After conductor types, material attributes, boundaries, and sources have been specified, choose **Setup Solution Options** (or **Setup Solution/Options** if you have purchased more than the nominal solution package) from the Executive Commands menu to:

- Select which finite element mesh is used during the solution process.
- Optionally, refine the finite element mesh.
- Specify whether fields and/or executive parameters are computed during a solution.
- Set the stopping criteria for adaptive field solutions.
- Specify the frequency at which eddy current, AC conduction, and eddy axial field simulations take place.
- **EMpulse only.** Define the transient motion solution parameters.
Meshing

Representing an electric or magnetic field solution over a relatively large area is a fairly difficult task. Fields cannot be accurately described with a single polynomial expression that covers the entire problem region. The approach taken by Maxwell 2D is to divide the problem region into many triangles and to represent the field in each triangle (element) with a separate polynomial. This collection of triangles is referred to as the finite element mesh.

An example of a mesh is shown below. The bottom portion of the figure shows the geometry for a three-conductor microstrip line. The top displays the mesh for the geometry.
Need for a Fine Mesh

Although this implementation of the finite element method is largely transparent, a general understanding of the method is necessary to make sure that the field solution (and hence the parameters that are computed) are as accurate as possible for a given amount of computing resources.

Maxwell 2D only computes the electric and magnetic fields at the nodes (vertices) of triangles. To obtain values for the electric or magnetic field at all other points, it interpolates the field from the nodal values of the finite element mesh. For example, in the DC conduction field solver, the value of the electric potential is stored at each node; potentials at points inside the triangles are interpolated from the nodal values.

If the mesh triangles are too large, the fields inside them cannot be interpolated accurately from the nodal values. The optimal mesh for a structure is one that contains enough triangles to accurately represent a field solution but not so many that the available computing resources are overwhelmed. The initial mesh generated for a structure is rarely the optimal mesh. The mesh has to be refined; that is, the geometry has to be broken down into more triangles.

To make the most of the available computing resources, refinement of the finite element mesh is performed automatically by Maxwell 2D. By performing an adaptive refinement analysis, it is able to optimize the placement of all new triangles. However, the Setup Solution Options command enables you to control the adaptive refinement process.
General Procedure

To specify the refinement criteria for the various field simulators:

1. Choose **Setup Solution Options**. The following window appears:

2. Choose the **Starting Mesh** with which to begin the solution process from one of the following:
   - **Initial**. This instructs Maxwell 2D to start with the initial, coarse mesh.
   - **Current**. This instructs Maxwell 2D to use the most recently refined mesh. In general, select this option.

3. Optionally, choose **Manual Mesh** to manually refine the finite element mesh in areas of interest.

4. Specify the **Solver Residual**, which indicates how close the field solution must come to satisfying the appropriate form of Maxwell’s Equations. Generally, accept the default.

5. Specify the **Solver Choice**, which indicates which type of matrix solver to use to solve the problem.

6. If you are setting solution criteria for an AC field simulator, specify a frequency in hertz, kilohertz, or megahertz in the **Frequency** field. All source quantities are
assumed to be sinusoidally oscillating at that frequency.
7. Identify the quantities to be solved for during the solution process:
   • **Fields** — Select to generate a field solution.
   • **Parameters** — Select to solve for the requested executive parameters (force
     torque, current flow, and so forth).
8. Choose **Adaptive Analysis** to request that the finite element mesh be adaptively
   refined in areas with the highest error.
   a. Specify what percentage of high-error triangles should be refined during each
      iteration by entering a value in the field **Percent Refinement Per Pass**.
   b. Specify stopping criteria for the solve — refine — error analysis cycle by
      entering values for **Number of Passes** and **Percent Error**. Maxwell 2D breaks
      out of the cycle as soon as one of these criteria is satisfied.
9. If you want to reset all fields to their default values, choose **Suggested Values**.
10. Choose **OK** to return to the Executive Commands menu.

The process of computing a field solution is an iterative one in which the system con-
verges on a field pattern that satisfies Maxwell’s equations. In general, you should first
accept the default stopping and refinement criteria. Then, after you generate the desired
field solutions, check their convergence to see if the stopping criteria need to be adjusted.

**Starting Mesh**

There are two options for specifying the type of mesh with which to start the solution pro-
cess: **Initial Mesh** and **Current Mesh**.

**Initial Mesh**

If you choose **Initial Mesh**, the system automatically creates a mesh at the start of the
solution process. The initial mesh is a relatively coarse one that, as much as possible,
uses the vertices (object points) of the geometry as the vertices of elements in the mesh.

When using the initial mesh, you should usually perform an adaptive solution that iter-
atively refines the mesh. Because the elements of an initial mesh are relatively large, a
non-adaptive solution that is based solely on the initial mesh is not likely to yield an accu-
rate parameter or field solution.
**Current Mesh**

If you choose **Current Mesh**, the system starts with the finite element mesh that was most recently refined. To take advantage of any previous adaptive or manual mesh refinements, choose **Current Mesh** as the starting mesh.

**Manual Mesh**

Choose **Manual Mesh** to manually refine the finite element mesh in areas of interest. The saved manual mesh then becomes the **Current Mesh**.

**Solver Residual**

The **Solver Residual** specifies how close a field solution must come to satisfying the appropriate form of Maxwell’s equations. In most cases, accept the default value. For magnetostatic problems that contain nonlinear materials, there is both a **Linear** and a **Nonlinear** residual.

The residual is a normalized measure of how close a field solution comes to satisfying the electromagnetic field equation that is being solved. The solution from each iteration is plugged back into the field equation. If it happens to be the exact solution, the residual is zero. Otherwise, the residual is non-zero and a small correction is added to the solution process for the next iteration. The iterative solution process continues until the residual is less than the specified target value.

The residual does not affect the finite element mesh. The system attempts to reduce the residual to the target value while using the same mesh.
Maxwell 2D — Setup Solution Options

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. In versions of Maxwell 2D before 6.4, this option was not available and the software always used an incomplete conjugate gradient solver (ICCG). In version 6.4, the option to use a direct matrix solver has been added. 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. Also, if the matrix size is very small (<1500) the direct solver will be used, since small size matrix normally corresponds to initial coarse mesh with ill-conditioned matrix (the time difference is negligible for small matrices). If the ICCG solver fails to converge while the solver choice is in the Auto position, the software will fall back to the direct solve automatically.

Frequency

If you are computing an eddy current, AC conduction, or eddy axial solution, specify the frequency at which the currents, external fields, and voltages oscillate in the Frequency field. Frequency can be specified in hertz, kilohertz, or megahertz.

The frequency that you choose affects the simulated loss. For example, at high frequencies, the skin effect increases the series resistance by forcing current to the outside of conductors. Magnetic losses associated with materials that have a non-zero imaginary relative permeability also increase with frequency.

Solve For Fields and Parameters

Select one or both of the following field quantities to be computed during the solution process (in general, leave both options selected):

Fields
Solves for your model's electric or magnetic fields.

Parameters
Computes any executive parameters (force, torque, capacitance, impedance, flux linkage, and so forth) or post-processing macros that were requested via the Setup Executive Parameters command.
Transient Solution Options

If you have purchased EMpulse and defined a transient model, the Solve Setup window for transient analysis appears.

To define the solution setup for the transient model, you must specify the motion and solution parameters, such as the speed, distance, and time increments of the motion analysis.

To define the transient solution:
1. Choose Setup Solution/Options from the Executive Commands window. The following window appears:

   ![Solve Setup Window]

   2. Select the type of Starting Mesh to use in the solution. Current uses the most recently defined mesh while Initial uses the mesh that was first created. Optionally, choose Manual Mesh to access the Meshmaker and manually create and refine the mesh.
3. Enter the solver Residual or accept the default. The residual defines how close a
field solution must come to satisfying the appropriate form of Maxwell’s equations.

4. Select Auto, Direct, or ICCG as the Solver Choice. Auto allows the software to choose the best solver for the model. The Direct solver is used for most projects. The ICCG solver is used for significantly larger and more complex meshes in the model. If the ICCG solver fails to converge to a result, it will use the direct solver, which will always converge.

5. Select Start from time zero or Continue previous solution as the starting point of the Solution. The Start from time zero settings forces the motion to begin at its initial conditions while Continue previous solution forces the motion to continue from the last place it stopped moving. The new results are appended to the former results in the Convergence display.

6. Enter the Stop Time. This refers to the values at which the motion will stop.

7. Enter the Time Step for the analysis. This defines the time increments.

8. Enter how often the fields will be saved in the Save fields time step field.

9. For XY models, enter the Model Depth. This refers to how far the object is displaced from its original position.

10. For XY models, enter the Symmetry multiplier. If you entered only a portion of the model, enter the value of how many times to multiply the symmetry to complete the model. For example, if you created only a quarter of the model, enter 4 in this field to complete the geometry.

11. Optionally, select Use Control Program and define the user control program to use during the solution generation process.

12. Optionally, select Suggested Values to restore all entered values to their defaults.

13. Choose OK to accept the solution setup or Cancel to ignore the setup and return to the Executive Commands window.
Transient Models

Transient models are standard geometric models in which one or more objects can be displaced with rotational or translational motion. All motion is constrained to occur within a defined band object. No motion can occur outside the band object.

For translational problems, the mesh outside the band object and inside the moving objects remains constant during the motion solution, while the mesh within the band is updated and recalculated for the displacement at each time step.

Rotational motion models can have only one point of rotation defined in any model. For rotational problems, no remeshing occurs during the solution process.

Translational motion models may have one or more objects that move linearly within the band object. The moving objects are constrained to slide at the same velocity and in the same direction.

Models cannot have objects with both rotational and translational motion.
Adaptive Analysis

Select **Adaptive Analysis** to instruct Maxwell 2D to perform an adaptive solution. During this process, the simulator iteratively refines the starting mesh as illustrated here.

The following happens when an adaptive analysis is performed.

1. Maxwell 2D generates a field solution using the mesh type that you specify.
2. The system computes the field energy and the residual. It then compares the computed residual with the residual that you specified as acceptable in the **Setup Solution Parameters** step. When the computed residual is less than your specified value, the field solution process stops.
3. The simulator computes the **Percent Error**, which is the percentage of the total system energy associated with the residual. The process stops if the **Percent Error** is less than your specified value, or if the specified **Number of Passes** has...
4. New triangles are added to the finite element mesh in areas of high error (residual).
5. Another solution is generated using the refined mesh, and the entire process (solve — error analysis — refine) repeats until the stopping criterion is satisfied.
Percent Refinement Per Pass

The value that you specify as the Percent Refinement Per Pass determines how many triangles are added after each iteration of the adaptive refinement process. For instance, entering 10 in this field causes the ten percent of the triangles with the highest error to be refined. Generally, accept the default value.

Stopping Criterion

Maxwell 2D breaks out of the adaptive solution cycle when one of the following criteria is met.

Number of Requested Passes

Specify the maximum number of refinement cycles (adaptive passes) that you want Maxwell 2D to perform. Typically, use a value between 10 and 15.

The size of the finite element mesh — and the amount of memory required to generate a solution — grows with each adaptive refinement of the mesh. Setting the Number of Requested Passes too high can cause Maxwell 2D to request more memory than is available.

Percent Error

Specify the acceptable Percent Error of the solution. This lets you control the solution accuracy. In general, accept the default for this field. Smaller values produce more accurate (but slower) solutions; larger values produce less accurate (but faster) solutions.

The system stops the adaptive refinement process when both of the following calculated values are less than your specified values:

- The percent error energy. Maxwell 2D computes the total field energy and the energy contributed to this total by the error residual. The percent error energy is the percentage of total energy that the residual contributes. A small percent error energy indicates that only a small amount of the total energy is associated with the residual (error) and that the solution is highly accurate.
- The percent change in energy between adaptive passes. Small energy changes between passes indicate the solution has converged.
Suggested Values

Select this to reset all fields to their suggested values. Doing so gives you a set of solution options that enables you to compute a reasonably accurate adaptive solution.

Use Control Program

User control programs are externally created executables that are called after each time step and allow you to control the source input, circuit elements, mechanical quantities, time step, and when to stop solution generation, based on the updated solutions.

The transient solver outputs current time step solutions to a file with a fixed name of solution.ctl. The solver maintains a copy of the previous solution in a file called previous.ctl because you may need first derivative information for its control purpose.

Control Program

The user control program uses the file with fixed name user.ctl to output control parameters to control the execution of the transient solver. Each file must use a predefined syntax which is flexible enough to cover a wide range of items. The process can be summarized as:

1. The transient solver reads control parameters from user.ctl.
2. The transient solver solves the current time step.
3. The transient solver writes out solution information to solution.ctl and copies the previous solution to previous.ctl.
4. The transient solver calls the user control program;
5. The user-control program writes control information to file user.ctl.
6. Return to the transient solver if the control program succeeds with exit status 0 or fails with exit status non-zero.
7. Go back to step 1 for the next time step.
Activating a Control Program

Use the following steps to access and invoke a user-control program for use with the transient solver.

> To access a user control program:
1. Select **Use Control Program**. The control program field becomes active, allowing you to enter the name of the user control program.
2. Choose **Setup**. The **External Control Program Setup** window appears:

![External Control Program Setup](image)

3. Enter the name of the control program to use in the **Control Program** field.
4. Enter any arguments needed by the control program in the **Arguments** field. The solver will call the program in the format:

   `program_name specified_arguments`

5. Choose **Configure**. This calls the control program, prepending the `-configure` flag to any arguments specified in the **Arguments** field. This option can be used to initialize data before the solver is called.
6. Optionally, select **Call after last time step for post processing**. This instructs the software to invoke the specified user control program after the solution has completed the final time step. This option prepends the `-post` flag to the list of arguments. The solver will call the program in the format:

   `program_name specified_arguments`
The control program will be called if the solve finished normally without any errors or if you choose Stop during the solution process. It will not be called if you choose Abort during the solution generation.

7. Choose OK to accept the configuration or Cancel to ignore the settings. You return to the Solve Setup window.

Control Program File Format

The user control programs have their own file formats which should be followed when creating control program files.

Solution and Previous Control File Formats

The solution.ctl and previous.ctl files are created by EMpulse and uses the following format:

begin_data
time <current_time>
forceT <tangential force>
forceN <normal force>
torque <torque>
speed <speed>
position <position>
winding1 <winding_name> <current_value>
windingV <winding_name> <voltage_value>
windingEMF <winding_name> <back_emf_value>
windingFlx <winding_name> <flux_linkage_value>
...

barl <current_value>
barV <voltage_value>
solidl <object_name> <current_value_for_an_active_solid_conductor>
...

condPwrLoss <total_power_loss_in_solid_conductor>
end_data
Maxwell 2D — Setup Solution Options

User Control File Format

The `user.ctl` file is created by the user control program and uses the following format:

```
begin_data
  windingSrc <winding_name> <source_value>
  windingC <winding_name> <capactiance_value>
  windingR <winding_name> <resistance_value>
  windingL <winding_name> <inductance_value>
  windingConnect <winding_name> <0 or 1>
  ...
  ...
  objectP <object_name> <polarity_value>
  objectSrc <object_name> <source_value_for_solid_conductor>
  ...
  ...
  loadTorque <value>
  loadForce <value>
  speed <value>
  damping <value>
  stop <0 or 1>
  timeStep <value>
  loadInertia <value>
  CallPost2d <macro_file_name>
end_data
```
Manual Mesh Refinement

Choose **Manual Mesh** from the **Setup Solution Options** window to manually refine the finite element mesh in your model using the 2D Meshmaker

**2D Meshmaker**

The 2D Meshmaker appears as shown below:

Use the **Mesh** and **Refine** menus to generate and refine critical areas of the initial mesh.
Meshmaker Commands

The following commands appear in the menu bar of the 2D Meshmaker window:

- **File**: Saves the refined mesh and reads in geometries to be meshed.
- **Mesh**: Seeds, creates, displays, shows information about, or deletes the finite element mesh.
- **Refine**: Allows you to manually refine an existing mesh. You can refine on a point-by-point basis, inside objects, or in a specific area.
- **Model**: Manipulates the parameters associated with the drawing region.
- **Window**: Creates and manipulates viewing windows.
- **Help**: Accesses the online documentation.
Tool Bar

The tool bar serves as a shortcut for executing various commands. Use it to make or delete the mesh, change the way the mesh is displayed, modify object attributes, or change the view into the drawing space.

Each button in the tool bar executes a different 2D Meshmaker command as shown below. When you click on a button, a brief description of the command appears in the message bar at the bottom of the screen.

```
File       Mesh       Refine     Model     Window

Mesh/Seed/Object
Mesh/Make
Mesh/Delete
Mesh/Display
Mesh/Information
Refine/Object
Model/Measure
Window/Change View/Zoom In
Window/Change View/Zoom Out
Window/Change View/Fit Drawing
```
In general, to manually refine the mesh:

1. If desired, add seed points to the model using one of the Mesh/Seed commands. These seed points become the vertices of triangles when you create the finite element mesh.

2. Choose Mesh/Make to generate and display the initial mesh.

3. Use the Refine commands to add more triangles to the mesh by breaking up the larger triangles in the initial mesh into smaller ones — increasing the density of the mesh.

Note: If you delete or refine the mesh after generating a field or parameter solution, the solution is automatically deleted.

There are several reasons for manually refining a mesh. The mesh that the Maxwell 2D creates for your geometry is initially very coarse. During the solution process, the system's adaptive refinement process increases the density of the mesh in areas of high error energy. However, adaptive refinement does not always adequately refine the mesh in areas of low energy (and thus low error energy). You may need to manually add triangles to the mesh in these regions to obtain a more accurate field solution. Generally, there are three basic reasons to manually refine a mesh:

- You know where the critical areas are and wish to give the adaptive refinement process a head start by refining in those areas before starting the solution process.
- The area in which you are interested is an area of low energy density — and does not get refined adequately during the adaptive process.
- You are modeling the skin effect (surface concentration of currents) in a structure via the Eddy Current field solver. The mesh that is initially generated by the 2D
Meshmaker is usually too coarse in the region where eddy currents occur in the conductor.

To get around this problem, refine the mesh between the surface of the conductor and the skin depth (via the Mesh/Seed/Skin command). Refining within this region improves the accuracy of the solution and is especially useful for finding eddy-current effects in conductors that do not carry source current.

There is no reason to refine the mesh in areas where the solution does not change rapidly. In some areas, such as where the solution is constant, an extremely coarse mesh is acceptable. For example, in electrostatic problems there is no need to refine the mesh in an object that is defined as a conductor. The electric potential is the same at all points in the conductor.

But in a problem region in which, for instance, the magnetic field varies dramatically — such as at the corner of an iron core, in an air gap, or wherever flux is channeled through narrow areas — a mesh containing one or two triangles is not going to be adequate. The field in such regions cannot be adequately interpolated from a handful of nodal values. Manually refining the mesh in such areas helps give the adaptive procedure a head start.

Therefore, when manually refining the mesh, your job is to guide the system in refining the mesh for your particular problem to the level necessary to obtain an accurate solution, but not so fine that it overwhelms the available computer memory and processing power or wastes time.

**Undoing a Refinement**

To reverse the effect of a manual mesh refinement, return to the Setup Solution menu and choose Initial Mesh as the starting mesh. Maxwell 2D will generate a new mesh at the beginning of the solution process.

Alternatively, choose Mesh/Delete to delete the current mesh. Then, choose Mesh/Make to create a new one.
Mesh Menu

Use the Mesh commands to create, seed, delete and display information about the finite element mesh. The Mesh commands are:

- **Seed**
  Refines the mesh by adding seed points to the geometric model. These seed points become the vertices of triangles when you create the mesh.

- **Make**
  Creates and displays the finite element mesh.

- **Line Match**
  Matches boundary lines.

- **Delete**
  Deletes the mesh.

- **Display**
  Displays the mesh, object boundaries, and seed points.

- **Information**
  Displays information about the mesh, including the number of triangles inside each object and the maximum and minimum x- and y-coordinates of selected objects.
Before creating the mesh, use the **Mesh/Seed** commands to add seed points to the geometric model. When you generate the coarse initial mesh, these seed points become the vertices of triangles — which increases the density of the mesh in the seeded areas.

To manually seed the mesh:
1. Examine the existing mesh to determine where it should be more heavily seeded. (If none exists, choose **Mesh/Make** to generate an initial mesh.)
2. Choose **Mesh/Delete** to delete the mesh.
3. Choose one or more of the **Mesh/Seed** commands to add seed points where they are needed.
4. Choose **Mesh/Make** to generate a new mesh.

The following **Mesh/Seed** commands are available:

- **Surface**: Adds seed points on the outside surface of an object.
- **Object**: Adds seed points on a grid inside an object.
- **Skin**: Adds seed points to an object in a grid between the surface and the skin depth.
- **QuadTree**: Seeds areas near the surfaces of objects by subdividing the problem space around object edges and adding seed points in a grid.
- **Delete**: Deletes all seed points.
- **SaveSeed**: Saves the seed points for repeated mesh building in non-interactive operations.

When you use the commands for specifying the generation of seed points, you are actually building a set of seeding operations which, when executed, generate a set of seed points for the vertices of additional mesh triangles. If you delete your mesh, the seed points remain, and are still available for the regeneration of the mesh in the on-line mode. However, the seed points are not available to non-interactive operations such as parametric sweeps. The **Mesh/Seed/SaveSeed** command enables you to overcome this limitation by saving the seeding operations information in a file which is available during parametric sweeps. The meshmaker uses the file to regenerate the seed points for subsequent mesh building operations. If you make repeated changes to objects within your model and do repeated mesh generations — as in parametric sweeps — you do not have to re-specify your seeding operations if you have saved them with this command.
Mesh/Seed/Surface

Choose Mesh/Seed/Surface to add seed points on the surface of objects. Use this type of seeding when you are interested in surface concentrations of current or charge.

To seed the surface of an object:
1. Choose Mesh/Seed/Surface. A window appears, listing all objects in the model:

![Surface Seed Window]

2. Select the object whose surface is to be seeded.
3. Enter the distance between individual seed points in the Seed Value field. The seed value is entered in the geometric model's units (displayed in the UNITS field in the status bar).
4. Alternatively, choose Suggested Value to let the software specify a seed value based on the dimensions of the selected object. The seed value is entered in the geometric model's units (displayed in the UNITS field in the status bar).
5. Choose Seed. The seed value appears under Value; the number of seed points to be added appears under # Points.
6. Repeat steps 2 through 4 for each object whose surface is to be seeded.
7. When you are finished, choose OK.

Note: Re-seeding the same surface deletes old seed points and replaces them with the number of new seed points that meets your seeding specification. A seed value of zero deletes all seed points on a selected surface without replacing them.

The system adds the seed points to the object's surface and displays them.
Mesh/Seed/Object

Choose Mesh/Seed/Object to add seed points to the inside of an object. The seed points are added in a rectangular grid. Use this type of seeding if you are interested in modeling charge or current distributions inside objects.

> To seed the interior of an object:
1. Choose Mesh/Seed/Object. The following window appears, listing all objects in the model.

   ![Object Seed Window](image)

2. Select the object that is to be seeded.
3. Enter the distance between individual seed points in the Seed Value field.
4. Alternatively, choose Suggested Value to let the software specify a seed value based on the dimensions of the selected object. The suggested seed value is in the geometric model’s units (displayed in the UNITS field in the status bar).
5. Choose Seed. The seed value appears under Value; the number of seed points to be added appears under # Points.
6. Repeat steps 2 through 4 for each object that is to be seeded.
7. When you are finished, choose OK.

The system adds the seed points in a grid inside the selected objects and displays them.
Mesh/Seed/Skin

Choose **Mesh/Seed/Skin** to add seed points in a grid between the surface of an object and its skin depth. Use this type of seeding in AC magnetics problems where you need to model the effects of eddy currents in conductors. Remember, the skin depth of eddy currents in a conductor is given by:

\[
\delta = \sqrt{\frac{2}{\pi f \sigma \mu_r \mu_0}}
\]

where:

- \( f \) is the frequency of the source current in the conductor.
- \( \sigma \) is its conductivity.
- \( \mu_r \) is its relative permeability.
- \( \mu_0 \) is the permeability of free space, \( 4\pi \times 10^{-7} \) webers/ampere-meter.

Seeding the mesh to the skin depth creates a very dense mesh in this area, which increases the accuracy of the eddy current simulation.

> To seed the skin depth of an object:

1. Choose **Mesh/Seed/Skin**. The following window appears, listing all objects in the model:

2. Select the conductor that is to be seeded.
3. Enter the skin depth seed value in the **Seed Value** field.
4. The 2D meshmaker adds seed points in a grid starting at the outer surface of the object. The seed value specifies the depth and density of the points on this grid.
   - Smaller seed values produce a densely spaced, shallow grid — modeling the skin
5. Alternatively, choose **Suggested Value** to let the software compute a seed value.

