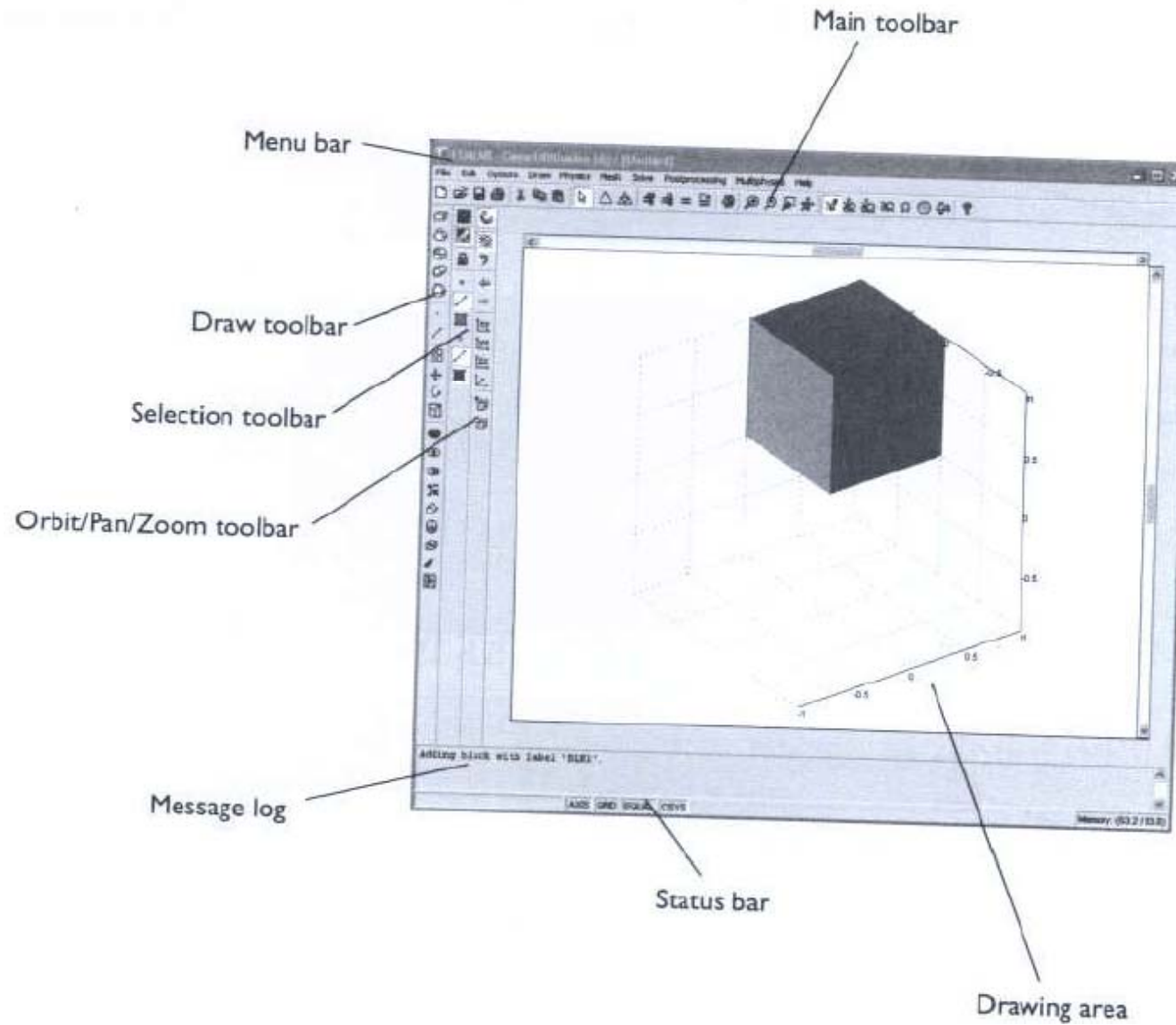
The FEMLAB application window provides a graphical user interface (GUI) that handles all aspects of the modeling process:

- Preprocessing and CAD
- Specification of the physics through equations, material data, boundary conditions, couplings, and other properties
- Meshing, assembly, and solution of the finite element model
- Postprocessing and visualization of the solution and other quantities

2D graphical interface



3D interface



- The Selection toolbar is available for preprocessing 3D models. It provides tools for rendering and selection of domains.
- The Orbit/Pan/Zoom toolbar is available for 3D models and during the postprocessing of 2D models. It contains tools for changing the view and adding light sources.
- The main axes area displays the model during various modeling stages. You can point-and-click to select parts of the model for specification of input data. You can also click-and-drag to draw and edit 1D and 2D geometry objects.
- The color scales map colors in the plots to the numerical values of the plotted properties.