**Note:** **Suggested Value** does not actually compute the skin depth. Instead, it computes a seed value based on the object's dimensions.

6. Choose **Seed**. The seed value appears under **Value**. The number of seed points to be added to the object appears under **# Points**.

7. Repeat steps 2 through 4 for each object whose skin depth is to be seeded.

8. When you are finished, choose **OK**.

The system adds the seed points in a grid based on the seed value you specify, and displays them. Below is an example of two conductors with different seed values. The conductor on the right is left unseeded to illustrate the difference in the meshes inside the two objects. Because the mesh inside the right conductor is so coarse in the region where eddy currents occur, the rapidly changing magnetic fields that occur there cannot converge. Seeding the mesh in the left conductor places extra triangles in this area, causing the solution there to be more accurate:
Mesh/Seed/QuadTree

Choose Mesh/Seed/QuadTree to add seed points to a geometry by subdividing the area around the outside edges of objects, then adding seed points in a grid based on these subdivisions. Use this type of seeding to produce a uniform mesh around the edges of objects in your model.

As shown below, the seed value you specify determines how the area is divided up and seeded. Two identical geometries have been seeded using different values.

• The geometry on the left side of the figure below was seeded with a value of 4. Since there are fewer divisions, the seed points are more evenly distributed throughout the problem region.

• The geometry on the right side of the figure below was seeded with a value of 8. When you specify a larger seed value, the 2D Meshmaker adds more divisions near the edges of the objects in the geometry — producing a dense grid of seed points in these areas.

To add seed points to the model using the quadtree method:
1. Choose Mesh/Seed/QuadTree. A window appears, prompting you to enter the number of levels (divisions) for the quadtree seeding.
2. Enter the **Number of Levels**. Seed values must be between zero and eight.
3. Choose **OK** or press **Return**.

The seed points are added to the model.

**Mesh/Seed/Delete**

Choose **Mesh/Seed/Delete** to delete any seed points that have been added to the geometric model.

> To delete the seed points:
1. Choose **Mesh/Seed/Delete**. A message appears asking you to confirm the deletion.
2. Choose **OK** to delete the seed points or **Cancel** to cancel the deletion.

**Note:**

> To delete individual seed points:
1. Choose one of the **Mesh/Seed** commands.
2. Specify a seed value of zero for each point that you want to delete.
3. Re-seed the mesh.

**Mesh/Seed/SaveSeed**

Choose this command to save the seedings of the mesh.

**Mesh Seeding for Parametric Sweeps**

> To use the same seed points for repeated mesh building in non-interactive operations:
1. Set up seed point operations by using the **Mesh/Seed/Surface**, **Mesh/Seed/Object**, **Mesh/Seed/Skin**, or **Mesh/Seed/QuadTree** command.
2. Choose **Mesh/Seed/SaveSeed** to save the seed point operations for use in the non-interactive mode.

**Deleting Mesh Seeding Operations**

> To clear the seed operations file by zeroing its values:
1. Choose **Mesh/Delete** to delete the mesh and thereby enable the **Mesh/Seed/Delete** command.
2. Choose **Mesh/Seed/Delete** to delete the seed points.
3. Choose **Mesh/Seed/SaveSeed** to clear all values in the seed operations file.
Mesh/Make

Choose Mesh/Make to create a finite element mesh for the model. In addition to the object points used as vertices, any seed points that you have added to the model (using one of the Mesh/Seed commands) are used as vertices of triangles.

When the 2D Meshmaker finishes generating the mesh, it automatically displays the mesh lines, seed points, and object boundaries.

To cancel the mesh generation, select the Abort Mesh Generation button, which appears while the system is making the mesh.

Triangle Aspect Ratios

The aspect ratios of the triangles in the mesh — the ratio of the base of a triangle to its height — are controlled by the Maxwell 2D Meshmaker. During the mesh refinement process, an aspect ratio for each triangle to be created is automatically calculated. Triangles whose aspect ratios are too small are not added to the finite element mesh.
Mesh/Line Match

Choose **Mesh/Line Match** to force the mesh points on a slave boundary to match the mesh points on its master boundary. This takes advantage of the periodicity in the problem and forces the mesh along the slave line to use the same number of vertices as on the master line.

### Warning:
Keep the following in mind when using this command:
- You must manually line match all matching boundaries defined using the **Setup Boundaries/Sources** command.
- For line matching to work, neither the master boundary nor the slave boundary can lie along the edge of the drawing space. They must always border an excluded object (such as the background).
- You must select at least one point on the master boundary.

To match a master and slave boundary:
1. Choose **Mesh/Line Match**.
2. Select the first point of the master line. To select the point, click the left mouse button on an object vertex that lies on the line.
3. Select the second point of the master line.
4. Select the first point of the slave line.
5. Select the second point of the slave line.

Once you have selected the second point of the slave line, the new mesh will automatically be generated.

### Note:
On matched lines, the point distributions will always match. However, the triangles that abut the matched lines need not match exactly or even approximately.
Point Placement

The order in which the first point and the second point are selected will affect the results of the line match. The figure below illustrates this. The position of line segment “a” is different on the two slave lines, even though they both are matched to the same master line. This is because the order in which the end points were selected are opposite for the two slave lines.

```
Master

1   a
  |   |   |
  2   1   2

Slave 1

1   a
  |   |   |
  2   1   2

Slave 2
```

In general, select the slave line using the same direction that you selected the master line (from left to right, top to bottom, and so forth).

Mesh/Delete

Choose **Mesh/Delete** to delete the finite element mesh.

> To delete the current mesh:
1. Choose **Mesh/Delete**. A message appears asking whether you are sure you want to delete the mesh.
2. Choose **OK** to delete the mesh or **Cancel** to cancel the deletion.

**Note:** If you delete the mesh, all existing field or parameter solutions will also be deleted. However, seed points are not deleted.
Mesh/Display

Choose **Mesh/Display** to display the finite element mesh. This command enables you to view or hide the following:

- The triangles in the finite element mesh.
- Any seed points that have been added to the mesh (using the **Mesh/Seed** commands).
- The outlines of geometric objects.

To display (or change the display of) the mesh:
1. Choose **Mesh/Display**. A window appears, listing the following options:
   - **Display Mesh**: Display triangles in the mesh.
   - **Display Seed Points**: Display the seed points in the mesh.
   - **Display Boundary Lines**: Display the outlines of objects in the model. This allows you to see objects more clearly when the mesh is displayed.
2. Do one of the following:
   - To display one of these options, click the left mouse button on it to select it.
   - To turn off its display, click the left mouse button on it again to deselect it.
3. Choose **OK** or press **Return**.
Mesh/Information

Choose Mesh/Information to view information on the finite element mesh. The following window appears when you select this command:

Each object in the model (including background) is listed under Object Name. The number of triangles in each object is shown under # Triangles.

To view the dimensions of an object, click the left mouse button on it to select it. Its dimensions appear in the following fields:

- Min (X) The object’s minimum x-coordinate.
- Min (Y) The object’s minimum y-coordinate.
- Max (X) The object’s maximum x-coordinate.
- Max (Y) The object’s maximum y-coordinate.

When you are done, choose OK to return to the 2D Meshmaker.
**Maxwell 2D — Manual Mesh Refinement**

**Refine Menu**

Use the **Refine** commands to refine the finite element mesh. Before using these commands, either read in an existing mesh or choose **Mesh/Make** to generate and display the initial finite element mesh. The **Refine** commands are:

- **Point** Adds points to the mesh at the triangle where you click the mouse, which then become the vertices of new triangles.
- **Area** Refines the mesh within a rectangular area.
- **Object** Refines the mesh within the selected object.

To abort a refinement, click the left mouse button on the **Abort Refinement** button, which appears while the mesh is being refined.
Choose **Refine/Point** to refine the mesh by adding points when you click the mouse. The points you add become the vertices of new triangles, increasing the number of triangles in the mesh.

Use this refinement method when you only need to add a few triangles to the mesh. If you need to refine the mesh in a larger area or inside a specific object, use the **Refine/Area** or **Refine/Object** commands.

Two methods for adding points to the mesh are available:

- **Circumcircle**
  - The point is added at the center of a circle whose circumference is defined by the vertices of the triangle, as shown above.

- **Centroid**
  - The point is added at the center of the triangle, as shown above.

In general, circumcircle refinement produces triangles with lower aspect ratios than centroid refinement — that is, the triangles added using the circumcircle method have sides which are more nearly equal. The computation error in such triangles is generally lower than in triangles with long, thin sides.

To refine the mesh on a point-by-point basis:

1. Choose **Refine/Point**. A window appears, asking you to define the method for adding triangles to the mesh.
2. Choose **Circumcircle** or **Centroid** as the method for adding triangles to the mesh.
3. Choose **OK** to begin adding points to the mesh. The cursor changes to a cross.

4. Click the left mouse button on a triangle to refine it. (Alternatively, select the point with the **keyboard**. The 2D Meshmaker adds the point to the mesh, where it forms the vertex of new triangles. (The exact location of the point depends on whether you selected **Circumcircle** or **Centroid** refinement.) It then redraws the mesh in the area where the point was added, showing the new triangles.

5. To stop refining the mesh, click the right mouse button.

### Refine/Area

Choose **Refine/Area** to refine the mesh in a rectangular area. This type of mesh refinement is generally done when you want to refine the mesh in areas that:

- Are not bounded by a geometric object (for instance, an area within the background object where you need accurate field values).
- Overlap several objects.
- Are too large for point refinement to be effective.

> To refine the mesh in the selected area:

1. Choose **Refine/Area**. The cursor changes to a cross.
2. Click the left mouse button on the first diagonal corner of the rectangle. (Alternatively, select the point with the **keyboard**.
3. Click the left mouse button on the second diagonal corner. (Again, as an alternative, select the point with the keyboard.) The **Area Refinement** window appears. The number of triangles that are currently in the mesh appears in the **Number of Triangles** field:
Maxwell 2D — Manual Mesh Refinement

4. Enter the total number of triangles in the mesh in the Desired number of triangles field. The 2D Meshmaker adds points to the mesh until the total number of triangles reaches the number that you specify. By default, the Desired number of triangles field is set to twice the number of triangles in the current mesh.

5. Select the method for adding points to the mesh. These points then become the vertices of new triangles. Two methods for adding points are available:
   - Circumcircle The point is added at the center of a circle whose circumference is defined by the vertices of the triangle.
   - Centroid The point is added at the center of the triangle.

6. Choose OK or press Return to begin refining the mesh. A window appears, allowing you to monitor the refinement’s progress in the following ways.
   - Refinement Goal View-only field showing the desired number of triangles in the mesh (entered in the Desired number of triangles field).
   - Number of Triangles View-only field showing the current number of triangles in the mesh. Updates periodically as triangles are added.

   When the 2D Meshmaker reaches the refinement goal, the mesh refinement stops.

   Aborting an Area Refinement

   You can abort a mesh refinement before it is complete. For instance, you may wish to stop the refinement if you decide that the mesh in the selected area is sufficiently dense before the refinement is finished.

   > To stop the mesh refinement:
     - Choose Abort Refinement.

   The mesh refinement stops.
Refine/Object

Choose Refine/Object to refine the mesh inside specific objects. You may specify either:

- The desired number of triangles in each object.
- The minimum area of each triangle.

The 2D Meshmaker refines the mesh inside the object until one of these goals is reached. The following window appears when you select this command:

> To refine the mesh inside an object or objects:
1. Choose Refine/Object.
2. Select the desired object.
3. Do one or both of the following:
   - Enter the maximum area of the object’s triangles under Refine Area.
   - Enter the desired number of triangles inside the object under Refine Number.
4. Choose Accept. The new Refine Area and Refine Number values appear for the selected object.
5. Repeat steps 2 through 4 for each object to be refined.

6. Choose Ok or press Return to begin refining the mesh in the selected objects. A window appears, showing the current number of triangles in the mesh. It updates periodically as the 2D Meshmaker adds points to the mesh, creating new triangles. By default, the Refine/Object command uses circumcircle refinement. When either the Refine Number or Refine Area is reached, the mesh refinement ends.

Object Information

The view-only fields in this window display the following information for each object:

- **Object Name**: The name of the object.
- **# of Triangles**: The current number of triangles inside the object.
- **Goal #**: The desired number of triangles inside the object (initially set to zero). Displays the number entered under **Refine Number**.
- **Goal Area**: The desired area of each triangle (initially set to zero). Displays the area entered under **Refine Area**.

**Note:** If the largest triangle in an area or an object is unrefineable, the refinement quits prematurely. A triangle becomes unrefineable when it is too small or when a highly refined region is very close to a coarse region. To continue with manual refinement after this happens, use the Refine/Point command.

**Refine Area and Refine Number**

Specify the criteria for refining the mesh inside an object in these two fields:

- **Refine Area**: The maximum desired area of each triangle in the selected object, entered in the units used for the model (shown to the right of the Enter button on the screen).

- **Refine Number**: The desired number of triangles inside the selected object.
Aborting an Object Refinement

As with the **Refine/Area** command, you can abort a mesh refinement for an object before it is complete. For instance, you may wish to stop the refinement if you decide that the mesh in the selected area is sufficiently dense before the refinement is finished.

> To stop the mesh refinement:
• Choose **Abort Refinement**.

The mesh refinement stops.
EMpulse

The transient solver allows you to analyze the magnetic fields, energy, force, power loss, speed, and flux of a model at various time steps of a solution over a specified period of time. This solver allows for non-sinusoidal current or voltage excitation, as well as rotational or translational motion defined in the Motion Setup window.

This solver becomes available only when the EMpulse license has been installed. Solvers for packages that you have not purchased — or have not yet entered an authorization codeword for — cannot be selected and are disabled. See the Maxwell Installation Guide for instructions on entering codewords for the various solver packages.

Note:

Only access the Motion Setup window when you need to define the motion on objects. For non-sinusoidal input problems, you need to define only the attributes of the model and generate the solutions as you would for a static model.
Transient Excitation

The transient solver allows for non-sinusoidal excitation such as square waves or harmonically distorted waveforms. Transient voltage sources can be assigned to solid or stranded conductors. Transient current sources can be assigned to solid or stranded conductors as either a current or current density.

For example, using the Functions button, you may assign a 120 volt, 60 Hz fundamental harmonic voltage source with a 20 volt, 5 kHz harmonic waveform superimposed on top. The resulting wave can then be analyzed at each time step of the solution.

Here, the time-dependent voltage is given by $V = 120 \sin(60 \times 360 \times t) + 20 \sin(5000 \times 360 \times t)$. 
Transient Motion

The transient solver allows for the motion of various objects within the model. **Translational** or **rotational** motion are possible.

The transient solver assumes the following conditions about the problem:
- If motion occurs in the model, no motion occurs outside the band object. The band object is the object within which all moving objects are contained.
- Only one type of motion (translational or rotational) occurs in the model.
- More than one object can be assigned identical motion within the band object.

Motion Attributes

Motion attributes are set by selecting only the *band* object in the **Motion Setup** window. Once an object has been defined as a band, the software automatically determines which objects are inside and marks them as moving.

Moving objects may possess either **translational** or **rotational** motion.

**Note:** RZ models can only possess translational motion, with their direction of motion restricted to the z-axis.

Units of Motion

The solver outputs the resulting plots of speed in the units you specified for the motion in the **Mechanical Setup** window.

For rotational problems, speed is given in either rpm or degrees/sec, depending on what you select for the units in the **Mechanical Setup** window. For translational problems, the speed is given in the defined unit of length (millimeters, inches, feet, and so forth) per second.

Similarly, all variables defined in the **Motion Setup Symbol Table** are evaluated in the units defined in the **Mechanical Setup** window. Thus, any functional values you define retain their units throughout the solution process.
If you have purchased the license for the time-stepping analysis package, EMpulse, and selected *Transient* as the solver type, choose **Motion Setup** from the **Setup Solution** menu to do the following:

- Define the translational or rotational motion of an object.
- Specify which objects are mobile and which are stationary.
- Analyze the motion at various time steps during the solution.

The **Motion Setup** window appears, showing the current model:

All existing objects and their respective movement attributes are listed in the box on the left side of the **Motion Setup** window. This area lists any undefined objects as **stationary**.

The model appears in the display area in the upper right side of the window. If you select the name of an object, the object in the viewing window is highlighted and any information about the object appears below the viewing window.
General Procedure

The general procedure for setting up and solving a transient model is as follows.

To define the transient model:

1. Choose Motion Setup from the Executive Commands menu. The Motion Setup window appears.
2. Toggle Single Select or Multi Select to define the single or multi-selection.
3. From the Select menu, choose the method of selection. Choose Deselect at any time to deselect all selected objects.
   • Choose By Area to select the objects in a defined area. To select objects by area:
     a. Choose By Area from the menu. The cursor changes to crosshairs.
     b. In the viewing window, click on the first corner of the box in whose area the objects will be selected.
     c. Click on the opposite corner of the box. All objects contained within the box are selected.
     Click the right mouse button at any time to end the command without selecting an object.
   • Choose By Name to select an object by clicking on the name in the object list.
   • Choose All Objects to select all the objects in the model.
4. Optionally, select the object whose motion is to be defined. For non-sinusoidal inputs, do not modify the motion attributes of any objects. By default, all objects in the model are stationary.
5. If motion occurs in the problem, define the object types as follows:
   • Select Set Band to define the object as a band object. A band object is a stationary object that contains the moving objects in the model. Once a band object is selected, all objects within it are given Moving attributes. Select Unset Band to make all objects stationary.
   • For objects with rotational motion, do the following:
     a. Select Rotation as the Type of Motion. Rotational motion moves the object about a defined point.
     b. Choose Set Position. The cursor changes to crosshairs.
     c. Double-click on the center of rotation in the viewing window or enter the U and V coordinates in the coordinates fields at the bottom of the window.
     d. Enter the Positive and Negative rotation limits in their respective fields.
For objects with translational motion, do the following:

a. Select **Translation** as the **Type of Motion**. Translational motion slides the object along a defined path.

b. For XY models, select **x-axis** or **y-axis** as the **Direction of Motion** or select the **angle from a-axis** field and enter the angle of translation from the x-axis.

c. For RZ models, select either **positive z-axis** or **negative z-axis** as the **Direction of Motion**.

d. Under **Translation Limits** enter the **Positive** and **Negative** limits of motion. These values refer to the deviation of position from the initial point.

6. Enter the initial position of the point of motion in the **Initial Pos** field.

7. Choose **Mechanical Setup** and define the inertia, torque, and basic motion settings.

8. Choose **Exit**. A window appears, prompting you to save your settings before exiting. Do one of the following:
   - Choose **Yes** to save the motion setup and exit.
   - Choose **No** to exit the window and ignore any changes.
   - Choose **Cancel** to remain within the **Motion Setup** window.
Mechanical Setup

Choose this to define the mechanical parameters of the motion setup. This allows you to specify the basic physical constraints of motion prior to executing the simulation.

To define the mechanical parameters:
2. Do one of the following:
   - For rotational motion, leave Consider Mechanical Transient selected to keep the setup fields active and enter the following values in their respective fields:
     - The Initial Angular Velocity of the moving object.
     - Optionally, select rpm or degrees/sec as the units for the initial velocity. This setting also determines in which units to output the velocity.
     - The Moment of Inertia on the rotating object in kg/m².
     - The Damping, in N·m·s/rad.
     - The Load Torque on the system, entered in N·m.
   - For translational motion, leave Consider Mechanical Transient selected to keep the setup fields active and enter the following values in their respective fields:
     - The Initial Velocity of the moving object in mm/sec.
     - The Mass of the moving object in kg.
     - The Damping, in N·s/m.
     - The Load Force, in N.
   - For either motion type, deselect Consider Mechanical Transient when you wish to define the angular velocity of the system as a constant. Enter the constant value in the Constant Velocity or Constant Angular Velocity fields.
3. Optionally, choose Options to select any parameters you wish to make functional. A window appears, prompting you to set the Damping, Load, or Velocity as functional parameters. Select any values you wish to make functional and choose OK to accept the options. The window closes.
4. Optionally, choose Functions to access the Motion Setup Symbol Table and define any functional values.
5. Choose OK to accept the setup or Cancel to cancel the assignment.

You return to the Motion Setup window.
Functional Mechanical Values

When you choose **Functions** from the **Mechanical Setup** window, an expression evaluator appears, listing the defined **functions** and their values:

![Expression Table]

To define the functions:
1. Select the function from the **Name** list or enter a new functional symbol in the blank field to the left of the equals sign. By default, \( P \) is the position variable, \( S \) is the speed variable, and \( T \) is the time variable.
2. Optionally, choose **Dataset** and define a set of piecewise linear **expressions**.
3. Enter the value of the function in the field to the right of the equals sign.
4. Do one of the following:
   - Choose **Update** to assign the new definition to an existing variable.
   - Choose **Add** to add a newly defined symbol to the table.
5. Choose **Done** to close the window.

You return to the **Mechanical Setup** window.

**Deleting Symbols**

To delete a symbol from the expression evaluator:
1. Select the name of the symbol to delete.
2. Choose **Delete**.
Functional Parameter Values

In EMpulse, the type of conductor in the model determines its range of functional variables.

Source Variables

The following range of functional source variables are available for their respective conductors:

- **Perfect conductor**
  - Total current source: `Func(p,s,t)`
- **Solid conductor**
  - Current source: `Func(p,s,t)`
  - Voltage source: `Func(p,s,t)`
- **Stranded conductor**
  - Current density: `Func(x,y,p,t,r,z)`
  - Current source: `Func(p,s,t)`
  - Voltage source: `Func(p,s,t)`

Winding Variables

The following range of functional variables are available for windings:

- **Polarity**: `Func(p,s,t)`
- **Resistance**: `Func(p,s,t)`
- **Inductance**: `Func(p,s,t)`
- **Capacitance**: `Func(p,s,t)`

Solid Conductor Variables

The following range of functional variables are available for solid conductors:

- **End resistance**: Must be constant.
- **End inductance**: Must be constant.
Mechanical Transient Variables

The following functional variables are available for mechanical transient setups:

- Initial Velocity: Func(p,s,t)
- Load Force: Func(p,s,t)
- Damping: Func(p,s,t)

Magnetization and Value Boundary Variables

The following functional variables are available for the magnetization and value boundaries in the model:

- Value boundary: Func(x,y,r,z)
- Magnetization angle: Func(x,y,r,z)
- Magnet, $H_{cx}$: Func(x,y,r,z)
- Magnet, $H_{cy}$: Func(x,y,r,z)
Modifying the Motion Setup

Once a motion setup has been specified, you can modify its motion attributes.

To modify the existing motion setup for an object:

1. Select the object to modify. The selected object is highlighted in the viewing window.
2. Define the type of object as follows:
   - Choose **Set Band** to define the object as a band object. The band object is a stationary object that contains the moving objects in the model. Once a band object is selected, all objects within it are given **Moving** attributes, while all objects outside it are given **Stationary** attributes. Select **Unset Band** to make all objects stationary.
   - Select **Rotation** as the **Type of Motion**. Rotational motion moves the object about a defined point. For objects with rotational motion, do the following:
     a. Choose **Set Position**. The cursor changes to crosshairs.
     b. Double-click on the center of rotation in the viewing window or enter the $U$ and $V$ coordinates in the coordinates fields at the bottom of the window.
     c. Enter the **Positive** and **Negative** rotation limits in their respective fields.
   - Select **Translation** as the **Type of Motion**. Translational motion slides the object along a defined path. For objects with translational motion, do the following:
     a. For XY models, select **x-axis** or **y-axis** as the **Direction of Motion** or select **angle from x-axis** field and enter the angle of translation from the x-axis.
     b. For RZ models select **positive z-axis** or **negative z-axis** as the **Direction of Motion**.
     c. Under **Translation Limits** enter the **Positive** and **Negative** limits of motion. These values refer to the deviation of the position from the initial point.
3. Enter the initial position of the moving object in the **Initial Pos** field.
4. Choose **Exit**. A window appears, prompting you to save your settings before exiting. Do one of the following:
   - Choose **Yes** to save the motion setup and exit.
   - Choose **No** to exit the window and ignore any changes.
   - Choose **Cancel** to remain within the **Motion Status Setup** window.
Rotational Motion

Use rotational motion for objects that rotate about a specified point in a system. The rotating object must be contained within a band object. No part of the rotating object may come in contact with the band object.

> To define the rotational motion of an object:
  1. Select the Moving object for which to define the motion.
  2. Select Rotation as the type of motion.
  3. Choose Set Position. The cursor changes to crosshairs.
  4. Define the center of rotation in one of the following ways:
     - Double-click on the center of rotation in the viewing window.
     - Enter the U and V coordinates in their respective fields and choose Enter.
  5. Enter the Positive and Negative rotation limits in their respective fields.
  6. Enter the initial position of the object in the Initial Pos field.

An X appears in the viewing window, marking the point around which the selected object rotates.

Translational Motion

Use translational motion for objects that move from one point to another along a specified path. Objects with translational motion are constrained to move within a surrounding band object.

> To define the translational motion of an object:
  1. Select the Moving object for which to define the motion.
  2. Select Translation as the type of motion.
  3. For XY models, select x-axis or y-axis as the Direction of motion or select angle from x-axis field and enter the angle of translation from the x-axis.
  4. For RZ models select positive z-axis or negative z-axis as the Direction of Motion.
  5. Under Translation Limits enter the Positive and Negative limits of motion. These values refer to the deviation from the initial position.
  6. Enter the initial position of the object in the Initial Pos field.

An arrow appears in the viewing window, indicating both the direction and distance of the displacement.
The view window allows you to see the various objects in the model as you assign materials to them.

**Changing the View of the Geometric Model**

Once the model appears on the screen, use the commands that appear beneath the window to change your view of it.

**Zoom In**

> To zoom in on a section of the geometric model:
> 1. Choose **Zoom In**.
> 2. Click the left mouse button on a point at one corner of the area to enlarge.
> 3. Click the left mouse button on the diagonal corner of the area.

The system then expands the portion of the structure in the selected region to fill the view window. This command works in the same way as the 2D Modeler **Window/Change View/Zoom In** command.

**Zoom Out**

> To zoom out of a section of the geometric model:
> 1. Choose **Zoom Out**.
> 2. Click the left mouse button on a point at one corner of the area to reduce.
> 3. Click the left mouse button on the diagonal corner of the area.

The system then redraws the screen and shrinks the model to fit in the selected region. This command works in the same way as the 2D Modeler’s **Window/Change View/Zoom Out** command.
Maxwell 2D — EMpulse

Fit All

Choose **Fit All** to view the entire geometric model in the display area. The Maxwell 2D automatically adjusts the field of view, making all objects as large as possible while keeping the entire structure visible. This command works in the same way as the 2D Modeler's **Window/Change View/Fit All** command.

Fit Drawing

Choose **Fit Drawing** to display the entire drawing region. The drawing region is defined using the command **Model/Drawing Size**. This command works in the same way as the 2D Modeler's **Window/Change View/Fit Drawing** command.

Fill Solids

Choose **Fill Solids** to display closed geometric objects as filled-in solids. By default, only the outlines of object borders are displayed. Choosing **Fill Solids** for complex geometries allows you to better visualize the relationships between each object in the model.

The **Fill Solids** and **Wire Frame** commands work in the same way as the 2D Modeler's **Window/Change View/Fill Solids** and **Window/Change View/Wire Frame** commands.

Wire Frame

After you choose **Fill Solids**, its button toggles to **Wire Frame**. Choose **Wire Frame** to switch back to a wire frame view of the geometric model.

Window Commands

The **Motion Setup** window's **Window** commands do the following:

- **Measure** Displays the distance between two selected points.
- **Grid** Defines the grid settings in the viewing window.
- **SnapTo Mode** Toggles the snap mode on and off for grids and vertices.
Maxwell 2D — EMpulse

**Window/Measure**
Choose this command to measure the distances between two points.

- To measure the distances:
  1. Choose **Window/Measure**. The cursor changes to crosshairs.
  2. Select the first point from which to measure.
  3. Select the second point. The **Measurement** window appears, listing the selected points, offset, distance, and relative angle.
  4. Choose **OK** to close the window.
  5. Click the right mouse button to exit the command.

**Window/Grid**
Choose this command to define the grid settings for the viewing window.

- To define the grid settings:
  1. Choose **Window/Grid**. The **Grid Settings** window appears.
  2. Select **Cartesian** or **Polar** and enter the spacing values for \( dU \) and \( dV \) or \( dR \) and \( dTheta \).
  3. Optionally, choose **Suggested Spacing** to use the default grid values if they have been modified.
  4. Toggle **Grid Visible** to select whether the grid is visible in the viewing window. The grid is visible by default.
  5. Toggle **Draw Key** to select whether the coordinate arrows (or “draw key”) are visible. The draw key is visible by default.
  6. Choose **OK** to accept the grid settings or **Cancel** to cancel the grid changes.

**Window/SnapTo Mode**
Choose this command to define the snap-to settings. Snaps define how close a point comes to touching a grid point or axis point.

- To define the snap-to settings:
  1. Choose **Window/SnapTo Mode**. The **SnapTo Mode** window appears.
  2. Toggle **Snap to grid** on or off to set the snaps on or off the grid lines.
  3. Toggle **Snap to vertex** on or off to set the snaps on or off the vertices.
  4. Choose **OK** to accept the snap-to settings or **Cancel** to cancel the changes.
Modifying the Motion Status Setup

If you choose **Motion Setup** after generating a field solution, the following message appears:

If you make changes to the motion status setup and save those changes, the transient solution data will be deleted and will have to be recomputed. Pick “View Only” if no changes are to be saved, “Modify” if changes are to be saved or “Cancel” to cancel this operation.

> Do one of the following:
  - Choose **View Only** to access EMpulse in view-only mode. You will be able to select and view all motion parameters; however, you will not be able to change them.
  - Choose **Modify** to change the existing motion parameters. Note that you must re-solve the problem after doing so.
  - Choose **Cancel** to return to the Executive Commands menu.

Exiting the Motion Setup

Once the motion setup is complete, exit the **Motion Setup** window and save your settings.

> To save the motion settings and exit:
  1. Choose **Exit**. A window appears, prompting you to save your settings before exiting.
  2. Do one of the following:
     - Choose **Yes** to save the motion setup and exit.
     - Choose **No** to exit the window and ignore any changes.
     - Choose **Cancel** to remain within the **Motion Setup** window.

The **Motion Setup** window closes. You return to the Executive Commands window.
Setting up a Parametric Solution

Maxwell 2D with parametrics has two commands for setting up the solution process. Choose them from Setup Solution in the Executive Commands menu.

Setup Solution/Options

This window is visually identical to the one described in the nominal solution documentation. The functional differences are as follows:

- Choosing Current as the starting mesh will allow variable solutions to be completed faster. Choosing Initial causes the solver to use the initial mesh to begin solving each variation. When Current is selected, the solver will use the most refined mesh available if the project geometry has not changed from a previous variation.
- Manual meshing only affects the basic (or “nominal”) setup. It has no meaning for variable solutions.
- Solve for: Fields is always considered to be turned on, regardless of the state of the checkbox.
- Adaptive Analysis is always considered to be turned on, regardless of the state of the checkbox.
- Frequency is always considered a variable, and can be used during a parametric solution. The frequency you specify here will be used as the nominal value.

You may wish to request fewer solution passes, since the solver will attempt that many passes for each variation.
Setup Solution/Variables

Choose this command to access the parametric table, which allows you to:

- Define which variables to use for a variable solution.
- Display the current list of variables and their values

When you choose **Setup Solution/Variables** from the Executive Commands menu, the following window appears:

The icons in the tool bar execute commands that are also on the menus.
Parametric Solution Options Commands

The following menus are available when setting up a parametric solution:

- **File**: Use the **File** commands to open existing files, to save the spreadsheet being created to disk files, and to exit.
- **Edit**: Lets you edit cells in the table and insert or delete rows (parametric setups).
- **Variables**: Lets you add, delete, animate, and view variables in the parameter sweep.
- **Data**: Lets you specify variable values for a parameter sweep.
- **Window**: Use the **Window** commands to select the active project window, to create new subwindows, and to manipulate project windows and sub-windows.

Parametric Solution Options Tool Bar

The icons in the tool bar execute commands that are also on the menus. The definitions of the tool bar icons are shown below:

- **File**
- **Edit**
- **Variables**

<table>
<thead>
<tr>
<th>Icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data/Fill</td>
<td></td>
</tr>
<tr>
<td>Data/Sweep</td>
<td></td>
</tr>
<tr>
<td>Data/Sort</td>
<td></td>
</tr>
<tr>
<td>Edit/Deselect All</td>
<td></td>
</tr>
<tr>
<td>Variables/Animate</td>
<td></td>
</tr>
</tbody>
</table>
### General Procedure

1. Choose **Setup Solution/Variables** from the Executive Commands menu.
2. Use the **Variables/Add** command in the table editor to add variable columns to the table for each variable you wish to vary in the parametric solution.
3. Use the **Data/Sweep** command in the table editor to add rows to the table for each setup you wish to solve.
4. Save the table, and exit the table editor using the commands on the **File** menu.

### Selecting Cells

You can select cells in a variety of ways:

- **Click** Select a single cell.
- **Click & Drag** Select a rectangular region of cells.
- **Shift-click** Extend the rectangular selection to include the target cell.
- **Control-click** Add a disconnected cell to the selection.
- **Control-click & Drag** Add a disconnected rectangular region to the selection.

Performing the above actions on a column or row heading selects those columns or rows.

When working with the table, commands are only executed on the current selection. For instance, if you select the **Variables/Animate** command with five rows selected, only those five rows are animated. If no cells are selected, the command operates on all the rows in the table.
Editing Individual Cells

Once you have an individual cell selected, you can edit its contents by directly replacing the value in the cell, or with the fields at the bottom of the active window:

> To change the values manually:
1. Select the cell or cells you wish to edit. The value of that cell appears in the entry field. The coordinates of the cell appear to the right of the entry field.
2. Do one of the following:
   - Double-click on the selected cell and enter a new value for it.
   - Use the Value field to edit the value:
     a. Click in the Value field at the bottom of the window and enter a new value for the cell. If you are editing a boolean (yes/no) cell of the table, only a Y registers as a “yes.” Any other letter or number registers as a “no.”
     b. Choose Enter. If the value you have entered is not legal for any of the selected cells (for instance, if you have both boolean and numeric cells selected) then Enter will not be enabled.

The value of the cell is now changed.
Setting up a Parametric Solution
Setup Solution/Options
Setup Solution/Variables
   Parametric Solution
   Options Commands
   Parametric Solution
   Options Tool Bar
General Procedure
Selecting Cells
Editing Individual Cells
Setup Solution/Variable Control
Configuring the Control Settings
   Motor Configuration
   Optimization Configuration

Setup Solution/Variable Control

Once the parametric table has been defined, choose this command to define the control program to use during the solution.

To define the control program:
1. Define the parametric table.
2. Choose Setup Solution/Variable Control. The following window appears:

3. Select Use control program to activate the fields and commands and define the table control settings.
4. Optionally, choose Motor to use the default motor control program. Once you choose Motor, the control program name appears in the Program Name field.
5. Optionally, choose Optimize to use the default optimization program. Once you choose Optimize, the optimization program name appears in the Program Name field.
6. Optionally, choose Configure to configure the selected control program settings.
7. Choose OK to accept the control settings.
Configuring the Control Settings

Once you have selected a control program, you can configure it for more appropriate settings.

Motor Configuration

If you selected Motor as the control program, choose Configure to define the motor settings. Note that you must have a license for Maxwell EMSS for this command to become active.

To define the motor settings:
1. After selecting Motor as the type of control program, choose Configure. The EMSS Motor Setup window appears.
2. Select the Machine Type to use from the pull-down menu. The core specifications for the motor appear in the Value list.
3. Optionally, select Do Transient Parameters to perform a time-stepping parametric analysis for the motor.
4. Enter a Description for the program in the blank field.
5. Select Wye or Delta as the type of winding used in the motor.
6. Choose OK.

Optimization Configuration

If you selected Optimize as the control program, choose Configure to define the optimization settings.

To define the motor settings:
1. After selecting Optimize as the type of control program, choose Configure. The Optimization Program window appears.
2. Select the Objective from the pull-down menu. This allows you to locate the root, minimum, or maximum value for the optimization.
3. Select the Input Variable from the pull-down menu.
4. Select the Output Variable.
5. Enter the Minimum and Maximum values for the optimization range.
6. Enter the Target Value for the solution.
7. Enter the acceptable Percent Error for the convergence of the solution.
8. Choose OK. The optimization settings are now defined.
Once you have specified the solution options, choose Solve (or Solve/Nominal Problem, if you have a license for the Parametrics or Transient modules) from the Maxwell 2D Executive Commands menu. The system then starts to generate a solution for your problem.

This chapter describes the general solution process. It also describes how to:

• Monitor the solution to verify that it is converging.
• View the matrices, forces, torques, and other solutions that are computed.
• Monitor the computing resources that were used in the solution process.
• View the geometric model.
• If you have a parametrics license, view the parametric solutions.

Use the Variables, Convergence, Solutions, Profile, and Model buttons (which appear at the top of the Executive Commands window) to perform these tasks.

Note: Forces and torques are computed after each adaptive pass of the general field solution; inductance, impedance, conductance and capacitance are computed using the final mesh generated during the field solution. Therefore, adequate refinement of the mesh during the field solution increases the accuracy of executive parameter solutions.
Generating Solutions

Once you have setup the solutions, you can generate them.

> To compute a solution, do one of the following:
  - If you do not have the license for the parametrics module, choose Solve.
  - If you have the license for the parametrics module, choose Solve/Nominal Problem.

Maxwell 2D then begins the process of adaptively refining the finite element mesh and computing the requested field and parameter solutions. The exact solution process depends on the options you selected under Setup Solution and Setup Executive Parameters. (For instance, you can perform a single, non-adaptive field or parameter solution.) The general adaptive solution process is described below.

- If no finite element mesh exists, the simulator creates one. Initially, this mesh is very coarse. When an adaptive solution is performed, the mesh is refined (broken down into smaller triangles) in areas of high error.
- The simulator then solves for the electric or magnetic field in the model, using the field solver you selected. During the solution, it computes the field at each node (triangle vertex) in the mesh. Field values inside the triangles are interpolated from these nodal values, allowing you to treat the solution as being continuous over the problem region.
- If you requested a force, torque, current flow or flux linkage solution under Setup Executive Parameters, the simulator computes these quantities next. Additionally, if an eddy current, eddy axial, DC conduction, or AC conduction field solution is being performed, the simulator computes the total power loss. This lets you see whether these quantities are converging along with the field solution.
- After the field and parameter solutions are complete, the simulator performs an error analysis in each triangle in the mesh. The triangles with the highest energy error are broken down into smaller triangles — producing a more accurate solution in these areas.

The simulator repeats this process until the convergence criteria you specified under Setup Solution Options are satisfied — either by completing the specified number of adaptive passes (solve — refine cycles) or by meeting the error energy stopping criterion. If you requested a matrix solution under Setup Executive Parameters, the simulator computes it when the adaptive solution process is finished.
Completing the Solution Process

After all nominal and parametric solutions have been computed, a window appears, telling you that the solution process is complete.

Choose OK to close the window and return to the Executive Commands menu.

Monitoring the Solution Process

As the field solutions and parameter solutions are being generated, the following progress bars appear in the Solution Monitoring area:

- **Making Initial Mesh** indicates that the initial finite element mesh is being created.
- **Solving Fields** indicates that the simulator is generating a field solution. A progress bar appears for each adaptive pass or non-adaptive solution. Messages describing the steps the field solver performs during the solution appear beneath the progress bar.
- **Refining Mesh** indicates that the simulator is adaptively refining the finite element mesh based on the error residual of the last field solution. This progress bar appears once for each adaptive pass.
- **Solving For:**
  - **Matrix/** indicates that the adaptive analysis is complete and that the simulator is now computing one of the parameters requested under Setup Executive Parameters. Messages describing the steps the field solver performs during the parameter solution appear beneath the progress bar. The solver selected by the user and the model determine which parameters are available for solution.
  - **Force/Torque/**
  - **Flux Linkage/**
  - **Current Flow/**
  - **Number Registers/**
  - **Transient**

To view information about a solution as it's being computed, do the following:

- To monitor information such as the total energy, the percent error energy, and so forth, choose Convergence from the top of the Executive Commands window after the first adaptive solution is complete.
- To monitor information such as CPU time and memory usage, choose Profile from the top of the Executive Commands window after the first adaptive solution is complete.

When the solution process is complete, a message appears. Press Enter or choose OK to close the message window.
Aborting the Solution Process

Choose **Abort** to abort the solution process. The **Abort** button appears in the **Solution Monitoring** area of the screen once the solution process has begun.

**Note:** When you choose **Abort**, Maxwell 2D stops the solution process and deletes all incomplete intermediary solution data. Any intermediary data that was already complete — such as refined meshes, previous adaptive field solutions and previous parameter solutions — are saved. For example, if you abort the solution between the third and fourth adaptive passes, the solutions from the third pass are still available.

Stopping and Restarting the Solution Process

*EMpulse* - **Transient Solver only**.

You can stop the transient solution at any time by choosing **Stop**.

When you choose **Stop**, a window appears informing you that the solver will stop generating the solution once it has completed the current pass.

Once stopped, the transient solution can be restarted, resuming the solution generation from its last completed pass, and continuing to refine the solution until the specified error has been achieved.

Refreshing the Plot

*EMpulse* - **Transient Solver only**.

Choose **Refresh** during the solution process to refresh any visible plots from the **Solutions** menu. The plots are redrawn to their most recent values.

Viewing the Geometric Model

After displaying solution, convergence, or profile information, choose **Model** from the top of the Executive Commands window to view the geometric model again.

Use the **Zoom In**, **Zoom Out**, **Fit All**, **Wire Frame**, **Fill Solids**, and **Fit Drawing** buttons to change your view of the model.
Viewing Executive Parameter Solutions

After the solution process is done, choose Solutions from the Executive Commands window to view the final matrix, force, torque, flux linkage, or current flow results. The following commands appear in the Solutions menu:

- Matrix
- Force/Torque
- Flux Lines
- Flux Linkage
- Current Flow
- Core Loss (Eddy Current only).
- Number Registers
- Transient Data (EMpulse only).

Choose the parameter to view the solution for. Depending on which field solver you selected for the model, some of these parameters may not appear. Those for which no solutions have been computed are “greyed out” and cannot be accessed.

Note: The Solution menu does not allow you to view or analyze the field solution. Choose Post Process from the Executive Commands menu to analyze solution data.
Solutions/Matrix

If a matrix solution was computed, choose **Solutions/Matrix** to display the results. For example, the following capacitance matrix was computed for a three-conductor microstrip line:

```
Maxwell Capacitance Matrix (Distributed F/m)

object1  object2
object1  5.8845e-09  -936.1
object2  -936.1     5.2935e+15
```

Matrix Norm: 5.2935E+15
Distributed Maxwell
Operations
Viewing a Matrix

The name of each conductor in the matrix is displayed in the appropriate row and column. The units for each type of matrix are:

- Farads for capacitance.
- Henries for inductance.
- Siemens/meter for conductance.
- Impedance is given in the form $Z=(R,L)$, where $R$ is in ohms and $L$ is in henries. All impedances are complex numbers in the form $Z=R+j\omega L$, where $\omega$ is $2\pi$ times the frequency of the AC voltages and currents during the solution.
- Admittance is given in the form $Y=(G,C)$, where $G$ is in mhos per meter and $C$ is in farads per meter. All admittances are complex numbers in the form $Y=G+j\omega C$, where $\omega$ is $2\pi$ times the frequency of the AC voltages during the solution.

If more values than can fit on the screen are available, use the scroll bars on the bottom and right side of the display area to view the rest of the matrix. For more information on capacitance, inductance, impedance, conductance, and admittance matrices, click here.

Distributed Maxwell

Initially, distributed matrix values — the per meter capacitance, inductance, impedance, admittance or conductance values are displayed. The Distributed label appears after the matrix name at the top of the display area.

Lumped Maxwell

Choose Lumped to display lumped matrix values — that is, the total capacitance, inductance, admittance or conductance for the actual length of the structure. This enables you to determine the net coupling.

To display the lumped matrix values:
2. Enter the length of the structure. For instance, if a device is 2 mm in length, enter .002. All matrix values will then be multiplied by .002 meters. The Lumped label appears after the matrix name at the top of the display area when a lumped matrix is being displayed. The length of the structure is also indicated.
Maxwell 2D — Solve

Distributed SPICE
Select this to view the distributed SPICE matrix.

> To view the matrix:
  • Choose Distributed SPICE from the pull-down menu

The Distributed SPICE Matrix appears.

Lumped SPICE
Select this option to view the lumped SPICE matrix.

> To view the matrix:
  • Choose Lumped SPICE from the pull-down menu

The Lumped SPICE matrix appears.
Coupling Coefficient

Choose this to display normalized matrix values. This option and the Matrix Norm display are useful when analyzing the results of a capacitance, inductance, impedance, conductance or admittance matrix computation. The coupling coefficient indicates the degree of coupling between conductors. Coupling is a measure of the interaction between the electric or magnetic fields of the conductors in the model. The type of coupling (inductive or capacitive) depends upon the model and is indicated in parentheses after Coupling Coefficient.

The values in a coupling coefficient matrix are calculated as follows:

\[ M_k = \frac{|m_{ij}|}{\sqrt{m_{ii}m_{jj}}} \]

where:

- \( k \) is the coupling coefficient for each term in matrix \( M \).
- \( M \) is the capacitance, inductance, impedance, conductance or admittance matrix.
- \( m_{ij} \) is an individual value in the matrix.

Coupling coefficient values are always between zero and one.

For instance, in the following capacitive coupling coefficient matrix, the diagonal — representing the structure’s self-capacitance — is equal to one. The coupling between conductors \( s_2 \) and \( s_3 \) \((M_{k23})\), 0.15751, is about 20 times that of conductors \( s_1 \) and \( s_3 \) \((M_{k13})\), 0.00746.

<table>
<thead>
<tr>
<th></th>
<th>ground</th>
<th>s1</th>
<th>s2</th>
<th>s3</th>
</tr>
</thead>
<tbody>
<tr>
<td>ground</td>
<td>1.0000</td>
<td>0.3406</td>
<td>0.3523</td>
<td>0.34125</td>
</tr>
<tr>
<td>s1</td>
<td>0.3406</td>
<td>1.0000</td>
<td>0.15855</td>
<td>0.00746</td>
</tr>
<tr>
<td>s2</td>
<td>0.3523</td>
<td>0.15855</td>
<td>1.0000</td>
<td>0.15751</td>
</tr>
<tr>
<td>s3</td>
<td>0.34125</td>
<td>0.00746</td>
<td>0.15751</td>
<td>1.0000</td>
</tr>
</tbody>
</table>

Generally, it is easier to evaluate matrices using the result of the Matrix Norm calculation, which is a single “global” value for the matrix.
Operations

The following operations can be used to export or modify the solution.

Export

Choose Operations/Export to export the solution as a text file.

To export the solution:
2. Enter the name of the file to export.
3. Select In SI units to export the file in SI units.
4. Choose OK.

The file is saved and exported.

Set Units

Select Operations/Set Units to specify the units in the solution.

To specify the units:
1. Choose Operations/Set Units. The Select Units Preferences window appears.
2. Select the Quantity you wish to modify.
3. Select the Conversion of the new units.
4. Choose OK.

The units have been converted to the newly specified values.
Matrix Norm

The **Matrix Norm** value allows you to quickly estimate how much coupling exists between the conductors in the model. It gives you a single, or “average,” value to use in comparing the coupling coefficient matrices of several versions of the model.

Basically, the matrix norm is equivalent to the “length” of the matrix, and it is calculated in the same way as the length of a vector. It is given by the following relationship:

\[
Norm = \sqrt{\sum_{i=1}^{n} \sum_{j=1}^{n} |m_{ij}|^2}
\]

where:

- \( i \) is the column in the matrix.
- \( j \) is the row in the matrix.
- \( n \) is the size of the matrix.
- \( m_{ij} \) is the matrix entry.

Usually, a smaller **Matrix Norm** means less coupling. If the matrix norms for several versions of the model are virtually identical, or if you want to view conductor-specific data, use the **Coupling Coefficient** command.
If a force solution or torque solution was computed, choose **Solutions/Force/Torque** to display the results. Information such as that shown below appears:

**Note:**
- For cartesian models:
  - Force is given in newtons per meter depth.
  - Torque is given in newton-meters per meter depth or in newtons.
- For axisymmetric models, force is given in newtons. Torque is not available for axisymmetric models.
Force

The following information is displayed if a force solution has been computed:

**Force Setup Objects** Lists all the objects that you selected for the force computation, using the **Setup Executive Parameters/Force** command. If more objects than can fit in the list box have been selected, use the scroll bar to view the rest of the objects.

**Force Value** The results given under this heading represent the net force acting on all the objects listed under **Force Setup Objects**. All force values are given in newtons.