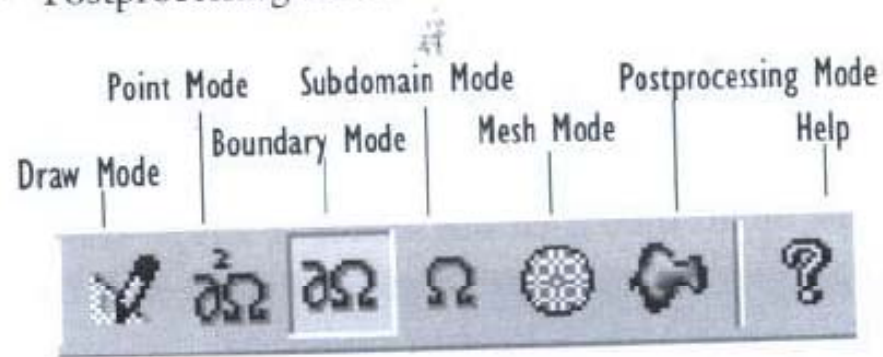
- The message log displays messages about changed model parameters, geometry modeling commands, progress and convergence of the solvers, and much more. When postprocessing 1D and 2D models, you can point-and-click to get numerical values of the plotted property. The values appear in the message log. Simply scroll the message log to look at old messages.
- The status bar, located just below the message log, provides details about the status of settings such as grid lines and axes. Here you can also see the current cursor coordinates. Toggle the state status bar items by double-clicking them.

NAVIGATING THE SELECTION AND DISPLAY MODES

FEMLAB contains various selection and display modes. They provide selection and display of the relevant part of the model. The following modes are available:

- Draw mode
- Point selection mode
- Edge selection mode (3D only)
- Boundary selection mode
- Subdomain selection mode

- Mesh mode
- Postprocessing mode



The mode navigation buttons.

Select the modes using the mode navigation buttons on the main toolbar or from a menu. To access the selection modes for points, edges, boundaries, and subdomains, point to **Selection Mode** in the **Physics** menu and then click the desired selection mode. Select draw mode, mesh mode, and postprocessing mode from the corresponding menus. The mode navigation buttons provide quick mode selection and a visual indication of the current mode. In 3D, the **Point mode** button has the symbol $\partial^3\Omega$, and the **Edge mode** button uses the same symbol $\partial^2\Omega$ as the **Point mode** in 2D.

MODEL NAVIGATOR

This section explains how to start a new model by selecting application modes and specifying variable names and other model properties in the **Model Navigator**.

OPTIONS AND SETTINGS

This section covers basic settings, for example, the axis or grid spacing settings. These can usually be made with commands from the **Options** menu or by double-clicking on the status bar. Use the **Constants** dialog box to enter model parameters.

GEOMETRY MODELING

Here you create the model geometry using the CAD tools on the draw menu and the draw toolbar.

Physics model

PHYSICS MODELING

In this step you enter all the descriptions and settings for the physics and equations in the model. This part contains one or more of the following sections:

Subdomain Settings

In this section you specify subdomain settings. They describe material properties, sources, and PDE coefficients on the subdomains. On the subdomains it is also possible to specify initial condition and element types.

Boundary Conditions

Here you specify boundary and interface conditions.

Edge Settings

Here you specify edge settings. They describe material properties and PDE coefficients on edges (3D models only).

Point Settings

Here you specify point settings. They describe properties and values for point sources and other values that apply to geometry vertices.

Application Scalar Variables

Some application modes use scalar variables that are independent of the geometry, for example, frequency.

Application Mode Properties

Depending on the application mode, you can change a number of its properties such as analysis type and equation formulations.

Coupling Variables

In extended multiphysics models and for certain applications, you can assign coupling variables that connect variables and expressions in various domains. In this section, you define these coupling variables.

Mesh, Compute, and Postprocess

MESH GENERATION

Here you create the finite element mesh for the model geometry. Normally you simply click the mesh buttons on the main toolbar. In some cases you must use other commands on the **Mesh** menu to customize the mesh.

COMPUTING THE SOLUTION

Often it suffices to click the **Solve** button on the main toolbar. Sometimes you might want to change some solver properties or analysis settings in the **Solver Parameters** dialog box.

POSTPROCESSING AND VISUALIZATION

Here you make the visualization settings as well as perform various postprocessing of the analysis results. You work with the **Plot Parameters** dialog box and other visualization and postprocessing tools.

Summary

- Femlab makes possible to model and simulate physics phenomena.
- Free form entry of partial differential equations and use of application mode.
- Model library for introductory Femlab practices.
- Matlab is linkable to Femlab.