**F(x)** The component of force in the x direction.

**F(y)** The component of force in the y direction.

**Mag(F)** The magnitude of the force acting on the objects. This value is also displayed when you choose **Convergence**.

**Angle(F)** The angle (in degrees) at which the force is acting.

Torque

The following information is displayed if a torque solution has been computed:

**Torque Setup Objects** Lists all the objects that you selected for the torque computation using the **Setup Executive Parameters/Torque** command. If more objects than can fit in the list box have been selected, use the scroll bar to view the rest of the objects.

**Anchor Point** Displays the coordinates of the anchor point you selected for the torque computation. The torque on the objects is computed about this point.

**Torque Value** The net torque (in newton-meters) acting on the objects. This value is also displayed when you choose **Convergence**.
If a flux line solution was computed, choose **Solutions/Flux Lines** to display the results. Use this command to view the flux linkage across the individual lines you specified; the net flux linkage across all lines is displayed when you choose **Convergence**. Information such as that shown below appears:

![Flux Lines Solution Table]

The following information is listed for each flux linkage line that you drew using the **Setup Executive Parameters/Flux Lines** command:

- **Line Name**: The name of the flux linkage line.
- **First Point**: The coordinates of the first point in the line.
- **Second Point**: The coordinates of the second point in the line.
- **Flux Linkage**: The results of the flux linkage solution.

- For cartesian (XY) models, the flux that crosses the line per meter of depth in the z direction is displayed.
- For axisymmetric (RZ) models, the total flux that crosses the surface made by rotating the line 360° about the z-axis is displayed.
- Magnetostatic flux linkage is given in webers.
- Electrostatic flux linkage is given in coulombs per square meter.
If more values than can fit on the screen are available, use the scroll bars on the right side of the display area to view the rest of the solution data.
Solutions/Flux Linkage

If a flux linkage solution was computed, choose **Solutions/Flux Linkage** to display the results. When you choose **Convergence**, information such as the following appears:

![Image of Flux Linkage Solution]

The following information is listed for each flux linkage line that you drew using the **Setup Executive Parameters/Matrix/Flux** command:

- **Object Name**: The name of the object on which the flux was computed.
- **Calculated Flux**: The calculated flux on the object.
- **Turns**: The number of flux turns in the object.
- **Total Flux**: The total flux on the object at a depth of one default unit.

- For cartesian (XY) models, the flux that crosses the line per meter of depth in the z direction is displayed.
- For axisymmetric (RZ) models, the total flux that crosses the surface made by rotating the line 360° about the z-axis is displayed.
Modifying Turns and Depth

Once a flux linkage solution has been generated, you can modify the number of turns and model depth to obtain a new value for the total flux of the object.

> To modify the number of turns for an object:
1. Select the row for the object to modify. The existing number of turns appears in the **Number of Turns** field.
2. Enter the new **Number of Turns** for the object.
3. Choose **Set**. The value is updated in the solution window.

When the new number of turns and the model depth are updated, the **Total Flux** is recalculated and appears in the solutions window.

Operations

Choose the **Operations** commands to define the units of the solution

Export

Choose **Export** from the **Operations** pull-down menu to export the solutions to a set of text files.

> To export the solution:
1. Choose **Export**. The **Save As** window appears.
2. Enter the file name in the **Save flux linkage data in file** field and choose **OK**.

The data file is exported to the defined file.

Set Units

Choose **Set Units** from the **Operations** pull-down menu to define the units of the solution.

> To define the units:
1. Choose **Operations/Set Units**. The **Set Unit Preferences** window appears, listing the current **Quantity** you are modifying.
2. Select the new units from the list.
3. Choose **OK**.

The new units are accepted, and the values in the window are recalculated for the new setting.
Solutions/Current Flow

If a current flow solution was computed, choose **Solutions/Current Flow** to display the results. Use this command to view the current flow across the individual lines you specified; the total current flowing across all lines is displayed when you choose **Convergence**. Information such as that shown below is displayed:

The following information is listed for each current flow line that you drew using the **Setup Executive Parameters/Current Flow** command:

- **Line Name**: The name of the current flow line.
- **First Point**: The coordinates of the first point in the line.
- **Second Point**: The coordinates of the second point in the line.
- **Current Flow**: The results of the current flow solution.

- For cartesian (XY) models, the current flow across the line in amperes per meter of depth in the z-direction is displayed.
- For axisymmetric (RZ) models, the total current in amperes flowing through the surface made by rotating the line 360° about the z-axis is displayed.
If more values than can fit on the screen are available, use the scroll bars on the right side of the display area to view the rest of the solution data.

**Solutions/Core Loss**

Choose **Solutions/Core Loss** to view the results of the core loss calculations you defined for your eddy current problem.

To view the core loss solutions:
- Choose **Solutions/Core Loss**. The core loss results appear:

  ![Core Loss Solution](image)

**Solutions/Number Registers**

Choose **Solutions/Number Registers** to view the results of any Post Processor macro solutions that wrote number registers to a file. Information like the following appears:

<table>
<thead>
<tr>
<th>Filename</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>ampfile</td>
<td>-0.983014</td>
</tr>
<tr>
<td>fluxfile</td>
<td>-1.77616E-07</td>
</tr>
</tbody>
</table>

To view the results of a post-processing macro using this command, you must have first saved the registers containing the results.

To save the registers, do the following while defining the macro:
1. Choose **Post Process/Calc/Number**.
2. Choose **Roll** to move the first register that is to be saved to the top of the stack.
3. Choose **Write**.
4. At the **Filename** prompt, enter a name to use for saving the register, then choose **Execute**. The name appears as the filename when the register values are viewed.
5. Repeat the process for each register that is to be saved.
Solutions/Transient Data

*EMpulse only.*

If you have created and solved a Transient motion problem, choose **Solutions/Transient Data** to view the plots of the transient solutions. A window similar to the following one appears:

> To observe the plots:
> 1. Select the plot to view from the **Plots** list. The selected plot appears.
> 2. Use the **View** commands below the plot to change the view of the plotted data.
Show Coords

Choose this command to display the coordinates of a selected point.

> To display plot coordinates:
1. Choose Show Coords. The cursor changes to crosshairs.
2. Select a point in the plot window. The Coordinates window appears, listing the coordinates of the selected point.
3. Continue selecting points until you have seen the coordinates of all the desired points.
4. Click the right mouse button.

The crosshairs return the default cursor.

Settings

Choose the Settings commands to define the format of the axes and graphs of the plots. Choose from the following commands:

Format Axes
Specifies axis scales, labels, plot headings, and minimum and maximum signal values to be plotted along with axis tick mark and grid settings. This command functions identically to the Parametrics Post Processor Plot/Format/Axes command.

Format Graphs
Specifies the color, line thickness, and line style of a previously plotted line. Also determines the type of markers displayed at solution data points, and whether the graph is visible on the plot. This command functions identically to the Parametrics Post Processor Plot/Format/Graphs command.
Viewing Convergence Data

Choose Convergence from the top of the Executive Commands window to monitor how the solution is converging and view information about it — or, if the solution is complete, to verify that it did converge. Information such as that shown below is then displayed. If more values than can fit on the screen are available, use the scroll bars on the bottom and right side of the display area to view the rest of the convergence data.

Number of Passes

The fields below indicate the solution status in terms of how many adaptive passes (solve — error analysis — refine cycles) are completed.

- **Completed**: Displays the number of adaptive passes that have been completed.
- **Remaining**: Displays the number of adaptive passes that have yet to be completed.

The total Number of Passes is set using the Setup Solution Options command.
Convergence Criteria

These fields show how close the current solution is to meeting the convergence criteria entered under **Setup Solution**.

- **Target Error**: Displays the target percent energy error — the percent energy error to achieve for the solution, entered with the **Setup Solution** command.
- **Energy Error**: Displays the percent energy error of the last completed solution.
- **Delta Energy**: Displays the percent change in the total energy between the last two adaptive passes.

When the **Energy Error** and the **Delta Energy** fall below the **Target Error** value, the Maxwell 2D stops the adaptive solution process — even if the desired number of adaptive passes has not been reached.

Frequency

If an AC field quantity is being computed, this field displays the frequency at which currents, voltages, and fields oscillate.
Convergence Data

The system displays the following information for each solution:

- **Pass**: The adaptive pass number. Only displayed for adaptive field solutions.
- **Triangles**: The number of triangular elements in the finite element mesh that were used during the solution. Note that this may be different than the total number of triangles. For instance, in an electrostatic, magnetostatic or eddy current solution, no fields are computed inside perfect conductors — thus, these triangles are not included in the count displayed in this field.
- **Torque**: The net torque (in newton-meters) on the objects selected using Setup Executive Parameters/Torque, computed about the specified anchor point. Appears only if torque is computed during the solution, allowing you to see if it is converging.
- **Flux**: The total flux linkage computed across the line(s) you specified using Setup Executive Parameters/Flux Linkage. Flux linkage values are given in webers (magnetostatic) or coulombs per square meter (electrostatic). Appears only if flux linkage is computed during the solution, allowing you to see if it is converging.
- **Current**: The total current flow (in amperes) computed across the line(s) you specified using Setup Executive Parameters/Current Flow. Appears only if current flow is computed during the solution.
- **Total Power Loss**: The net power loss (in watts) for the model. Power loss is automatically computed during DC conduction, AC conduction, eddy current, and eddy axial field solutions.

After a solution is complete, use the **Energy Error** to determine if additional adaptive passes are needed to increase the accuracy of the solution. If the **Energy Error** is still declining from pass to pass, additional adaptive passes may increase the accuracy of the field solution. If the **Energy Error** has leveled off, additional adaptive passes may not be useful.

**Note**: Note that the final force, torque, flux linkage and current flow solutions can also be viewed using the Solutions command.
The Convergence Display button appears on the bottom of the Convergence menu. Use it to display convergence information graphically. Click the left mouse button on it to access a menu with the following commands:

- **Table**: Displays convergence information in a table (the default).
- **Plot Triangles**: Plots the number of triangles in the mesh for each field solution (adaptive or non-adaptive).
- **Plot Total Energy**: Plots the total energy for each field solution (adaptive or non-adaptive).
- **Plot Percent Error Energy**: Plots the percent energy error for each field solution (adaptive or non-adaptive).
- **Plot Force**: If force is computed, plots the total force for each adaptive solution.
- **Plot Torque**: If torque is computed, plots total torque for each adaptive solution.
- **Plot Flux**: If flux linkage is computed, plots the total flux linkage for each adaptive solution.
- **Plot Current**: If current flow is computed, plots the total current flow for each adaptive solution.
- **Plot Total Power Loss**: If an AC conduction, DC conduction, eddy current or eddy axial field simulation is being performed, plots the total power loss for each field solution (adaptive or non-adaptive).

If you display a convergence plot while an adaptive solution is being computed, the plot automatically updates to show the new data points from each adaptive pass.

- Choose the desired information to be plotted. A plot like the following one is then drawn. In this example, the energy error for each adaptive solution was plotted, using the Convergence Display/Plot Percent Error Energy command. As can be seen below, the plot approaches zero as the number of adaptive passes rises — indicating that the solution’s accuracy is increasing. The dotted line on the plot shows the target energy error (entered via the Setup Solution command).
- Use the Zoom In, Zoom Out and Fit All buttons to change your view of the plot.
- After plotting convergence data, select Convergence Display/Table to return to a tabular display of convergence information.
Topics:
Solve
Generating Solutions
Completing the Solution Process
Monitoring the Solution Process
Aborting the Solution Process
Stopping and Restarting the Solution Process
Refreshing the Plot
Viewing the Geometric Model
Viewing Executive Parameter Solutions
Viewing Convergence Data
   Number of Passes
   Convergence Criteria
   Frequency
   Convergence Data
Convergence Display
Viewing Profile Data
**Viewing Profile Data**

Choose **Profile** from the top of the Executive Commands window to see the computing resources used in the solution process. Information such as that shown below appears:

The profile data is a log of the tasks performed by the Maxwell 2D during the field and parameter solutions, how long they took, and how much RAM/disk memory was required. For example, the profile of an adaptive pass of a nonlinear magnetostatic solution in which both fields and parameters were computed might look like this:

<table>
<thead>
<tr>
<th>Command/Info</th>
<th>Real Time</th>
<th>CPU time</th>
<th>Mem size</th>
<th>Number of elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pass 4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>mesh2d_adap</td>
<td>00.03.09</td>
<td>00.02.16</td>
<td>1801K</td>
<td>6320 triangles</td>
</tr>
<tr>
<td>nl2d_solve</td>
<td>00.09.37</td>
<td>00.09.02</td>
<td>8362K</td>
<td>6272 triangles</td>
</tr>
<tr>
<td>nl2d_torque</td>
<td>00.00.09</td>
<td>00.00.08</td>
<td>8362K</td>
<td>6272 triangles</td>
</tr>
<tr>
<td>nl2d_force</td>
<td>00.00.10</td>
<td>00.00.08</td>
<td>8362K</td>
<td>6272 triangles</td>
</tr>
</tbody>
</table>

Notice that all computations performed during the adaptive pass — including the pass-by-pass force and torque solutions — are listed.
Maxwell 2D — Solve

Command/Info

Lists the software module that performs a task during the solution process, and the type of task that was performed. For example, mesh2d_adap is the software module that adaptively refines the finite element mesh. nl2d is the nonlinear magnetostatic field simulator. The listings nl2d_solve, nl2d_torque, and nl2d_force identify the resources used during the field, torque and force solutions, respectively.

Real Time and CPU Time

These two fields indicate how much real time — that is, the amount of time that a clock would indicate — and how much CPU time was required to perform the task.

Memory Size

This field indicates how much RAM/virtual memory was required to complete the task. This memory is freed for other uses after the task is complete.

Number of Elements

This field indicates how many of the triangular elements in the mesh were used during the solution. The resources needed to compute the solution grow with the number of elements because the electromagnetic equation being solved has to be solved in each element separately.
Solving a Problem with Variables

Maxwell 2D’s Parametric Analysis module has two commands for launching the solution process. Choose them from the Solve menu.

Solve/Nominal Problem

Choose Solve/Nominal Problem to generate a nominal solution for the model on which the parametric problem is based. This command operates the same way as the Solve command for the non-parametric version of the software.
Solve/Variables

Choose **Solve/Variables** to generate a solution for the parametric sweep you defined using the **Setup Solution/Variables** command.

**Solution Process**

To generate a solution for the parametric sweep, Maxwell 2D performs the steps listed below. This process is summarized below:

1. The variables you selected for the sweep are set to the values specified in the first unsolved parametric setup listed on the spreadsheet.
Maxwell 2D — Solving a Problem with Variables

2. Maxwell 2D generates an electric or magnetic field solution for the setup. The solution parameters you specified using the Setup Solution/Options command control how the field solution is generated — for example, whether an adaptive solution is performed. If you changed the value of a geometric constraint, the system generates a finite element mesh based on the geometry’s new dimensions prior to computing fields. Otherwise, the mesh used for the solution is the initial mesh.

Note: The field solution is saved only if you specified Y under Save Fields when setting up the parametric sweep.

3. The simulator then generates solutions for any executive parameters — force, torque, capacitance, inductance, flux linkage, and so forth — that you specified using the Setup Executive Parameters commands.

4. After completing the executive parameter solutions, the system executes any post-processing macros that you selected via the Setup Executive Parameters/Post Processor Macros command.

The system repeats this process until solutions are generated for all parametric setups. Progress bars appear to show the status of each step. When the parametric sweep is completed, a message appears, informing you that the solution is complete.

Choose OK to return to the Executive Commands menu.

Note: If the setup fails to converge before the Number of requested passes criterion specified in Setup Solution Parameters is reached, the solution process for this setup stops and the next one begins. A setup that has been stopped without converging will show a Y(yes) in both the Solved and Solve columns of the variables table.
Aborting a Solution

Choose Abort to stop a parametric sweep. Be aware of the following:

- Parametric setup solutions that were computed prior to the one that was aborted are still available.
- The solutions that are available for the current setup depend on when you aborted the parametric sweep. For instance, if you stopped the solution while a post-processing macro was executing, the field and parameter solutions computed for that setup are still available.

If you select Solve/Variables again after aborting a solution, Maxwell 2D resumes the parametric sweep. The system computes fields and parameters and runs any post-processor macros for the parametric setup whose solution was aborted, regardless of whether they were computed earlier.

Errors in Parametric Solutions

If a problem occurs during a parametric solution — such as the creation of overlapping or self-intersecting objects when a geometric constraint is set to a new value — the parametric sweep is aborted and an error message appears. Before continuing the parametric solution, change the value of the variable that caused the problem. Solutions for parametric setups that were computed before the error condition occurred are still available.

To check for errors with geometric constraints, choose Variables/Animate while setting up the solution. If a constraint error exists, an error message appears while you are executing this command.
Displaying Solution Information

Use the following menus to view information about parametric and nominal solutions.

**Variables**

Choose **Variables** to do the following:

- View parametric setups during the solution process.
- View the final error energy, solution energy, and number of triangles for each parametric solution.
- Display the results of any executive parameter solutions (force, capacitance, current flow, and so forth) or post-processing macros that were computed during the parametric sweep.
- Display solution, profile, and convergence information for individual parametric setups.

The following figure shows the solution information for the setup of a sweep on the variable **Source** (a variable specifying the amount of current flowing through the coil in a plate). Note that you cannot view field solutions for completed setups while a parametric sweep is in progress:
Model

Use the Model command to view the geometric model. In parametric problems, this command is generally used to display the geometries that result when a geometric variable is constrained to different values. You can change your view of the geometry if you need to.

Nominal Geometry

To view the geometric model for the nominal problem, do one of the following:
1. If the Executive Commands window is displayed, choose Model.
2. If the parametric spreadsheet is displayed,
   1. Choose Return to Nominal Problem.
   2. Choose Model.

Parametric Geometry

To view the geometric model for a parametric setup, do the following:
1. Choose Variables to display the parametric spreadsheet.
2. Select the setup whose geometry you wish to view.
3. Choose Model. The geometric model used during that parametric solution appears.
4. To return to the Executive Commands window, choose Return to Nominal Problem.
Solutions

Use the Solutions commands to view the following for the nominal problem or the selected parametric solution:

- All components of force and torque solutions.
- A complete inductance, capacitance, impedance, admittance, or conductance matrix.
- Current flow and flux linkage solutions, including the lines over which the solutions were computed.
- Number registers containing the output of any post-processor macros.

Nominal Solutions

> To view executive parameter solutions for the nominal model,

- If the Executive Commands window is displayed, choose one of the following Solutions commands:
  - Matrix: Displays any inductance, capacitance, impedance, admittance, or conductance matrix solutions.
  - Force-Torque: Displays all components of any force or torque solutions.
  - Current Flow: Displays any current flow solutions.
  - Flux Linkage: Displays any flux linkage solutions.
  - Core Loss: Displays any core loss solutions.
  - Number Register: Displays the contents of the number register, if a post-processing macro was run.
  - Modal Characteristic Impedance: Displays the modal characteristic impedance matrix.

- If the parametric spreadsheet is displayed, you can:
  - Choose Return to Nominal Problem.
  - Choose one of the Solutions commands listed above.
Parametric Solutions

To view executive parameter solutions for a parametric setup:
1. Choose **Variables** to display the parametric spreadsheet.
2. Select the desired parametric setup.
3. Choose one of the **Solutions** commands to view executive parameter solutions for the selected setup.
4. To return to the Executive Commands window, choose **Return to Nominal Problem**.
Convergence

Use the Convergence command to view the following:

- The number of completed adaptive passes (if an adaptive field solution was performed).
- The stopping criteria entered via the Setup Solution/Options command.
- The energy, error energy, and number of triangles for each field solution (adaptive or non-adaptive).
- The results of any force, torque, flux linkage, or current flow solutions (adaptive or non-adaptive).

Nominal Convergence

To view convergence information for the nominal model:

- If the Executive Commands window is displayed, choose Convergence.
- If the parametric spreadsheet is displayed:
  1. Choose Return to Nominal Problem.
  2. Choose Convergence.

Parametric Convergence

To view convergence information for a parametric setup:

1. Choose Variables to display the parametric spreadsheet.
2. Select the desired setup.
3. Choose Convergence. Convergence information for the field solution associated with that setup appears.
4. To return to the Executive Commands window, choose Return to Nominal Problem.
Two different types of solution statistics can be viewed using the **Profile** menu.

### General Profile Statistics

To view the computing resources used during all solutions (whether parametric or nominal):

- If the Executive Commands window is displayed, choose **Profile**.
- If the parametric spreadsheet is displayed:
  1. Choose **Return to Nominal Problem**.
  2. Choose **Profile**.

Information such as the following is displayed for parametric sweeps:

<table>
<thead>
<tr>
<th>Command/Info</th>
<th>Real Time</th>
<th>CPU Time</th>
<th>Mem Size</th>
<th>Num Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parametric solution</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>setup1</td>
<td>00:00:08</td>
<td>00:00:03</td>
<td>839K</td>
<td>146 triangles</td>
</tr>
<tr>
<td>setup2</td>
<td>00:00:08</td>
<td>00:00:03</td>
<td>839K</td>
<td>146 triangles</td>
</tr>
<tr>
<td>setup3</td>
<td>00:00:11</td>
<td>00:00:03</td>
<td>839K</td>
<td>168 triangles</td>
</tr>
<tr>
<td>setup4</td>
<td>00:00:35</td>
<td>00:00:03</td>
<td>839K</td>
<td>168 triangles</td>
</tr>
<tr>
<td>Total</td>
<td>00:00:35</td>
<td>00:00:12</td>
<td>839K</td>
<td>168 triangles</td>
</tr>
</tbody>
</table>

The total memory and CPU time for each variable setup that was solved during the solution process is listed. Totals are given for the entire parametric sweep. The number of triangles represents the highest number of triangles in the mesh for that setup's field solution and for the entire sweep.

**Note:**

In the Microsoft Windows version of the software, the CPU time is equal to the real time and memory usage information is not available.
Parametric Profile Statistics

To view the computing resources used during a single parametric solution:
1. Choose Variables to display the parametric spreadsheet.
2. Select the desired parametric setup.
3. Choose Profile. Information such as that shown below is displayed:

4. To return to the Executive Commands window, choose Return to Nominal Problem.
Post Process

Choose the Post Process commands from the Executive Commands menu to access the post processors. The post processors allow you to:

- Plot and analyze nominal field solutions.
- Plot parametric and variables field solutions.
- (EMpulse only) Plot the transient, time-stepping field solutions.

The post processors only become active after their respective solutions have been generated for the problem.

Post Processor

The Post Processor has the following commands that allow you to perform the following functions:

- **Nominal Problem**
  - Accesses the 2D Post Processor.

- **Fields**
  - *EMpulse only*: Solves the nominal field solutions for the transient solution.

- **Variables**
  - Accesses the parametrics post processor and displays the parametric table.

- **Transient Data**
  - *EMpulse only*: Accesses PlotData for plotting transient solutions.

When you choose one of the Post Process commands, its respective post processor appears.

Note: If you have only the basic license installed for Maxwell 2D, only one Post Process command appears, allowing you to analyze the nominal solutions only.

While you are accessing a post processor, the Maxwell 2D Executive Commands menu remains active. This lets you inspect material properties, boundary conditions, convergence information, and so forth while viewing the field solution. If you exit Maxwell 2D while the Post Processor is being accessed, the software gives you the option of continuing to view the field solution even after you’ve exited the main program, or of exiting from both software modules.
Post Process/Nominal Problem

Choose this command to access the 2D Post Processor.

The 2D Post Processor for the nominal and field solutions has the following menus that allow you to view and analyze your project and its solution:

- **File**: Opens and saves post processing macros. Also allows you to load a solution into the 2D post processor.
- **Global**: Sets and displays the global display parameters, such as color, units, and grid spacing.
- **Window**: Defines the window settings.
- **Show**: Displays information about the finite element mesh.
- **Post**: Plots common field quantities on lines and planes. Also defines field registers to use with the line calculator.
- **Calc**: Computes derived quantities from the general field solutions.
- **Help**: Accesses the online documentation.

Post Processor Units

The units of the model are specified in the 2D Modeler and cannot be modified in the 2D Post Processor.

The unit of length has no effect on the units used for electromagnetic quantities that are referenced in the Post Processor. Electromagnetic quantities are always expressed in SI (MKS) units. For instance, even though the geometry of a problem is entered in millimeters, energy density is still displayed in joules/meter$^2$, not joules/mm$^2$. 
Post Process/Fields

**EMpulse only.**

Choose this command to access the fields Post Processor. The Post-Process Saved Fields window appears, allowing you to select the field to analyze in the Post Processor:

- To select the fields to analyze:
  1. Select the time at which you will analyze the field from the Time list.
  2. Choose Post Process to access the 2D Post Processor and analyze the field at the selected time, or Cancel to cancel the analysis and return to the Executive Commands window.

**Deleting Time Steps**

You can also delete unwanted time steps that are not needed in the analysis.

- To delete any time steps:
  1. Select the time step to delete from the Time list.
  2. Choose Delete.

The time step is deleted. Deleted time steps cannot be recovered without regenerating a transient solution.
File Menu

Use the Post Processor’s **File** commands to:

- Load solution files into the Post Processor.
- Create and run Post Processor macros, recorded sequences of commands that enable you to automate various post-processing operations.
- Exit the Post Processor.

When you choose **File** from the Post Processor menu bar, the following menu appears:

File Commands

The commands on the **File** menu are:

- **Open**: Loads a solution into the Post Processor.
- **Trans**: Lets you record, execute, and delete macros.
- **Exit**: Exits the Post Processor.
File/Open

Choose **File/Open** to open and read a solution file into the Post Processor. This command operates like the **File/Open** command in the 2D Modeler, except that solution files (*.sob) are automatically displayed.

File/Trans

In the post processor, you can use transcripts to record a sequence of commands, and then execute those commands again later as a macro. Once created, a macro can be executed at any time in the Post Processor.

Use macros to save yourself a lot of time and effort. For example, calculating a specific quantity (such as the inductance of a transmission line) can take several keystrokes and involve a number of commands. If you store that series of commands and keystrokes in a macro, you can calculate the quantity with a single command.

Use the **File/Trans** commands to create and execute macros. The following commands are available:

- **Start** Begin recording a sequence of commands.
- **Stop** Finish recording commands, and save the macro into a transcript file.
- **Execute** Execute a macro by its name.
- **Delete** Remove a macro from the **File/Trans** menu.

Previously defined macros may also appear in the menu.
File/Trans/Start

File/Trans/Stop

Use these two commands in conjunction to define a macro and save it to a transcript file. The macro may also be added to the File/Trans menu.

To define a macro:
1. Choose File/Trans/Start to begin recording.
2. Enter the name you would like to give to the macro.
3. Choose Continue.
4. Perform all of the actions that you wish to be recorded in the macro.
5. Choose File/Trans/Stop to finish recording the macro.
6. You now have the option of adding the macro to the menu.
   • If you wish to add the macro to the File/Trans menu, set the option to Yes.
   • If you do not wish to add the macro to the File/Trans menu, set the option to No (the default).
7. Choose Continue.

The macro is now complete and has been saved to a transcript file with the current project. The name of the file is the macro name you specified plus the .trn file extension. If you chose to add the macro to the menu, it will now appear on the File/Trans menu, and can be executed by choosing the name from the menu. If you did not add the name to the menu, you can still execute it using the File/Trans/Execute command.
File/Trans/Execute

Use the File/Trans/Execute command to execute a previously defined command by entering its name. If the macro’s name has been added to the menu, you may also execute it by choosing the name from the menu.

To execute a command by entering its name:
1. Choose File/Trans/Execute.
2. Enter the macro’s name.
3. Choose Continue. If the macro is not currently listed on the menu, you now have the option of adding it to the menu.
   • If you wish to add the macro to the File/Trans menu, set the option to Yes.
   • If you do not wish to add the macro to the File/Trans menu, set the option to No (the default).

The entire sequence of recorded actions is replayed exactly as it was recorded. For example, if you created a macro to display an arrow plot of the $E$ vector superimposed on a contour plot of the voltage, those commands will be executed, and the plots will be displayed.

File/Trans/Delete

Use the File/Trans/Delete command to remove a macro name from the menu. This does not delete the transcript file, and the macro may still be invoked by name using the File/Trans/Execute command.

To delete a macro’s name from the menu:
1. Choose File/Trans/Delete.
2. Enter the name of the macro you wish to remove. (If you do not wish to remove any macro from the menu, do not enter a name.)
3. Choose Continue.

The macro’s name will be removed from the menu after you exit the Post Processor.
File Menu
File Commands
File/Open
File/Trans
  File/Trans/Start
  File/Trans/Stop
  File/Trans/Execute
  File/Trans/Delete
File/Exit

**File/Exit**

Choose **File/Exit** to exit the Post Processor.

> To exit the Post Processor:
  * Choose **File/Exit**. A message appears asking you whether you would like to exit.
  * Choose **Yes** to return to the Executive Commands window.
  * Choose **No** to remain in the Post Processor.
Global Menu

Use the Global commands to do the following:

- Specify which objects are to be displayed.
- Change the colors of objects.
- Clear and refresh all viewing windows.
- Set global parameters such as the desired unit of length, the grid spacing, and the method of selecting points.

When you choose Global from the Post Processor menu bar, the following menu appears:
Global Commands

The function of each Global command is as follows:

- **Display**: Displays the objects set to Yes. Objects set to No do not appear on screen.
- **Recolor**: Allows you to change the color in which objects are displayed.
- **Refresh**: Clears all viewing windows and redraws any object that was on screen when the command was chosen.
- **Defaults**: Allows you to do the following:
  - Specify the desired unit of length.
  - Specify default names for axes, lines, planes, objects, and other entities that are assigned names.
  - Adjust the field of view by entering a convenient scale for grids and axes.
  - Define the “snap-to-point” behavior of the cursor when points are selected from the screen.
  - Enable or disable keyboard entry.

The Global commands are used in conjunction with the Window commands to adjust viewing windows and system parameters. The Global commands affect all viewing windows at once, while the Window commands affect one viewing window at a time.
Global Settings

The following settings can be applied globally to the problem.

Displaying Objects

Choose Global/Display to specify which objects to display. The objects appear in all viewing windows. It is not possible to display different sets of objects in different windows.

Use the Global/Recolor command to change the colors of displayed objects.

Mouse Behavior

Choose Global/Defaults to specify how mouse generated points are interpolated. Options are Object snap and Grid snap (both of which are selected by default). You can alternatively enter points from the keyboard by using the Keyboard entry option.

Units

Choose Global/Defaults to change the unit of length in which coordinates are displayed and entered. The new unit of length has no effect on the display size of objects. For example, if an object display is 2 millimeters wide before the unit of length is changed to mils, its display remains 2 millimeters wide afterwards. The only difference is that the dimension is now expressed as 78.44 mils.

Zoom, Fill, and Set

If the geometric model of a structure has already been loaded into the simulator, a view of that model appears in the Global/Display and Global/Recolor menus. Use the Zoom (similar to Window/Zoom), Fill (similar to Fit Drawing), and Set (similar to Window/Setup/Settings) commands to modify the view of the structure.

Executing Commands

The Global/Display and Global/Recolor commands can be executed as follows:

- Choose Execute and Return to execute the command and return to the main display.
- Choose Execute to execute the command and remain in the window of the current command. Use this option to view the results of the changes made by the command before returning to the main display.
Choose **Global/Display** to specify which objects to display on the screen. The following window appears after you choose this command:

### Object Lists

Choose the name of an object list to set all objects in that list to **Yes**. The following object lists are predefined:

- **all**: Includes all objects.
- **all-2d**: Includes 2D objects only.

### Display Object(s)

The names of all objects currently in memory appear in the list. Set the objects that you wish to display to **Yes**.

The objects that are on the screen when the command is chosen are preset to **Yes**. To set them all to **No**, set **all** in the **Objects Lists** list box to **No**.
Fill View

Set this field to Yes to make objects appear as large as possible without extending beyond the window. In the process, several grid and mouse options are automatically revised. Choose Global/Defaults to manually reset them.

Displaying Objects

1. Set the objects that you wish to display to Yes by doing one of the following:
   • Set one of the names of the object lists in the Object Lists list box to Yes.
   • Set the objects listed in the Display Object(s) list box to Yes.
2. Set Fill View to Yes or No.
3. Execute the command.

To change the color in which objects are displayed, choose Global/Recolor.

Adjusting the View

If an object that you request to display does not appear on the screen after Execute or Execute and Return is chosen, it is possible that the object is outside the field of view or that the field of view is not set wide enough to include the object. For example, if the field of view associated with a window is 0.1 mm wide, it is impossible to see objects that are 10 mm wide.

Choose Global/Display and set Fill View to Yes to automatically adjust the field of view to include all of the objects selected for display.

Similarly, if two small objects are separated by a large distance relative to their size, setting Fill View to Yes scales the field of view to include both objects. However, because of their small size relative to the required field of view, they may not be visible. Be aware of this problem when displaying small objects separated by large dimensions.
Global/Recolor

Choose **Global/Recolor** to change the color of objects. The following window appears:

---

### Recolor Object(s)

The names of all objects appear in the window. Select the objects that are to be recolored by setting them to **Yes**. After you do, the default color appears in a box next to the object name.

### New Color(s)

The palette of colors appears in the **New Color(s)** column next to the object.
Changing the Color of Selected Objects

To assign a new color to an object:

1. Select the object to which you want a new color assigned.
2. Click the left mouse button on the color box that appears next to the desired object. A color palette with 16 colors appears.
3. Click the left mouse button on one of the colors.
4. Repeat steps 1 through 3 for each object you want to recolor.
5. Execute the command.

Note: If you recolor objects while you are creating a geometric model and then save your files, objects are saved in their new colors. However, if you recolor objects while using the Post Processor, the new colors are only in effect until you quit the Post Processor.

Global/Refresh

Choose Global/Refresh if the screen gets too cluttered with temporary information that is no longer needed. The system erases all objects, lines, and other data from all viewing windows and then immediately redraws the objects and lines. Only the objects and lines that are on screen when the command is chosen are redrawn.

Temporary information includes such things as plots, shaded views of objects, and meshes that are displayed with commands in the Post Processor. To refresh one viewing window at a time, choose Window/Refresh.
Choose **Global/Defaults** to do the following:

- Specify the desired unit of length.
- Adjust the field of view by entering a convenient scale for grids and axes.
- Define the “snap-to-point” behavior of the cursor when points are selected from the screen.
- Enable or disable keyboard entry.

When you choose this command, the following window appears:
Units

The **Units** field displays the current units of the model, specified in the 2D Modeler. Initially, all length dimensions are expressed in millimeters.

**Note:**
The unit of length has no effect on the units used for electromagnetic quantities that are referenced in the Post Processor. Electromagnetic quantities are always expressed in SI (MKS) units. For instance, even though the geometry of a problem is entered in millimeters, energy density is still displayed in joules/meter$^2$, not joules/mm$^2$.

Grids

Change the fields under **Grids** to display or modify the spacing associated with grids, axes, and axis divisions. Object dimensions are not affected by these fields. The displayed values are always in the current unit of length.

This changes the field of view for all windows. Use the **Window** commands to change the field of view for one window without altering the global parameters.

**2D Grid Division**

Use the **2d grid division** field to control the distance (in the current unit of length) between two adjacent grid points in 2D viewing windows. For example, a value of **2.000e+00** indicates that the distance between two adjacent grid points represents 2 millimeters (assuming that millimeters is the current unit of length).

If you use the **Window/Zoom** command to zoom in or out on an object in a particular 2D window, the spacing of the grid in that window changes. When you zoom in, the dots on the grid get further apart. When you zoom out, they get closer together. However, the distance represented by neighboring grid points remains constant; that is, if the neighboring grid points are 2 millimeters apart before the zoom, they are also 2 millimeters apart after the zoom.

Enter a new value to change the distance represented by adjacent grid points. Although adjusting the value of the **2d grid division** field changes the apparent size of objects on screen, the dimensions associated with objects do not change. An object that is modeled as 3 inches long before the spacing was changed remains 3 inches long after the command is executed.
Mouse Grid Spacing

Use the **Mouse grid spacing** field to specify the spacing of the grid to which the cursor snaps when points are selected with the mouse in the 2D window.

The mouse grid is invisible and can be set independently of the 2D grid that is displayed on the screen. The default mouse grid setting is half of the distance between displayed 2D grid points.

Mouse

Change the fields under **Mouse** to alter the way in which points are selected with the mouse. When selecting points from the screen, you typically move the cursor to the desired point and click the left mouse button. The system snaps to the closest mouse grid point or object point and uses the coordinates of that point rather than the exact location of the cursor.

The mouse grid is invisible. It can be set independently of the grids that are displayed on screen.

Object Snap

Set the **Object snap** field to **Yes** to force the system to grab an object point whenever you click a mouse button near an object point. If the **Grid snap** mode is also in effect, the system snaps to either an object point or a grid point, depending on which is closer.

An object point is any point on the boundary of an object that is used to define the object. Typically, object points are located at corners (vertices). For example, a cube has eight object points.

Grid Snap

Set the **Grid snap** field to **Yes** to force the system to grab the nearest point on the mouse grid. If **Object snap** is also in effect, the system snaps to either an object point or a grid point, depending on which is closer.
Keyboard Entry

Set the *Keyboard entry* field to *Yes* only if you wish to type in the coordinates of each point you select. If this field is set to *Yes*, each time a point is selected fields with u- and v-coordinates are displayed in the current unit of length. Accept them or enter new values. If either *Grid snap* or *Object snap* is selected, the default coordinates are those associated with the closest grid or object point.

In commands involving a series of points, such as *Sketch/Line*, points other than the first can be selected by entering the coordinates directly or by entering u- and v-offsets. Therefore, an *Offset From* prompt appears after the *Sketch/Line* command is chosen. *Offset From* indicates to what point, previous or first, the offset is relative.

**Note:** If the *Object snap*, *Grid snap*, and *Keyboard entry* fields are all set to *No*, the system is in “free mode” and selects whatever screen point you click a mouse button on, regardless of its coordinates. This mode is not recommended for use in creating objects because it is very difficult to select the same point twice to complete an object.

Setting Defaults

> To set the global defaults:

1. Set the fields in the *Grids* and *Mouse* list boxes.
2. Choose *Execute* to complete the command.

After you do, all dimensions and coordinates are displayed in the new unit of length, and you must specify all subsequent dimensions and coordinates in the new units.
Window Menu

Use the **Window** commands to do the following:

- Zoom, shift, magnify, and refresh the view in the active subwindow.
- Add, subtract, or change the sizes of subwindows.
- Display the distance between two points.

Window Commands

The function of each **Window** command is as follows:

- **Setup**: Allows viewing windows to be added, removed, and resized. Viewing parameters associated with the active window can also be adjusted with this command.
- **Zoom**: Zooms in on an area of the active window.
- **UnZoom**: Returns the view in the active window to what it was before zooming.
- **Shift**: Shifts the field of view in the active window by the specified amount.
- **Magnify**: Magnifies the view in the active window by the specified amount.
- **Refresh**: Refreshes the active window by redrawing all objects and lines.
- **Measure**: Displays the distance between two points.
Windows and Subwindows

Windows and subwindows in the Post Processor are handled similarly to those in the 2D modeler.

Post Processor Windows

The Post Processor Window commands do not operate in the same manner as the Window commands in the 2D Modeler.

The Post Processor is actually a modified version of a general Post Processor capable of displaying and manipulating 3D field quantities. Therefore, not all of the subwindow features that are present in the Post Processor will be applicable for your 2D applications.

For example, the viewing angles of the various subwindows can be modified to display the model from any angle. This feature, which makes sense for 3D models, has little use in viewing 2D models — except perhaps for displaying raised contour plots from a different angle.

Active Subwindow

Initially, the Post Processor screen provides you with one active subwindow, which is displayed fully expanded on the screen. This subwindow is window 3, and is the “active” subwindow. To display other subwindows, choose Window/Setup/OnOff. In general, the Window commands operate on the active subwindow without affecting the other subwindows.

Once you have displayed several subwindows at once, click on the subwindow you wish to make the active subwindow.

The active window frame and the window itself may change color when it is selected. This depends on how you have set up the following:

- On workstations, the Maxwell*activeBackground and Maxwell*borderBackground options in your .Xdefaults file.
- On PCs, the Active Border option in the Windows Control Panel Color utility.

The active window is automatically moved to the top of the stack and remains active until you select a new one.
Manipulating Windows

For even finer control over the position and size of windows than the Window commands allow, use the window frames of the subwindows.

Field of View

The field of view refers to the section of the problem region that is being viewed. To magnify or shift the field of view, use the Window/Shift, Window/Magnify, and Window/Zoom commands.

Resizing and Repositioning Windows

A subwindow can be turned off or expanded to fill a larger portion of the screen. New subwindows can also be added to the screen. Use the Window/Setup/OnOff, Window/Setup/FullScreen, and Window/Setup/QuadScreen commands to manipulate the subwindows in this way.

If you are using a PC, you can gain even finer control of the position and size of subwindows than the Window commands allow using the Motif or Microsoft Windows frame to resize or move the windows.
<table>
<thead>
<tr>
<th>Window Menu</th>
<th>Window Commands</th>
<th>Windows and Subwindows</th>
</tr>
</thead>
<tbody>
<tr>
<td>Window/Setup</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Window/Setup/Settings</td>
<td>Window/Setup/Full Screen</td>
<td></td>
</tr>
<tr>
<td>Window/Setup/Quad Screen</td>
<td>Window/Setup/Quad All</td>
<td></td>
</tr>
<tr>
<td>Window/Setup/On-Off</td>
<td>Window/Zoom</td>
<td></td>
</tr>
<tr>
<td>Window/UnZoom</td>
<td>Window/Shift</td>
<td></td>
</tr>
<tr>
<td>Window/Magnify</td>
<td>Window/Refresh</td>
<td></td>
</tr>
<tr>
<td>Window/Measure</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Window/Setup**

Choose **Window/Setup** to change the view into the problem space, re-arrange viewing windows, and turn windows on or off. The following commands are available:

- **Settings**: Sets viewing parameters for the active window such as the line of sight, whether keys, axes and grids are displayed, and so forth.
- **Full Screen**: Expands the active window to the full size of the screen.
- **Quad Screen**: Returns the active window to its original size and location.
- **Quad All**: Returns all windows to their original size and location.
- **On-Off**: Allows viewing windows to be added to the screen.

The **Settings**, **Full Screen**, and **Quad Screen** commands only affect the active window. Therefore, select the active window by clicking on it before choosing one of these commands.
Choose **Window/Setup/Settings** to alter the viewing parameters associated with the active window. Altering these parameters allows you to redefine the line of sight from which objects are viewed and to specify whether or not keys, grids, and axes should be displayed.

When you choose this command, the following window appears:

> To change the settings for a window:
1. Set the desired window settings.
2. Choose **Execute** to complete the command.
Theta and Phi

The line of sight is the direction from which the problem space is viewed. As shown below, the direction is specified by two angles:

- The angle from the z-axis (Phi).
- The angle from the x-axis (Theta).

Visualize the line of sight when $Phi = 0$ and $Theta = 0$ as coinciding with the z-axis. Then, to determine where the line of sight is for other values of $Phi$ and $Theta$, first rotate the line of sight away from the z-axis (pivoting around the y-axis) by $Phi$ degrees. Then, rotate the line of sight away from the x-axis (pivoting around the z-axis) by $Theta$ degrees.

For 2D models, keep the defaults $Phi = 0$, and $Theta = 270$. In this way, a front-on view of the model will always be displayed.

Show Axis

Set this field to No to prevent the x-, y-, and z-axes from being displayed in the active window.

Show Grid

Set the Show $xy$ grid, Show $xz$ grid, and/or Show $yz$ grid fields to Yes to display grids in one or all of the corresponding planes.
Show Octant Only

Initially, only the positive x-, y-, and z-axes are displayed. To display the negative portions of the axes as well, set this field to Yes. If an object that is being displayed falls outside the positive octant, the negative axes are automatically displayed.

Show Key

Set this field to No to prevent the grid key from appearing in the corner of the active viewing window.

Show Section Key

Set this field to No to prevent the uv section key from appearing in the active viewing window. This section key shows the orientation of the plane that currently defines the cross section viewed in 2D windows.

Window/Setup/Full Screen

Choose Window/Setup/Full Screen to expand the active window to the full size of the screen.

Note:

If you plan to work with one viewing window expanded to the full screen size for a long period of time, choose Window/Setup/On-Off to turn off the windows that are hidden from view. Otherwise, the system continues to draw objects in the windows underneath, slowing the system down. The same holds true if you use the Print option in the Maxwell Control Panel to print screen captures on PostScript printers: turn off the hidden windows. Otherwise, the system creates a screen description that includes the underlying windows, resulting in files that are larger than necessary.

Window/Setup/Quad Screen

Choose Window/Setup/Quad Screen to return the active window to its original size and position. The window's initial position depends on its number. For example, the initial position of Window 1 is in the upper left corner of the screen.
Window/Setup/Quad All

Choose Window/Setup/Quad All to return all windows to their original size and position. Only those windows that have been turned on with the Window/Setup/On-Off command are displayed.

Window/Setup/On-Off

Choose Window/Setup/On-Off to add or subtract viewing windows from the screen. A pop-up window listing all available windows appears, as shown below (the default settings for the windows are shown).

> To enable or disable a particular window:
1. To display a window, set the window to Yes. Windows set to No do not appear on the screen.
2. Choose Execute to complete the command.

Note: Window 3, the window that is displayed on many command menus, is the only window that cannot be turned off.
Window/Zoom

Choose **Window/Zoom** to zoom in on a region of the active window.

- To zoom in on an area of the window:
  1. Choose **Window/Zoom**.
  2. Select a point at one corner of region that is to be zoomed.
  3. Select the point in the diagonal corner.

  The system then expands the selected region to fill the active window.

**Note:**

The **Window/Magnify** command does not operate while a window is in a zoomed state. Choose **Window/UnZoom** before attempting to use this command.

Window/UnZoom

Choose **Window/UnZoom** to set the field of view of the active window back to the size it was before any zooming was performed. The settings associated with the unzoomed state are those that were defined when the **Global/Defaults** and **Window/Magnify** commands were last chosen.

Window/Shift

Choose **Window/Shift** to shift the field of view in the active window horizontally or vertically. You will specify a reference point and its new location.

- To shift the field of view in the active window:
  1. Choose **Window/Shift**.
  2. Click the left mouse button on a point in the active window to select the reference point.
  3. Click the left mouse button on a point in the active window to select the new location for the reference point.

After you pick the two points, the field of view is shifted so that the reference point appears in the new location.
Window/Magnify

Choose **Window/Magnify** to magnify the view in the active window by a specified amount. This command also expects you to select a reference point and a new position for the reference point so that the objects of interest remain inside the field of view.

To magnify the view in the active window:

1. Choose **Window/Magnify**. A window appears, showing the **Multiple** field.
2. Enter a value for the multiple. A number greater than 1 causes objects to appear larger. A number less than 1 causes objects to appear smaller, resulting in a wider field of view. For example, a 2 results in objects appearing twice as large. A .5 results in objects appearing half as large.
3. Choose **Execute**.
4. Click on a point in the active window to select a reference point.
5. Click on a point to select the new location of the reference point.

The magnified image then appears on the screen. Objects appear larger or smaller, depending on the value you specified. The reference point you selected appears at the desired location.

**Note:** The command **Window/Magnify** does not operate while a window is in a zoomed state. Choose **Window/UnZoom** before attempting to use this command.
Window/Refresh

Choose **Window/Refresh** if the active viewing window gets too cluttered with temporary information that is no longer needed. The system erases all objects, lines, and other data from the active viewing window and then immediately redraws the objects and lines. Only the objects and lines that are on screen when the command is chosen are redrawn.

Temporary information includes such things as plots, shaded views of objects, and meshes that are displayed with the post-processing commands.

To refresh all viewing windows simultaneously, choose **Global/Refresh**.
Choose **Window/Measure** to display the distance between two selected points in the active viewing window. The following window appears:

- **Distance**
  - \( x_1 \)
  - \( x_2 \)
  - \( y_1 \)
  - \( y_2 \)

To display the distance between two points in the active window:

1. Choose **mouse** associated with the first point.
2. Move the cursor to an object point in the active window and click the left mouse button to pick the point. The coordinates of the point appear in the menu.
3. Choose **mouse** for the second point and pick the second point in the same way as the first. The value that appears in the field **distance** represents the distance between the two points.
4. Repeat steps 1 through 3 to choose additional points for which you wish to display the distances.
5. Choose **Return** to exit the command.

Note: Choose **Global/Defaults** to determine the snap-to behavior of the mouse.
Level of Precision

Fifteen significant digits are displayed for distances and for the coordinates of object points. Displaying this level of precision is required if you plan to use the data to specify the position of objects.

For example, if you use a distance as input to another command — such as a command on the Edit menu — be sure to use the full level of precision. Otherwise, it is possible to end up with two objects that look like they are aligned but are actually offset by a small amount. In some cases, the difference will be enough to cause problems when attempting to unite the objects or when attempting to mesh the objects.

Note: In general, whenever you explicitly type in coordinates as input to commands which control the position of objects, use as much precision as is available.
Show Menu

Use the Show commands to inspect the finite element mesh associated with a structure and to display shaded views of objects.

The Show commands are available when analyzing the solution.

Show Commands

The function of each Show command is as follows:

- **WireMesh**: Displays a wireframe model of the entire finite element mesh.
- **ShadMesh**: Displays a shaded view of the entire mesh, including the elements in the interior of objects. The system shrinks the triangles before displaying them, allowing the interior of the mesh to be partially visible.
- **InfoMesh**: Displays information about the mesh, including the largest and smallest triangles for each object.

These commands are used to inspect the mesh in the Post Processor.
Show/WireMesh

Choose Show/WireMesh to display a wire model of the finite element mesh such as the one shown below. In this example, mesh lines have been drawn for the triangles that lie inside the objects in the model:

> To display a wire mesh:
1. Choose Show/WireMesh. A menu similar to the following one appears, displaying
the names of all the objects in the model:

2. Set **Draw mesh** to **Yes** for the objects that you wish to display.
3. For better quality screen captures, set **Better Hardcopy** to **Yes**.
   - On workstations, set this to **Yes** only if you are printing a screen capture to a PostScript printer or saving it to a PostScript file. The system stores the plotted image that's displayed on the screen as a bitmap, allowing you to make better quality PostScript screen captures.
   - On PCs, set this to **Yes** to improve the quality of your screen captures, regardless of the printer type.
   Doing so requires additional memory.
4. Choose **Execute** to complete the command.

A wire model of the finite element mesh inside the selected objects is then displayed. The edges of the triangles are displayed in an unshaded view.
Choose **Show/ShadMesh** to display shaded views of the triangles that make up the finite element mesh. An example of such a view is shown below. In this example, only the triangles that lie inside the objects are shown:

> To display a shaded view of the mesh:
1. Choose **Show/ShadMesh**. A menu similar to the one shown here appears, listing
2. Set \textit{Draw interior mesh} to \textbf{Yes} for the objects for which to display the mesh.
3. For better quality screen captures of the mesh, set \textbf{Better Hardcopy} to \textbf{Yes}.
   \begin{itemize}
   \item On UNIX workstations, set this to \textbf{Yes} only if you are printing a screen capture to a PostScript printer or saving it to a PostScript file. The system stores the plotted image that's displayed on the screen as a bitmap, allowing you to make better quality PostScript screen captures.
   \item On PCs, set this to \textbf{Yes} to improve the quality of your screen captures, regardless of the printer type.
   \end{itemize}
   Doing so requires additional memory.
4. Choose \textbf{Execute} to complete the command.

A shaded view of the triangles inside the selected objects is then displayed. For purposes of clarity, the triangles are shown slightly smaller than they actually are so that their boundaries are obvious.
Show/InfoMesh

Choose Show/InfoMesh to display information about the mesh. A window similar to the following one appears:

Maximum and Minimum

The maximum and minimum coordinates of the extents of the problem space are displayed in the fields Maximum X, Maximum Y, Maximum Z, Minimum X, Minimum Y, and Minimum Z. Minimum and maximum z-coordinates are displayed by default. They are always zero.

Total Triangles

The Total Triangles field shows the total number of triangles in the finite element mesh.
Vertex Points

The Vertex Points field shows the total number of vertex points in the finite element mesh. Each triangle has six vertex points, sharing each point with multiple neighbors.

Largest and Smallest Triangle (By Object)

An indication of how uniformly the mesh has been refined in each object is given by the size of the largest and smallest triangle in that object. The values shown are areas stated in the current units.
Use the **Post** commands shown below to do the following:

- Display shaded, contour, vector, and quantity versus distance plots of common field quantities.
- For an electrostatic simulation, you can plot the electric potential, \( \phi(x,y,z) \), the electric field, \( \mathbf{E}(x,y,z) \), and the electric flux density, \( \mathbf{D}(x,y,z) \).
- For a magnetostatic or eddy current simulation, you can plot the current density, \( \mathbf{J}(x,y,z) \), the magnetic field, \( \mathbf{H}(x,y,z) \), and the magnetic flux density, \( \mathbf{B}(x,y,z) \).
- For an eddy axial simulation, you can plot the electric field, \( \mathbf{E}(x,y,z) \), the current density, \( \mathbf{J}(x,y,z) \), and the z-component of the H-field.
- For a conduction (AC or DC) simulation, you can plot the electric potential, \( \phi(x,y,z) \), the electric field, \( \mathbf{E}(x,y,z) \), and the current density, \( \mathbf{J}(x,y,z) \).
- Define field registers for use with the line calculator.
- Display shaded, contour, vector, and quantity versus distance plots of field quantities stored in plane and line registers.

When you choose **Post** from the Post Processor menu bar, the following menu appears:

**Note:** For simplicity's sake, the information here is written with cartesian \((x,y,z)\) geometries in mind. It applies as well to axisymmetric geometries \((r,\theta,z)\). Coordinates are expressed as 3 dimensional points. In Maxwell 2D, \(z\) will be set to zero for cartesian geometries, and \(\theta\) will be set to zero for axisymmetric geometries.
Post Commands

The function of each Post command is as follows:

- **Plot**: Displays vector plots, contour plots, shaded plots, and value versus distance graphs of common field quantities.
- **Line**: Enters lines as registers in the line calculator and plots the field quantity in the top line register as a function of distance along a line.
- **Plane**: Enters planes as registers in the plane calculator and displays contour, arrow, and shaded plots of the field quantity in the top register of the plane calculator.
- **BH Examine**: (Magnetostatic problems with nonlinear materials only.) Allows you to compute the B and H values for selected points in a nonlinear material.
- **BH Plot**: (Magnetostatic problems with nonlinear materials only.) Allows you to plot the location of the points that were selected with the Post/BH Examine command on a nonlinear material's BH-curve. This enables you to analyze saturation levels.

The Post/Plot command accesses an interface for plotting basic field quantities. The Post/Line and Post/Plane commands allow you to plot quantities in the top registers of the calculators that are derived from the basic field quantities using the Calc commands. Together, these commands provide you with powerful post-processing features that let you display, manipulate, and analyze the field solution.
Topics:
- Plotting Solutions
- Line Segments
- Plotting Common Field Quantities
- Plotting Derived Field Quantities
- Plotting Derived Quantities Along a Line
- Analyzing Saturation Levels in Nonlinear Materials
- Aborting Plots

Maxwell 2D — Post Menu

Plotting Solutions

Use the Post commands to plot field quantities — either common field quantities, such as $D$, $B$, $E$, and $H$, or derived quantities from the solution calculators.

Line Segments

To do any kind of plotting along a path, you first need to define the line segment over which the plot is to be displayed. For example, to display a graph of how the magnitude of the voltage varies along a given path, first define a line segment that follows the desired path using the Post/Line/Define command.

Plotting Common Field Quantities

Use the Post/Plot command to display a common quantity such as the B-field, H-field, J-field, and so forth. You can specify whether to plot over the entire problem region or along a path defined by the Post/Line/Define command.

Plotting Derived Field Quantities

Maxwell 2D supports a set of calculators that allow you to load basic field quantities into registers and to compute and then plot derived quantities from the basic fields. For example, because the current density field, $J(x,y,z)$, is equal to the curl of the H-field, $J = \nabla \times H$

you can compute the current density field by loading $H$ into the plane calculator and taking its curl. You can perform other functions using the plane calculator — such as integrating a quantity in the problem region.

To display a shaded, contour, or arrow plot of a derived field quantity in the problem region, use the Post/Plane command.

Note:
Remember that you can only choose the Post/Plane/Contour and Post/Plane/Shade commands when you have a scalar value in the top register of the plane calculator. For this example, if you choose magnitude before exiting the plane calculator, you can then choose either of these commands to plot this scalar value.
Plotting Derived Quantities Along a Line

It is also possible to plot value versus distance graphs of a derived quantity along a line. To plot along a line, use the Post/Line command. You can perform other functions using the line calculator — such as integrating a field quantity over a line.

Analyzing Saturation Levels in Nonlinear Materials

To view saturation levels in nonlinear materials, use the Post/BH Examine command to select the points at which you want to determine the level of saturation. Then, use the Post/BH Plot command to see where the $B$ and $H$ values of these points lie on the material’s BH-curve.

Aborting Plots

Choose Abort from the progress bar to stop a plot that is currently in progress.
Post/Plot

Choose Post/Plot to display vector plots, contour plots, shaded plots, and value versus distance graphs. The following window appears:

The general procedure for using this command is as follows:

1. If appropriate, define the line over which the field quantity is to be plotted. Use the Post/Line/Define command.
2. Choose Post/Plot to display the Plot Menu.
3. Set the Value (the field quantity) that is to be plotted to Yes. For example, set Mag E to Yes to display a plot of the E-field’s magnitude.
4. Set the desired Plot Type to Yes. For example, set Shaded to Yes to display a shaded plot.
5. Choose the Location in the problem space where the field quantity is to be plotted. For example, to display the magnitude of a field quantity over a line, set Line to Yes, then choose the desired line segment.
6. Set the Window in which to display the plot to Yes. In general, leave this option set to the default (Window 3, the main window that is displayed in the Post Processor).
7. Do one of the following to set the color for the plot:
To display a single-color plot, leave **Spectrum** set to **No** and choose a color by:

- Clicking the left mouse button on the square next to **Color**. A palette of 16 colors appears.
- Select the color you want. The new color appears in the square.

To display a spectrum plot, set **Spectrum** to **Yes**.

8. *(AC solvers only.)* Set the **Phase** field to the phase angle at which the selected field quantity is to be plotted.

9. Choose **Execute**.

The plot appears in the selected window.

The **Plot Menu** only allows you to plot common field quantities. To plot quantities that have been computed using the calculators, use **Post/Plane** and **Post/Line**.

### Values

You can plot field quantities, material properties, energy, and power, depend on the solver selected for the model.

### Note:

For AC solvers, the phase angle at which these quantities are plotted is entered in the **Phase** field.

When you plot a material property for an anisotropic material, you are shown the value of the property in a single direction. For XY problems, value of material properties in the x direction are plotted; for RZ problems, in the z direction. For isotropic materials, the value of a property is the same in all directions.

The following material properties are available for plotting:

- permeability ($\mu_r\mu_0$)
- permittivity ($\varepsilon_r\varepsilon_0$)
- conductivity ($\sigma$)
Electrostatic Field Quantities

The following values may be plotted for an electrostatic field simulation:

**Voltage**
Scalar field representing the electric potential, \(\phi(x,y,z)\).

**E Vector**
Vector field representing the electric field, \(\mathbf{E}(x,y,z)\).

**D Vector**
Vector field representing the electric flux density, \(\mathbf{D}(x,y,z)\).

**Mag E**
Scalar field representing the magnitude of the electric field, \(|\mathbf{E}(x,y,z)|\).

**Mag D**
Scalar field representing the magnitude of the electric flux density, \(|\mathbf{D}(x,y,z)|\).

**epsilon_x**
Permittivity.

**energy**
Stored electric field energy.

AC and DC Conduction Field Quantities

The following values may be plotted for AC or DC conduction field simulations:

**Voltage**
Scalar field representing the electric potential, \(\phi(x,y,z,t)\).

**E Vector**
Vector field representing the electric field, \(\mathbf{E}(x,y,z,t)\).

**J Vector**
Vector field representing the current density field, \(\mathbf{J}(x,y,z,t)\).

**Mag E**
Scalar field representing the magnitude of the electric field, \(|\mathbf{E}(x,y,z,t)|\).

**Mag J**
Scalar field representing the magnitude of the current density \(|\mathbf{J}(x,y,z,t)|\).

**epsilon_x**
Permittivity (AC Conduction).

**sigma_x**
Conductivity.

**power**
Power dissipated.
Magnetostatic, Transient, and Eddy Current Field Quantities

The following values may be plotted if a DC magnetic (magnetostatic), transient, or AC magnetic (eddy current) field simulation has been performed:

**Flux Lines**
Scalar field representing the z-component of the magnetic vector potential, $A$. Because all current is assumed to flow perpendicular to the cross section, plotting equipotential contours of $A$ is equivalent to plotting magnetic flux lines.

**B Vector**
Vector field representing the magnetic flux density, $B(x,y,z)$.

**H Vector**
Vector field representing the magnetic field, $H(x,y,z)$.

**Mag B**
Scalar field representing the magnitude of the magnetic flux density, $|B|(x,y,z)$.

**Mag H**
Scalar field representing the magnitude of the magnetic flux density, $|H|(x,y,z)$.

**Mu_x**
Permeability. This is $\mu_z$ for RZ problems.

**Epsilon_x**
Permittivity (Eddy Current). This is $\varepsilon_z$ for RZ problems.

**Sigma_X**
$\sigma$. This is $\sigma_z$ for RZ problems.

**Energy**
For linear materials, the actual energy defined by $\frac{1}{2}\int BV \cdot HB\,dV$.
For nonlinear materials, the energy given by $\int BV \cdot HB\,dV$.

**Coenergy**
For nonlinear materials, the energy given by $\int BV \cdot (dH)(dV)$.

**App Egy**
For nonlinear materials, the apparent energy given by $\frac{1}{2}\int BV \cdot HB\,dV$.

**Power**
Power dissipated (Eddy Current).
Eddy Axial Field Quantities

The following values may be plotted if an eddy axial field simulation has been performed:

- **H(z)**: Scalar field representing the z-component of the magnetic field, \( \mathbf{H}_z(x,y,z,t) \).
- **J Vector**: Vector field representing the current density field, \( \mathbf{J}(x,y,z,t) \).
- **E Vector**: Vector field representing the electric field, \( \mathbf{E}(x,y,z,t) \).
- **Mag J**: Scalar field representing the magnitude of the current density \( |\mathbf{J}|(x,y,z,t) \).
- **Mag E**: Scalar field representing the magnitude of the electric field, \( |\mathbf{E}|(x,y,z,t) \).
- **\( \epsilon_x \)**: Permittivity.
- **\( \mu \)**: Permeability.
- **\( \sigma_x \)**: Conductivity.
- **energy**: Stored electric field energy.
- **power**: Power dissipated.

**Type of Plot**

Choose the **Plot Type** from the **Plot Menu**. The available options are:

- **Arrow**: Displays arrows representing the magnitude and direction of a vector field quantity.
- **Shaded**: Displays colors representing the magnitude of a scalar field quantity.
- **Contour**: Displays the contours of equal value associated with a scalar field quantity.
- **Graph**: Displays a graph of the desired field quantity versus the distance along a line.
Arrow Plot

Set Arrow to Yes to plot a vector field quantity. Arrows are used to represent the direction (and optionally, the magnitude) of the field quantity. The following figure shows the E-field for two striplines on a substrate:

To redisplay the arrow plot with different scaling options, choose Post/Plane/Arrow.
Shaded Plot

Set **Shaded** to **Yes** to display a shaded plot of a scalar field quantity. The magnitude of the quantity being plotted is represented by a range of colors. A color scale showing the range of values also appears on the screen. A black and white rendering of a shaded plot of voltages for a cracked plate is shown below. Only scalar quantities can be displayed as shaded plots.

To redisplay a shaded plot that is scaled differently, choose **Post/Plane/Shade**.
Contour Plot

Set Contour to Yes to display the “equal value” contours of a scalar field quantity. A contour plot of the electric field is shown below. Only scalar quantities can be displayed as contour plots.

To redisplay a contour plot that is scaled differently, choose Post/Plane/Contour.
Graph (Line Plots)

Set **Graph** to **Yes** to display a value versus distance plot of a scalar field quantity over a path (line).

The following figure shows a line plot in which the magnitude of the electric potential is plotted along a line that cuts through the middle of a structure. From the plot, you can see that the magnitude of the electric potential starts off negative on one side of the structure, increases to zero volts in the middle, and continues to increase on the other side of the structure.

![Line Plot Example](image.png)

Unlike shaded, contour, and vector plots — which display the portion of a scalar field quantity that lies in the problem region — a graph displays the portion of a field quantity that lies along a path. Therefore, when displaying a line graph you must first identify a line. Create the desired line segment with the command **Post/Line/Define**.
Adjusting a Line Graph's Display

After a line plot is displayed, you can adjust the way in which a line plot is displayed or view coordinates for points by using the following commands in the **Plot Options** menu:

- **Plot Settings**
  - Specifies axis scales, labels, plot headings, and the minimum and maximum field values to be plotted. Also determines the axis tick mark and grid settings. For more details, see the description of the **Plot/Format Axes** command.

- **Graph Settings**
  - Specifies the color, line thickness, and line style of a trace on a plot. Also determines the type of markers displayed at solution data points, and whether the trace is visible on the plot. For more details, see the description of the **Plot/Format Graphs** command.

- **Zoom In**
  - Magnifies a specific portion of the graph.

- **Zoom Out**
  - Reduces a section of the graph.

- **Fit All**
  - Shows all of the available data on the graph.

- **Show Coordinates**
  - Shows the coordinates of a point on the graph.

- **Save As**
  - Save the current plot to a file.

### Zoom In

To zoom in on a portion of the graph:

1. Choose **Zoom In** from the **Plot Options** menu.
2. Click the left mouse button at one corner of the rectangular region you want to magnify.
3. Drag the mouse until the rectangle drawn on the screen represents the region in which you are interested.
4. Click the left mouse button again.

The selected region is shown at the appropriate magnification. To cancel the zoom operation while selecting the region, click with the right mouse button.
To zoom out of a portion of the graph:
1. Choose **Zoom Out** from the **Plot Options** menu. Click the left mouse button at one corner of the rectangular region you want to reduce.
2. Drag the mouse until the rectangle encloses the region you are interested in.
3. Click the left mouse button again. The selected region is shown at the appropriate magnification.

To cancel the zoom operation while selecting the region, click with the right mouse button.

**Fit All**
To redisplay the entire plot on the graph:
- Choose **Fit All** from the **Plot Options** menu. The entire plot, as defined by **Plot Settings**, is displayed.

**Show Coordinates**
To display the coordinates of a point on the graph:
1. Choose **Show Coordinates** from the **Plot Options** menu.
2. Click the left mouse button on the desired point to display its coordinates.
3. Continue to click the left mouse button on additional points.

When you are done, click the right mouse button.

**Save As**
To display the coordinates of a point on the graph:
1. Choose **Save As** from the **Plot Options** menu.
2. Enter the path and the filename where you want to save the plot.
3. Choose **OK**.
Location of Plot

After choosing the type of plot, choose whether to display the plot over the entire problem region or over a line segment.

Plane

Displays a field quantity over the entire problem region. **Plane** can only be used with **Shaded** or **Contour** plot types.

Line

Displays a quantity versus distance graph. The names of all lines that have been defined with the command **Post/Line/Define** are displayed. Choose one, and then choose **Execute**. After you do, the system plots the selected scalar value as a function of distance along the line. **Line** can only be used with the plot type **Graph**.

Window

Choose the subwindow in which to display the plot by setting the desired window to **Yes**. Window 3 — the main subwindow in the Post Processor — is selected by default. To display additional windows from which to choose, use the **Window/Setup/On-Off** command.

Color

Select the color of the plot in one of the following ways:

- Set **Spectrum** to **Yes** to display a plot of several different colors. In general, areas with low field values are plotted in blue, and areas with high field values are plotted in red.
- Choose a single color to display a plot using different shades of one color. The lowest field values are plotted in the darkest shade of the color, and the highest field values are plotted in the lightest. To do so:
  1. Set **Spectrum** to **No**.
  2. Click the left mouse button on the color box. A color palette appears.
  3. Choose the desired color.

**Note:** If your computer has a black and white screen or you are printing a black and white screen capture, plot field quantities using a single color. Choosing **Spectrum** results in the plot being a single shade of gray because all colors are displayed at the same intensity.
Better Hardcopy

Improves the quality of printed plots.

- On UNIX workstations, set this to **Yes** only if you are printing a screen capture to a PostScript printer or saving it to a PostScript file. The system stores the plotted image on the screen as a bitmap, allowing you to make better quality PostScript screen captures.
- On PCs, set this to **Yes** to improve the quality of your screen captures, regardless of the printer type.

Be aware that this uses more computer memory than if an ordinary plot is drawn.

Phase

*(AC Conduction, Eddy Axial, Eddy Current)*

All AC field quantities oscillate at the frequency specified when you generated a solution for the model. The Maxwell 2D represents these quantities as phasors. For instance, the magnetic flux density, $B$, can be represented as:

$$B(x, y, t) = B(x, y) \cos(\omega t + \theta)$$

where:

- $\omega$ is the angular frequency at which the magnetic field and source currents are oscillating, specified during the solution.
- $\theta$ is the phase angle (the offset from a cosine wave that peaks at $\omega t=0$).

To view field quantities at different points in their cycle, enter a different value for the phase angle, $\theta$, in the **Phase** field. Enter it in degrees.
Post/Line

Choose Post/Line to define line segments, enter line segments as registers in the line calculator and display value versus distance graphs of the field quantity in the top register of the line calculator.

The following commands appear when you choose Post/Line:

- **Define** Allows you to specify a line segment over which a field quantity is to be plotted or integrated.
- **Entry** Enters line segments as registers in the line calculator.
- **Display** Displays the location of any line segment that has already been entered as a line register.
- **Plot** Plots a value versus distance graph of the scalar field quantity in the top line register.
- **Value** Maps values from the top plane register to the top line register.

The Post/Line commands are graphical support commands for the line calculator. They allow you to plot value versus distance graphs of field quantities that have been manipulated with the line calculator.

**General Procedure**

The general procedure for using the Post/Line commands to display value versus distance plots is as follows:

1. Choose Post/Line/Define to create the path in which you are interested.
2. Choose Post/Line/Entry to declare the line segment as a register.
3. Choose Calc/Plane to gain access to the plane calculator.
4. Use the appropriate plane calculator commands to load a field quantity (such as $B$ or $H$) into a plane register. Then manipulate that quantity to obtain the desired results. For example, take the dot product of $B$ and $H$ to obtain the energy density field.
5. Choose Calc/Line to gain access to the line calculator.
6. Choose Calc/Line/value to map the field quantity that is in the top register of the plane calculator to the top register of the line calculator.
7. Choose Post/Line/Plot to plot the value of the field quantity in the top line register versus distance along the selected line.
Only use this procedure to plot field quantities that need to be manipulated in the line calculator. To plot common field quantities that do not need to be manipulated, use the command Post/Plot instead.

**Post/Line/Define**

Choose Post/Line/Define to identify a line segment over which a field quantity is to be plotted or used as a line calculator register. The following window appears:
Defining Line Segments

To define a line segment:
2. Do one of the following:
   • Draw a line consisting of one or more straight line segments.
   • Draw an arc.
   • Define the outside edge of an object as a line.
   If necessary, use the Zoom, Fill, Set, and Refresh buttons at the bottom of the menu to change the display of the problem region before defining the line segment.

3. Enter a name for the new line segment in the Line segment name field.
   Otherwise, the default name (set with the Global/Defaults command) is used.
4. Choose one of the following to complete the command:
   • Choose Execute to define the line and remain in the Line Segment Menu.
   • Choose Execute and Return to define the line and quit the Line Segment Menu.

Note: The mouse must be in “object snap” mode if you are to pick “object” points (points at the vertices of objects) when defining the line segment. If necessary, use the Global/Defaults command to set the mouse mode.

Straight Line Segments

To define a straight line segment:
1. Choose Enter Line. The cursor changes to a crosshair.
2. Click the left mouse button on the first point in the line.
3. Click the left mouse button on each subsequent point.
4. Click the left mouse button twice on the final point.

Note: If desired, click the right mouse button to delete the last point you added.

The x- and y-coordinates of the line’s origin and endpoint are displayed on the left side of the screen.
To define an arc as a line segment:
1. Choose Enter Arc. A window appears, asking you to define the direction of the arc.
2. Do one of the following:
   • Choose Clockwise to create an arc that is drawn clockwise from the start point to the endpoint.
   • Choose Counterclockwise to create an arc that is drawn counterclockwise from the start point to the endpoint.
3. Move the cursor to the center of the arc and click the left mouse button.
4. Move the cursor to the start point of the arc and click the left mouse button.
5. Move the cursor to the endpoint of the arc and click the left mouse button.
6. After the endpoint of the arc is defined, a window appears, asking you to define the number of segments in the arc.
7. Either accept the default of 10 or enter a new value. Because the system represents all curves by a series of line segments, you need to specify the desired number of segments. The default of 10 creates an approximation of the arc consisting of 10 line segments.
8. Choose Continue.

The x- and y-coordinates of each point on the arc are displayed on the left side of the screen.

To define the outside edge of an object as a line segment:
1. Choose Enter Object. The cursor changes to crosshairs.
2. Click the right mouse button on the desired object.

The x- and y-coordinates of each vertex point on the object are displayed on the left side of the screen.
Modifying Previously Defined Line Segments

1. Set the desired line segment in the **Line Segment List** list box to **Yes**. The coordinates defining the line appear.
2. Edit the coordinates of the points that define the line segment.
3. If desired, enter a new name in **Line segment name** to prevent the existing line segment from being overwritten.
4. Choose **Execute** or **Execute and Return** to complete the command. No change is made to the line segment until you choose one of these commands.

Displaying Line Segments

1. Set the desired line in the **Line Segment List** list box to **Yes**.
2. Choose **Display**. The system displays the line segment.

Deleting Line Segments

1. Set the desired line in the **Line Segment List** list box to **Yes**.
2. Set **Delete** to **Yes**.
3. Choose **Execute and Return** or **Execute** to complete the command.

Post/Line/Entry

Use **Post/Line/Entry** to link a line segment to a line register.

1. Choose **Post/Line/Define** to create the line segment.
2. Choose **Post/Line/Entry**. A list of the names of all line segments appears.
3. Set the name of the desired line segment to **Yes**.
4. Choose **Execute**.

You cannot use the line calculator until at least one line segment has been linked to a register.

**Note:** When the command **Post/Plot** is used to display a graph over a line segment, the system automatically loads that line segment as a line register.
Post/Line/Display

Choose **Post/Line/Display** to display the location of all lines that are linked to line registers.

Post/Line/Plot

Choose **Post/Line/Plot** to display a scalar or vector value versus distance plot of the data in the top line register. The result is a plot similar to the one generated by selecting the **Graph** plot from the **Plot Menu**. The value of the field quantity stored in the top line register is plotted over the length of the appropriate line segment. You can format the line plot as you would one generated by the **Graph** command from the **Plot Menu**.

If the quantity in the line register is a vector, the Post Processor displays three separate plots, one for each component of the vector. To plot a scalar field quantity, you can also use the **Post/Plot** command.

Post/Line/Value

After a line segment has been created to serve as a line register, choose **Post/Line/Value** to map the field quantity in the top plane register to the line register. This command is identical to the **Calc/Line/value** command.

> To map the field quantity:
1. Create the desired line with **Post/Line/Define**.
2. Declare the line as a register with **Post/Line/Entry**.
3. Choose **Calc/Plane** and place the desired field quantity in the top register of the plane calculator.
4. Choose **Post/Line/Value**. If no other line register exists, a prompt appears, asking you to enter the number of points in the line.
5. Enter a value or choose **Return** to accept the default of 100 points.

The Post Processor then explicitly maps a field value to evenly spaced points along the line. Values at all intermediate points along the line are interpolated from the explicitly defined points. All subsequent lines automatically receive the same number of explicitly defined points as the first register.
Post/Plane

Choose **Post/Plane** to display shaded, contour, and arrow plots of the field quantity in the top register of the plane calculator.

The following commands are available:

- **Contour** Displays a contour plot of the scalar quantity in the top plane register.
- **Contour Display** Redisplays the last contour plot created.
- **Shade** Displays a shaded plot of the scalar quantity in the top plane register.
- **Arrow** Displays an arrow plot of the field in the top plane register.
- **Arrow Region** Displays an arrow plot in the region you select.
- **Arrow Display** Redisplays the last arrow plot created.
- **Max - Min** Displays the maximum and minimum values of the quantity in the top plane register along with their locations in the problem region.
- **Value** Lets you determine the value of a field quantity at any point in the problem region, and map that value to the top register of the number calculator.

The **Post/Plane** commands are graphical support commands for the plane calculator that allow you to display plots of field quantities that were manipulated with the calculators.

**General Procedure**

1. Choose **Calc/Plane** to access the plane calculator.
2. Use the appropriate calculator commands to load a field quantity (such as the H-field) into the top register. Then, manipulate that quantity as desired.
3. Choose **Post/Plane/Contour, Post/Plane/Shade,** or **Post/Plane/Arrow** to display a contour, shaded, or arrow plot of the field quantity in the top plane register.

**Note:** Remember that you need a scalar value to create a contour or shaded plot, and a vector value to create an arrow plot.

This procedure is primarily used to display field quantities that need to be manipulated in the plane calculator. To plot common field quantities such as the E-field or the H-field, use the **Post/Plot** command instead.
Scale to Window or Scale to Problem

When you choose the Post/Plane/Shade or Post/Plane Contour command, a menu similar to the following one appears, allowing you to set maximum and minimum plotting limits on the field value that is to be plotted. The new release provides two new command buttons for both shaded and contour plots — Scale To Window and Scale To Problem — which give you additional control over the plotting operation.

- Choose Scale To Problem (the default) to include all the triangles in the problem when the software sets the scaling for the plot.
- Choose Scale To Window to include only the triangles contained in the current window to set the scaling for the plot. This is very useful if you Zoom In on a region of the drawing for close examination and then want to do a contour or shade plot of the zoomed region.
Post/Plane/Contour

Choose Post/Plane/Contour to plot equipotential contours of the scalar quantity in the top register of the plane calculator.

To plot the contours of the scalar quantity:
1. Use the Calc/Plane commands to load a scalar field into the top register of the plane calculator.
2. Choose Post/Plane/Contour. Enter values in the following fields.
   - Maximum Value
   - Minimum Value
   - Divisions

   Maximum and minimum values of the contours to display. The defaults are the maximum and minimum values for the entire field.

   Number of divisions for field values. For example, if the quantity in the plane calculator ranges between 100 and 500, specifying four divisions results in contours for the values 100, 200, 300, 400, and 500.

- Contour Height
- Color/Spectrum

   Specify a contour height to obtain a raised effect. The height of each contour off the plane represents the magnitude of the contour. The contour associated with the maximum value is raised off the plane by the Contour Height.

   Set Spectrum to Yes to display a spectrum plot. Set Spectrum to No to display the plot in a single color. To change the color,
   1. Click on the color box.
   2. Select the desired color.

3. Choose Return.

A progress bar and Abort button appear in the upper-left corner as the contour plot is generated. Choose Abort at any time to stop the plot.
Post/Plane/Contour Display

Choose **Post/Plane/Contour Display** to redisplay a contour plot that has been erased.

### Post/Plane/Shade

Choose **Post/Plane/Shade** to display shaded plots of the scalar quantity in the top register of the plane calculator.

> To display a shaded plot:
1. Use the **Calc/Plane** commands to load a scalar field into the top plane register.
2. Choose **Post/Plane/Shade**. Enter values in the following fields:
   - **Maximum Value, Minimum Value** Maximum and minimum values of the contours to display. The defaults are the maximum and minimum values for the entire field.
   - **Divisions** Number of divisions for field values. For example, if the quantity in the calculator ranges between 100 and 500, specifying four divisions results in contours for the values 100, 200, 300, 400, and 500.
   - **Draw Objects** Toggle **Draw Objects** to **Yes** (the default) to show object outlines in your shade plot. Toggle **Draw Objects** to **No** to omit object outlines.
   - **Plot Height** Specify a plot height to obtain a raised effect. The height of each shaded area off the plane will represent the magnitude of the contour. The shaded area associated with the maximum value is raised off the plane by the **Plot Height**.
   - **Color/Spectrum** Set **Spectrum** to **Yes** to display a spectrum plot. Set **Spectrum** to **No** to display the plot in a single color. To change the color, click on the color box, then click on the desired color.

3. Choose **Return**.

A progress bar and an **Abort** button appear in the upper-left corner as the shaded plot is generated. Choose **Abort** at any time to stop this plot.
Post/Plane/Arrow

Choose this create an arrow plot of the vector field quantity in the top register of the field calculator.

> To display an arrow plot:
1. Use the Calc/Plane commands to load a vector field quantity into the top register of the plane calculator.
2. Choose Post/Plane/Arrow. The available fields are as follows:
   - **Number of vectors**: Displays the number of vectors in the plane register.
   - **Maximum Magnitude**: Displays the maximum magnitude of the values in the top register of the plane calculator.
   - **Minimum Magnitude**: Displays the minimum magnitude of the values in the top register of the plane calculator.
   - **Arrow Length**: Allows you to specify the maximum arrow length in the current unit of length. The smallest arrow length allowable is 0.2 times the default arrow length, and the largest arrow length allowable is 4 times the default arrow length.
   - **Scale Arrows**: Set to Yes to have the Maxwell 2D represent the magnitude of the field by the size of the arrows.
   - **Shade Arrows**: Set to Yes to have the Maxwell 2D shade the arrows. The various shades do not represent magnitude. They are simply shaded views of the three-dimensional arrows.
   - **Color**: Select the color for the plot. Click the mouse button on the color square to make a palette appear, then click on the desired color.
   - **Better Hardcopy**: Improves the quality of printed plots. On workstations, choose Yes only if you are printing to a PostScript printer or to a PostScript file. On PCs, choose Yes to improve the quality of your screen captures, regardless of the printer type. Note this uses more computer memory than an ordinary plot.

3. Choose Execute to complete the command.

A bar appears in the upper-left corner as the arrow plot is generated.
Post/Plane/Arrow Region

Choose **Post/Plane/Arrow Region** to display an arrow plot of the field quantity in a rectangular area. This lets you plot vector quantities in specific areas of interest, instead of over the entire problem region.

To display an arrow plot in a rectangular area:

1. Use the **Calc/Plane** commands to load a vector field quantity into the top register of the plane calculator. If desired, manipulate that quantity to derive some other quantity of interest.
2. Choose **Post/Plane/Arrow Region**. The following window appears:

3. Select the first corner of the rectangular region:
   - To select the point with the mouse:
     a. Choose **Mouse** under **Point 1**.
     b. Click the left mouse button on the desired point. Its coordinates appear in the X and Y fields.
   - To select the point with the keyboard, simply enter the desired coordinates in the X and Y fields under **Point 1**.
4. Select the second corner of the region (**Point 2**) using the mouse or keyboard as...
5. Within the selected region, arrows representing the vector's direction are plotted on a grid. To specify the density of arrows in the region, enter an **X Division** and a **Y Division** for the grid. Higher values result in a more dense distribution of arrows; lower values a less dense distribution. In general, accept the default.

6. Choose **Execute**. A progress bar appears as the simulator computes the value of the vector quantity within the selected region. Afterwards, a window appears with fields that control the arrow plot's appearance.

From this point on, the **Post/Plane/Arrow Region** command operates like the **Post/Plane/Arrow** command.

**Post/Plane/Arrow Display**

Use **Post/Plane/Arrow Display** to redisplay the arrow plot that was plotted with the last **Post/Plane/Arrow** or **Post/Plane/Arrow Region** command. After you choose the command, a progress bar appears in the upper-left corner of the screen as the arrow plot is generated.
The **Post/Plane/Max-Min** command is customarily used to display maximum and minimum values (and their coordinates) of the field quantity in the top register for the entire plane. You can also use this command to compute and display the maximum and minimum values contained in a zoomed-in area.

1. To display the maximum and minimum values of the quantity in the top plane register:
   - Use the **Calc/Plane** commands to load a field quantity into the top register of the plane calculator. If desired, manipulate that quantity to derive some other quantity.
   - Choose **Post/Plane/Max-Min**. A menu displaying the geometry and the x- and y-coordinates of the maximum and minimum values of the quantity in the top plane register appears.
   - Choose **Extremes** to have the system display the labels **Max** and **Min** on the corresponding points in the geometric model.
   - Choose **Return** to end the command.

2. To compute and display the maximum and minimum values for a zoomed-in area:
   - Choose **Post/Plane/Max-Min**. The following window appears:
2. Choose **Zoom** and use the standard zoom procedure to mark your area of interest and magnify it.

3. Choose **Compute Max/Min** to calculate and display the maximum and minimum values and their coordinates within the region defined by the zoom window.

To return to the original display of the entire problem space:

1. Choose **Fill** to display the entire problem area within the viewing space.
2. Choose **Compute Max/Min** to calculate and display the maximum and minimum values and their coordinates for the entire plane.

**Post/Plane/Value**

Choose **Post/Plane/Value** to do the following:

- Display the value of a plotted field quantity at selected points in the problem region. If a scalar field quantity was plotted, this command displays its magnitude at the selected point. If a vector field quantity was plotted, this command displays the magnitude of each component of the vector at the selected point.
- Enter this field value into the top register of the number calculator, allowing you to use it in calculations of other derived quantities.

To display field values at selected points:

1. Plot the desired field quantity.
   - Use the **Post/Plot** command to plot common field quantities.
   - Use the **Post/Plane** and **Calc/Plane** commands to plot derived field quantities.
2. Choose **Post/Plane/Value**. The following window appears:
3. Select the point whose field values are to be displayed.
   - To select the point with the mouse:
     a. Choose Mouse. The Plane Value window disappears.
     b. Click the left mouse button on the desired point. The window reappears
        with the point's coordinates displayed in the \( x \) and \( y \) fields.
   - To select the point with the keyboard, enter the desired coordinates in the \( x \) and \( y \) fields.
      The value of the field quantity at the point is then displayed in the Value field.
4. To store the field value in the top register of the number calculator, set Enter in
   number calculator to Yes.
5. Repeat steps 3 and 4 to continue viewing field quantities and loading them into the
   number calculator.
6. To exit the command, choose Return.
Choose **Post/BH Examine** to do the following:

- View the **B** and **H** values of points that you select.
- Select points in nonlinear materials where saturation levels are to be plotted via the **Post/BH Plot** command. This is the first step in determining saturation levels in the nonlinear materials in your model.

The following window appears when you select this command:

> To select BH-curve points:
> 1. Choose **Post/BH Examine**.
> 2. Choose **Enter Data Points**. The cursor changes to crosshairs.
> 3. Click the left mouse button on the desired points in the geometric model. As you...
select each point, a cross appears on its location and it is assigned a number.

**Note:** To determine saturation levels using the Post/BH Plot command, you must select points in objects that were assigned nonlinear materials.

4. When you are finished choosing points, click the right mouse button. The x- and y-coordinates of the points appear in the display area on the left side of the window.
5. Optionally, choose Display Data Points to display the numbers assigned to each point you selected in the geometry.
6. To compute \( B \) and \( H \) at each point, choose Compute BH Data. The magnitude of \( B \) and \( H \) are displayed below the point’s coordinates.
7. Choose Return to exit the command.

**Things to Consider**

By default, the mouse is set to Grid Snap and Object Snap, which means that you can only select points that lie on the grid or on object vertices. To more accurately select points inside objects, use the Global/Defaults command to:

- Change the grid spacing to produce a more dense grid.
- Set Keyboard Entry to Yes, enabling you to enter the coordinates of the desired points using the keyboard instead of the mouse.
- Set Object Snap and Grid Snap to No, leaving the mouse in “free mode” where any point in the problem space may be selected.
(Magnetostatic problems with nonlinear materials only.)

When the B-field in a nonlinear material rises above the “knee” of the material’s BH-curve, further increases in $H$ result in relatively small increases in $B$. At this point, the material is said to be magnetically saturated. Choose this command to analyze the level of saturation in nonlinear materials.

To analyze saturation levels:
1. Select points in the desired nonlinear materials using Post/BH Examine.
2. Choose Post/BH Plot. The BH-curves associated with the materials at those points are then plotted, as shown below. Each point is labeled on the BH-curves, enabling you to see whether the material is magnetically saturated at those locations. You may need to zoom into the plot using the Plot Options/Zoom In command to more accurately see where each point lies on the curve.

To change the way in which BH-plots are displayed, use the Plot Options commands.
Viewing Nonlinear Permanent Magnet Curves

(Nonlinear Permanent Magnets only.)

Once you have defined the nonlinear BH-curve in the magnetization direction of permanent magnets, the 2D Post Processor linearizes the material model.

In the magnetization direction, the nonlinear BH-curve is substituted by a straight line whose slope is equal to the slope of the nonlinear BH-curve at H=0 and crosses the B axis at the same point where the nonlinear curve crosses it. In the direction perpendicular to the magnetization, the curve is substituted by a straight line that intersects the origin.

> To observe the BH-curve of a nonlinear permanent material:

1. Choose **Post/BH Examine**. The **Examine BH Data** window appears.
2. Choose **Enter Data Points** and use the mouse to specify points within the object with a nonlinear magnet.
3. Optionally, choose **Display Data Points** to view the selected points in the viewing window.
4. Choose **Compute BH Data**. The values of $\text{mag}(B)$ and $\text{mag}(H)$ for each point appear below the coordinates of the selected points.
5. Choose **Return**. The window closes.
6. Choose **Post/BH Plot**. The points specified in the **Examine BH Data** window are plotted on both linearized BH-curves. These curves correspond to the parallel and perpendicular directions with respect to the original magnetization.
7. Optionally, choose **Plot Options** to modify the view of the plot.
8. Choose **Return** to close the window and return to the 2D Post Processor.
Maxwell 2D — Calc Menu

Calc Menu

The Calc menu provides you with access to a set of calculators that allow you to display, manipulate, and analyze field solutions. Use it to perform mathematical operations on the computed field quantities to aid the analysis of your model.

Calc Commands

The calculators accessible from the Calc menu are:

- **Number**: Provides access to a calculator that allows you to manipulate single pieces of data.
- **Line**: Provides access to a 1D calculator that allows you to manipulate field quantities over lines.
- **Plane**: Provides access to a 2D calculator that allows you to manipulate field quantities over a plane or surface.

The Calc menu is accessed from the fields Post Processor.
Calculators

Several solution calculators are available in the Post Processor.

Number Calculator

The number calculator is typically used to perform operations on the numerical results of integrations and other calculator operations. It also supports a stack of registers, with each able to hold a single scalar or vector value. The result of integrations may be placed in the number calculator.

Line Calculator

This calculator is typically used to integrate a field quantity over a path or to plot a special quantity of interest as a function of distance along a line. The line calculator supports a stack of line registers, with each line register holding the portion of a field quantity that lies on a line.

To use the line calculator, follow these steps:

1. Create a line register by specifying which line segment to associate with the register. Do so using the Post/Line/Entry command. The new register goes to the top of the stack.
2. Load field values into a line register by mapping those values from the top register of the plane calculator. Do so by choosing Calc/Line/value in the line calculator.

For example, assume that the top register of the plane calculator contains the E-field. Choosing Calc/Line/value causes the portion of the E-field that lies on the desired line to be loaded into the line register.
Plane Calculator

This calculator is typically used to integrate a field quantity over a cross-section or to display a vector, shaded, or contour plot of a derived field. The plane calculator supports a stack of plane registers, with each able to hold the portion of a field quantity that lies in the 2D cross-section that is modeled.

For example, to compute the total energy, $U$, associated with an electric field,

$$U = \frac{1}{2} \int_{vol} E \cdot D \, dv$$

you can:

1. Load the E-field into the top register.
   
   Register 1 \( E(x) \ E(y) \ 0 \)

2. Load the D-field into the top register.
   
   Register 1 \( D(x) \ D(y) \ 0 \)
   Register 2 \( E(x) \ E(y) \ 0 \)

3. Take the dot product of the two fields. This is proportional to the energy density field.
   
   Register 1 \( (D(x) \cdot E(x) + D(y) \cdot E(y) + 0 \cdot 0) \)

4. To obtain the total energy, you must integrate the energy density field and divide the results by two.

The plane calculator contains commands for performing such tasks.

Plotting

Any field quantity that is in the top register of the plane or line calculator can be displayed on screen.

Plotting Over a Plane

To plot a quantity that is in the top register of the plane calculator, use the Post/Plane commands. These commands allow you to display shaded plots, contour plots, and vector plots.
Plotting Along a Line
To plot a quantity that is in the top register of the line calculator, use the Post/Line commands. These commands allow you to graph a field quantity as a function of position over a line.

Direct Plotting
Basic field quantities (such as the H-field and E-field) can be plotted directly using the Post/Plot command. This command does not require you to load the desired field quantity into a register before plotting.

Reading and Writing
In some cases, it is desirable to superimpose two sub-solutions to obtain the total solution for a problem. For example, assume that a device has two sources. With one source turned on, the field is tangential to a particular boundary; with the other source turned on, the field is perpendicular to that same boundary.

> To fully model this problem:
1. Solve the problem with one source turned on and the other turned off.
2. Load that solution into a register and write it out to a disk file.
3. Solve the problem again with the first source turned off and second source turned on.
4. Load the new solution into a register, read in the first solution from disk, and add the two registers.

Use the read and write options associated with the calculators to superimpose sub-solutions.
Maxwell 2D — Calc Menu

Registers

The top half of the calculator menu displays the stack of registers associated with the calculator. Each register is capable of holding an entire field quantity. No registers are created until you load in a field quantity; therefore, this region of the calculators is initially blank.

- Fields listed as Vector quantities are those that have both direction and magnitude at each point in space. The x-, y-, and z-components (or r-, \( \phi \)- and \( z \)-components) of these vector quantities are stored in the register.
- Fields listed as Scalar quantities have a magnitude only.
- Registers that begin with an R contain real quantities.
- Registers that begin with a C contain complex quantities.

If the number of registers to be displayed exceeds the available display space for one page of them, you can view the additional pages by doing the following:

- Select Next to display the next page of registers.
- Select Prev to return to the previous page of registers.

Calculator Commands

The bottom portion of the calculator menu contains the commands associated with the calculator. In many cases, there are more commands than can fit in the bottom of the menu. Choose Next or Prev to scroll through the list of available commands.

Displaying Other Calculators

From the plane, line, or number calculators, you can access any other calculator. To access the commands and registers associated with a different calculator, choose the Plane, Number, or Line options that are located on the right side of the calculator.
Choose **Calc/Plane** to manipulate field data over the model's 2D cross-section. The following window appears:

The stack of registers associated with the plane calculator is displayed in the top portion of the screen. Each register can store a field quantity. The commands associated with the plane calculator are displayed in the bottom portion of the screen.

While in the other calculators, you can access the plane calculator by choosing the **Plane** option located on the right side of the calculator.
Material

The **Material** command allows you to multiply or divide a field quantity by a material property such as the relative permittivity, conductivity, or relative permeability. The currently selected parameter field solver determines which field quantities and material properties are available when the **Material** command is selected.

At each triangle in the mesh, the field quantity is multiplied or divided by the value of the selected material property. This takes the different material attributes of each object in the model into account.

> To multiply or divide a field quantity by a material property:

1. Load the desired field quantity into the top register of the calculator.
2. Choose **Material**. The **Material Operations** window appears with one or more material property fields. (The properties that are available for each problem type are discussed under **Loading Field Data** in the sections for the different field quantities.)
3. Select the desired material property.
4. Do one of the following:
   - To multiply the quantity in the top register by the selected material property, toggle **Multiply** to **Yes**.
   - To divide the quantity in the top register by the selected material property, toggle **Multiply** to **No**.
5. Choose **Return** to complete the operation or **Cancel** to cancel the action.

There must be a vector field quantity in the top register for this command to be operable.
Loading Field Data

Initially, all registers are empty. To load a field quantity into a new plane register, choose the desired quantity from the list of calculator commands. The field quantities that can be loaded into the plane calculator depend on which field solver you selected, and are listed in the following sections. The selected field solver also determines what materials are available to use in your calculations. The available materials for each solver are also listed in the following sections.

Magnetostatic and Transient Field Quantities

The following field quantity selections are available if a magnetostatic or transient motion simulation has been performed. When you select one, the calculator loads it into the top register.

- **B_Vector**: The magnetic flux density.
- **H_Vector**: The magnetic field strength.
- **A_Vector**: The magnetic vector potential, \( A_Z \) or \( A_\Phi \).
- **J_Vector**: Loads the current density vector \( J \).
- **energy (linear)**: For linear materials, loads the actual energy density defined by \( \frac{1}{2} B \cdot H \). This is the apparent energy for nonlinear materials.
- **app energy (nonlinear)**: For nonlinear materials, loads the energy density given by \( \int B \cdot dH \).
- **coenergy (nonlinear)**: For nonlinear materials, loads the energy density given by \( \int B \cdot dB \).
- **energy (nonlinear)**: For nonlinear materials, loads the energy density given by \( H \cdot dB \).

The following material properties are available through the Material command for magnetostatic problems:

- For problems that contain only linear materials, the permeability (\( \mu \)) is the only available material property.
- For problems that contain nonlinear materials, both the permeability (\( \mu \)) and the differential permeability (\( \text{diff } \mu \)) are available. The differential permeability is equal to...
the instantaneous slope of the BH-curve, as shown here:

\[ \frac{dB}{dH} = \text{instantaneous slope of nonlinear BH-curve} \]

\[ \mu_r = \text{slope of linear BH-curve} \]
Electrostatic Field Quantities

The following field quantities are available in the Plane Calculator if an electrostatic simulation has been performed. When you select one, the calculator loads it into the top register.

- **voltage**  The electric potential, $\phi$.
- **E_Vector**  The electric field.
- **D_Vector**  The electric flux density.
- **energy**  The stored electric field energy density.

Permittivity (\textit{epsilon}) is the only material property that is available for the Material command.
Eddy Current Field Quantities

The following field quantities are available if an eddy current simulation has been performed:

- **A_Vector**
  - The magnetic vector potential, \( A_Z \) or \( A_\Phi \).

- **B_Vector**
  - The magnetic flux density.

- **H_Vector**
  - The magnetic field strength.

- **J_Vector**
  - The current density. Select one of the following:
    - **Displacement** — Loads the component of the current density due to displacement currents.
    - **Eddy** — Loads the component of the current density due to eddy currents.
    - **Source** — Loads the component of the current density due to source currents.
    - **Total** — Loads the total current density, which is the sum of the displacement, source, and eddy current components.

- **energy (linear)**
  - For linear materials, loads the actual energy density defined by \( \frac{1}{2} \mathbf{B} \cdot \mathbf{H} \). This is also the apparent energy density for nonlinear materials.

- **app energy (nonlinear)**
  - \( \frac{1}{2} \mathbf{B} \cdot \mathbf{H} \).

- **coenergy (nonlinear)**
  - For nonlinear materials, loads the energy density given by \( \int \mathbf{B} \cdot d\mathbf{H} \).

- **energy (nonlinear)**
  - For nonlinear materials, loads the energy density given by \( \int \mathbf{H} \cdot d\mathbf{B} \).

- **power**
  - Loads the ohmic loss density (time-averaged). It does not include core losses.

- **EM_loss**
  - Loads the total power loss density from a coupled eddy project which includes both ohmic and core loss (if it was set up as an executive parameter). Core loss is an executive parameter. EM loss will always include ohmic loss by default.

Conductivity \((\text{sigma})\), permeability \((\text{mu})\), and permittivity \((\text{epsilon})\) material properties are the material properties available through the **Material** command.
Maxwell 2D — Calc Menu

**DC Conduction Field Quantities**

The following field quantities are available if a DC conduction simulation has been performed:

- **voltage** Loads the electric potential into the top register.
- **E_Vector** Loads the electric field into the top register.
- **J_Vector** Loads the conduction current into the top register.
- **energy** Loads the stored electric field energy density into the top register.
- **power** Loads the power density into the top register.

Conductivity (\( \sigma \)) is the only material property available from the Material command. There must be a vector field quantity in the top register for this command to be operable.

**AC Conduction Field Quantities**

The following field quantities are available if an AC conduction simulation has been performed. When you select one, the calculator loads it into the top register.

- **voltage** The electric potential.
- **E_Vector** The electric field.
- **D_Vector** The electric flux density.
- **J_Vector** The current density. Select one of these current density components:
  - **Displacement** — Loads the component of the current density due to displacement currents.
  - **Conduction** — Loads the component of the current density due to conduction currents.
  - **Total** — Loads the total current density, which is the sum of the displacement and conduction components.
- **energy** Loads the stored electric field energy density into the top register.
- **power** Loads the power density (time-averaged) into the top register.

The permittivity (\( \varepsilon \)) and the conductivity (\( \sigma \)) are the material properties that are available through the Material command.
### Eddy Axial Field Quantities

The following field quantities are available if an eddy axial simulation has been performed:

<table>
<thead>
<tr>
<th>Field Quantity</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>H_Vector</strong></td>
<td>Loads the eddy current strength into the top register.</td>
</tr>
<tr>
<td><strong>J_Vector</strong></td>
<td>Loads the current density into the top register. Select one of the following:</td>
</tr>
<tr>
<td></td>
<td>- <strong>Displacement</strong> — Loads the component of the current density due to displacement currents.</td>
</tr>
<tr>
<td></td>
<td>- <strong>Conduction</strong> — Loads the component of the current density due to conduction currents.</td>
</tr>
<tr>
<td></td>
<td>- <strong>Total</strong> — Loads the total current density, which is the sum of the displacement and conduction current density.</td>
</tr>
<tr>
<td><strong>energy</strong></td>
<td>Loads the stored electric field energy density into the top register.</td>
</tr>
<tr>
<td><strong>power</strong></td>
<td>Loads the power density (time-averaged) into the top register.</td>
</tr>
</tbody>
</table>

The permittivity (\( \varepsilon \)), permeability (\( \mu \)), and conductivity (\( \sigma \)) are the material properties that are available through the **Material** command. There must be a vector field quantity in the top register for this command to be operable.

### Thermal Field Quantities

The following field quantities are available if a thermal simulation has been performed:

<table>
<thead>
<tr>
<th>Field Quantity</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Temperature</strong></td>
<td>Loads temperature in degrees Celsius.</td>
</tr>
<tr>
<td><strong>Flux</strong></td>
<td>Loads the thermal flux density in W/m².</td>
</tr>
<tr>
<td><strong>Grad_T</strong></td>
<td>Loads the thermal gradient in degrees Celsius/m.</td>
</tr>
<tr>
<td><strong>EM_Loss</strong></td>
<td>Loads the total loss density from a coupled eddy project which includes both ohmic and core loss if it was set up as an executive parameter. Core loss is an executive parameter. EM loss will always include ohmic loss by default.</td>
</tr>
</tbody>
</table>
Register Operations

The following Register operations are available in the plane calculator:

- **name**: Assigns a name to a plane register.
- **push**: Copies the first register and moves all others down by one. The new register is linked to the same cutplane or rectangle as the one that was copied.
- **pop**: Moves all registers up by one, throwing the top register away.
- **exchange**: Exchanges the first and second registers.
- **roll**: Moves all registers up by one, rolling the bottom register to the top.
- **clear**: Erases all registers in the plane calculator.

Scalar Operations

Scalar operations available in the plane calculator include:

- **constant**: Assigns a constant value to all points in the plane.
- **integrate**: Integrates the scalar quantity in the top register over cross-section and places the result in the number calculator.
- **rz_intgrl**: (*RZ models.*) Integrates the scalar quantity in the top register over the volume created by revolving the axisymmetric cross-section 360° about the z-axis.
- **vec_x**: (*XY models.*) Creates a vector field by assigning the scalar field in the top register to be the \( x \)-component.
- **vec_y**: (*XY models.*) Creates a vector field by assigning the scalar field in the top register to be the \( y \)-component.
- **vec_r**: (*RZ models.*) Creates a vector field by assigning the scalar field in the top register to be the \( r \)-component.
- **vec_phi**: (*RZ models.*) Creates a vector field by assigning the scalar field in the top register to be the \( \phi \)-component.
- **vec_z**: Creates a vector field by assigning the scalar field in the top register to be the \( z \)-component.
- **grad**: Takes the gradient of the quantity in the top register of the calculator.
Vector Operations

Vector operations available in the plane calculator include:

- **vec_cons**: Allows you to assign a constant vector value to all points on the plane.
- **normal**: Calculates the unit vector field that is normal to the plane associated with the top register, pushes the top register down in the stack, and loads in the normal field.
- **magnitude**: Calculates the magnitude of the vector field in the top register.
- **scalar_x**: \((XY \text{ models.})\) Takes the x-component of the vector in the top register and creates a scalar field.
- **scalar_y**: \((XY \text{ models.})\) Takes the y-component of the vector in the top register and creates a scalar field.
- **scalar_r**: \((RZ \text{ models.})\) Takes the r-component of the vector in the top register and creates a scalar field.
- **scalar_phi**: \((RZ \text{ models.})\) Takes the \(\phi\)-component of the vector in the top register and creates a scalar field.
- **scalar_z**: Takes the z-component of the vector in the top register and creates a scalar field.
- **div**: Takes the divergence of the vector field in the top register.
- **curl**: Takes the curl of the vector field in the top register.

Transient Operations

If you are in EMpulse, the following operations are available:

- **A_Vector**: Loads the magnetic vector potential, \(A_Z\) or \(A_\Phi\), into the top register.
- **B_Vector**: Loads the magnetic flux density into the top register.
- **H_Vector**: Loads the magnetic field strength into the top register.
- **J_Vector**: Loads the conduction current into the top register.
- **Materials**: Multiplies the quantity in the top stack by a material property value.
- **Energy**: Loads the stored field energy into the top register.
- **Coenergy**: For nonlinear materials, loads the energy given by \(\int_{V} B \cdot (dH) \, (dV)\).
- **App Energy**: For linear materials, loads the actual energy defined by \(\frac{1}{2} \int_{V} B \cdot H \, dV\). This is the apparent energy for nonlinear materials.
General Operations

General operations available in the plane calculator include:

- **add**: Adds the top two plane registers.
- **conjugate**: (Eddy current, AC Conduction, Eddy Axial.) Takes the complex conjugate of the field quantity in the top register. If an imaginary number is given by \( C = A + jB \), its complex conjugate is given by \( C^* = A - jB \).
- **multiply**: Multiplies the scalar fields in the top two registers. If both fields are vectors, components are multiplied together; that is, the x-components are multiplied together, as are the y- and z-components.
- **divide**: Divides the second register by the first register. The quantity in the first register must be a scalar, and division by zero is not allowed.
- **r_multiply** (RZ models.) Multiplies the quantity in the top register by r.
- **r_divide** (RZ models.) Divides the quantity in the top register by r.
- **dot**: Takes the dot product of the vector fields in the top two registers.
- **cross**: Takes the cross product of the vector fields in the top two registers (second register cross top register).
- **smooth**: Does geometric averaging inside each material to arrive at a smoothed version of the vector in the top register.
- **read**: Reads in a register that has been previously written to disk. The mesh associated with the register being read in must be exactly the same as the mesh currently in memory.
- **write**: Writes the contents of the top register to the disk file that you specify. The system adds an extension (.reg for binary files and .arg for ASCII files) and saves the file in the current project directory. Use the ASCII option if you plan to manipulate the data using other software tools. Use the binary option to save the data in a more compact fashion.
- **export**: Saves the field values comprising the top stack register in a file that you name.
- **decompose**: Decomposes complex quantities in the form \( a + jb \) into their real, imaginary, magnitude, or phase equivalents.
Phase

*(Eddy Current, AC Conduction, Eddy Axial)*

The field quantities downloaded into the plane calculator represent instantaneous values of time-varying quantities:

\[ A(x, y, z, t) = A(x, y, z) \cos(\omega t + \Theta(x, y, z)) \]

where:

- \( \omega \) is the angular frequency at which the quantities are oscillating, specified during the solution.
- \( \Theta(x,y,z) \) is the phase angle (the offset from a cosine wave that peaks at \( t=0 \)).

Enter the phase of the field quantity in the top register in the **Phase** field.

**Plotting the Contents of the Top Plane Register**

To display the field quantity that is in the top register of the plane calculator, do one of the following:

- Choose **Post/Plane/Contour** to display a contour plot. The field in the top register must be a scalar field.
- Choose **Post/Plane/Shade** to display a shaded plot. The field in the top register must be a scalar field.
- Choose **Post/Plane/Arrow** to display an arrow plot. The field in the top register must be a vector field.

In each case, the post processor plots whatever quantity is in the top register of the plane calculator.

Common field quantities such as the magnetic field strength (H-field) can be plotted using **Post/Plot** and do not need to be loaded into the plane calculator.
Calc/Plane/Export

Use this command to save the field values comprising the top stack register in a file that you name. The saved file also contains the \( x \) and \( y \) coordinates (cartesian models) or \( r \) and \( z \) coordinates (axisymmetric models) which define the bounded plane space containing the field values.

To export a register:
1. Choose Export. The following window appears:

2. Use one of these procedures to set up the export file:
   - Choose Use Input File to read in a list of grid coordinates that define the extent of the plane space. Type the name of the input file in the File Name field.
   - Choose Use Uniform Grid to let the software calculate the coordinates to define a uniform grid which encloses the plane space.
     a. Enter the grid coordinates (\( xy \) or \( rz \)) in the min and max coordinates entry fields to define the extent of the plane space.
     b. Enter the distance between grid points in the Grid spacing entry field. The units are the same as those used for the model. If you omit the grid spacing entry, an error message appears when you try to exit the menu.
     c. Enter the name of the file in which you want to save the export data in the

   ![Export Plane Register Window](image)
The Decompose command, which is available for field data produced by complex solvers such as AC Conduction and Eddy Current, decomposes complex quantities in the form $a + jb$ into their real, imaginary, magnitude, or phase (in radians) components. As with other register operations, this command operates on the register at the top of the stack.

The results of the Decompose operation for the four components are in the table below:

<table>
<thead>
<tr>
<th>Component</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Real</td>
<td>$a$</td>
</tr>
<tr>
<td>Imaginary</td>
<td>$b$</td>
</tr>
<tr>
<td>Magnitude</td>
<td>$\sqrt{a^2 + b^2}$</td>
</tr>
<tr>
<td>Phase (radians)</td>
<td>$\tan^{-1}\left(\frac{b}{a}\right)$</td>
</tr>
</tbody>
</table>

Note: If you omit the Output File Name entry, the following error message appears when you try to exit the menu:

Invalid output file name
To decompose a complex register:

1. Choose **Decompose**. The following window appears:

   ![Decompose Complex Register Window](image)

   2. Select **Real**, **Imaginary**, **Magnitude**, or **Phase** to designate the target data type for the decompose operation. The type that you select appears to the right of the **Decompose into** text below the four selections.

   3. Select **Execute** to execute the command. A display of the selected type equivalent of the data appears on the top of the stack, replacing the display of the original complex register.

   The following figure shows the result of decomposing a complex A_Vector into its real value. For comparison, the decomposed version of the register appears above the original complex register. The real version of the register may appear to be in complex form, but it is in fact a real value in the form of \( a + j0 \), which is the format that the complex solvers use:

   ![Decomposed Register](image)
Choose **Calc/Number** to display the number calculator. The number calculator allows you to manipulate single values or single vectors. The results of volume, plane, and line integrations can be saved in this calculator:

While in the other calculators, you can access the number calculator by selecting the **Number** options (on the right side of the calculator).

The stack of registers associated with the number calculator is displayed in the top part of the screen. The commands associated with the number calculator are displayed in the bottom part.
Topics:

Calc/Number
  Register Operations
  Scalar Operations
  Vector Operations
  General Operations
  Phase
Calc/Line
  Creating Line Registers
  Register Operations
  Scalar Operations
  Vector Operations
  General Operations
  Phase
  Displaying the Field in the Top Line Register

Maxwell 2D — Calc Menu

Register Operations

Register operations available in the number calculator include:

- **push**: Copies the first register and moves all others down by one.
- **pop**: Moves all number registers up by one, throwing the top register away.
- **exchange**: Exchanges the first and second number registers.
- **roll**: Moves all registers up by one, rolling the top register to the bottom.
- **clear**: Erases all registers in the number calculator.

Each number register contains a single vector or scalar value.

Scalar Operations

Scalar operations available in the number calculator include:

- **constant**: Places a constant in the top register.
- **vec_x**: *(XY models.)* Creates a vector using the scalar in the top register as the x-component.
- **vec_y**: *(XY models.)* Creates a vector using the scalar in the top register as the y-component.
- **vec_r**: *(RZ models.)* Creates a vector field by assigning the scalar field in the top register to be the r-component.
- **vec_phi**: *(RZ models.)* Creates a vector field by assigning the scalar field in the top register to be the φ-component.
- **vec_z**: Creates a vector using the scalar in the top register as the z-component.
Vector Operations

Vector operations available in the number calculator include:

`vec_cons` Places a vector value in the top register.

`dot` Takes the dot product of the vectors in the top two registers.

`cross` Takes the cross product of the vectors in the top two registers.

`scalar_x` *(XY models.)* Takes and creates a scalar field using the x-component of the vector in the top register.

`scalar_y` *(XY models.)* Creates a scalar field using the y-component of the vector in the top field.

`scalar_r` Creates a scalar field using the r-component of the vector in the top register.

`scalar_phi` Creates a scalar field using the $\phi$-component of the vector in the top register.

`scalar_z` Creates a scalar field using the z-component of the vector in the top register.
General Operations

The following general operations are available in the line calculator:

- **add**: Adds the top two registers.
- **multiply**: Multiplies the top two registers.
- **divide**: Divides the second register by the first register. The quantity in the first register must be a scalar, and division by zero is not allowed.
- **invert**: Inverts the quantity stored in the top register.
- **magnitude**: Takes the magnitude of vector in the top register.
- **power**: Takes the quantity in the top register to the desired power.
- **write**: Writes the contents of the top register to the disk file that you specify. The system adds an extension (.reg for binary files and .arg for ASCII files) and saves the file in the current project directory. Use the ASCII option if you plan to manipulate the data using other software tools. Use the binary option to save the data in a more compact fashion.
- **read**: Reads in a register that has been previously written to disk. The mesh associated with the register being read in must be exactly the same as the mesh currently in memory.

Phase

*(Eddy Current, AC Conduction, Eddy Axial)*

Displays the phase of the quantity downloaded into the number calculator.
Choose **Calc/Line** to manipulate field data over a line. The following window appears:

The line calculator is essentially a linear version of the two-dimensional plane calculator. The stack of registers associated with the line calculator is displayed in the top portion of the screen. Each register can store the portion of a field quantity that lies along a particular line. The commands associated with the line calculator are displayed in the bottom portion of the screen.

While in the other calculators, you can access the line calculator by choosing the **Line** option located on the right side of the calculator.
Creating Line Registers

The general procedure for creating a line register and loading it with a field value is as follows:

1. Define a line segment using **Post/Line/Define**. The line segment can be a polyline, arc, or object edge.
2. Choose **Post/Line/Entry** to link that line segment to a line register.
3. Choose **Calc/Plane** and load the quantity of interest into the top register of the plane calculator.
4. Choose **Calc/Line/value** to map field values from the top register of the plane calculator to the line register that you just created.

The register that is created is linked to one specific line in the problem region. When a field quantity is mapped to a line register, the system takes the values of that field along the path and stores them in the register.

<table>
<thead>
<tr>
<th>Register Operations</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>enter</strong></td>
</tr>
<tr>
<td><strong>name</strong></td>
</tr>
<tr>
<td><strong>push</strong></td>
</tr>
<tr>
<td><strong>pop</strong></td>
</tr>
<tr>
<td><strong>exchange</strong></td>
</tr>
<tr>
<td><strong>roll</strong></td>
</tr>
<tr>
<td><strong>clear</strong></td>
</tr>
</tbody>
</table>
Scalar Operations

Scalar operations available in the line calculator include:

- **constant**: Assign a constant value to all points along the line.
- **vec_x**: (XY models.) Creates a vector using the scalar in the top register as the x-component.
- **vec_y**: (XY models.) Creates a vector using the scalar in the top register as the y-component.
- **vec_r**: (RZ models.) Creates a vector field by assigning the scalar field in the top register to be the r-component.
- **vec_phi**: (RZ models.) Creates a vector field by assigning the scalar field in the top register to be the φ-component.
- **vec_z**: Creates a vector using the scalar in the top register as the z-component.

Vector Operations

Vector operations available in the line calculator include:

- **vec_cons**: Assign a constant vector value to all points on the line.
- **scalar_x**: (XY models.) Creates a scalar field using the x-component of the vector in the top register.
- **scalar_y**: (XY models.) Creates a scalar field using the y-component of the vector in the top register.
- **scalar_r**: (RZ models.) Creates a scalar field using the r-component of the vector in the top register.
- **scalar_phi**: (RZ models.) Creates a scalar field using the φ-component of the vector in the top register.
- **scalar_z**: Creates a scalar field using the z-component of the vector in the top register.
General Operations

General operations available in the line calculator include:

- **add** Adds the fields in the top two registers.
- **multiply** Multiplies the fields in the top two registers. At least one of the two registers must contain a scalar field. At each point, the scalar value in one register is multiplied by the scalar or vector in the other.
- **divide** Divides the second register by the first register. The quantity in the first register must be a scalar, and division by zero is not allowed.
- **magnitude** Takes the magnitude of the vector quantity stored in top register.
- **value** Maps field values from the top plane register to the top line register. Performs the same function as **Post/Line/Value**.
- **tangent** Takes the component of the vector field in the top plane register that is tangential to the path associated with the top line register.
- **normal** Takes the component of the vector field in the top plane register that is normal to the path associated with the top line register.
- **integrate** Integrates the scalar quantity in the top line register over the appropriate line and places the result in the number calculator. A running total of the integration along the path is stored in the top line register.
- **rz_intgrl** *(RZ models.)*Multiplies the scalar quantity in the top line register by \(2\pi r\).
- **write** Writes the contents of the top register to a specified file. The system adds an extension (.reg for binary files and .arg for ASCII files) and saves the file in the current project directory. Use the ASCII option if you plan to manipulate the data using other software tools. Use the binary option to save the data in a more compact fashion.
- **read** Reads in a register that has been previously written to disk. The mesh associated with the register being read in must be exactly the same as the mesh currently in memory.

Because each register is linked to a specific line in the problem region, it typically does
not make sense to add and multiply registers that are not linked to the same line.

**Note:** When no other registers exist when you map in a new field quantity, the system prompts you for the **Number** of points in line. If you accept the default of 100, the system explicitly stores values at 100 points along the line; values between points are interpolated. All subsequent lines receive the same number of points. The minimum number of points in a line is ten points.

**Phase**

*(Eddy Current, AC Conduction, Eddy Axial)*

The field quantities downloaded into the line calculator represent instantaneous values of time-varying quantities:

\[
A(x, y, z, t) = A(x, y, z) \cos(\omega t + \theta(x, y, z))
\]

where:

- \(\omega\) is the angular frequency \((2\pi f)\) at which the quantities are oscillating, specified during the solution.
- \(\theta(x,y,z)\) is the phase angle (the offset from a cosine wave that peaks at \(t=0\)).

> To download the instantaneous value of a field quantity at different points in its cycle:
1. Enter the phase angle, \(\theta\), in the **Phase** field. Phases are entered in degrees.
2. Choose **Value** to download the field quantity in the top register of the plane calculator onto the line.

**Displaying the Field in the Top Line Register**

To display the field quantity that is in the top register of the line calculator, choose **Post/Line/Plot**. A quantity versus distance plot is then displayed. If a vector is in the top register, plots of all three components are displayed.

Such plots for common field quantities can be displayed using **Post/Plot** and do not need to be loaded into the line calculator.
Introduction to Parametric Analysis

The Parametric Analysis module is an add-on package for Maxwell 2D that enables you to quickly and easily perform variational analysis of 2D models. It allows you to simulate design variations using a single model, instead of having to explicitly set up and solve a series of models as is necessary with other Maxwell 2D packages. The Parametric Analysis module lets you:

- Identify the basic design parameters that are to be varied during the simulation — such as geometric dimensions, material properties, excitations, and solution frequency.
- Select the quantities that are computed during the parametric solution — such as electric or magnetic fields, inductance, capacitance, current flow, force, torque, and soon. You can also define macros using the post-processing calculators to compute other quantities not listed above.
- Specify the values to which each design parameter is set during the solution, generally by sweeping them through a range of values. This lets you define a series of variations on the original, “nominal” model, each of which will be solved during the parametric sweep.

It then automatically sets the design parameters to the values you specified, and computes a solution for each variation on the nominal model. After the parameter sweep is completed, you can analyze the results using the module’s post-processing functions.
Nominal and Parametric Models

The Parametric Analysis module breaks a Maxwell 2D project down into two parts — a nominal model and a parametric model.

Nominal Model

To perform a parametric analysis, you must first create a nominal model. Essentially, it is an ordinary 2D model that has been parameterized, and serves as the model on which the parametric analysis is based.

The nominal model is created like any other 2D model, except that the design parameters that will be changed during a parametric sweep must be defined as variables while you are drawing the geometry, assigning material properties, and so forth. (For instance, any dimensions that will be varied must be defined using geometric constraints.) However, no variables need to be defined if you do not plan to run a parametric analysis on a model.

The nominal model's field and executive parameter solutions are computed independently of any parametric solutions, although both may use the same set of solution criteria. When computing solutions, the simulator uses the original values for all design variables that were specified when the model was created, ignoring any values they might be set to during a parametric sweep.

Parametric Model

The parametric model consists of a series of variations on the nominal model. In it, the design variables defined for the model are assigned the values you specified while setting up the parametric sweep. The number of variations that can be defined for a parametric model is limited only by your computing resources and imagination.

Each variant on the nominal model is known as a “parametric setup.” It represents the model that results when you set one or more of the variables you defined when creating the model to different values. During the solution process, field and executive parameter solutions are computed for each setup, and post-processing macros are executed using the results of the setup's field solution. You can then compare the results obtained for each setup to determine how each design change affects the model's performance. (Note that changing the values of more then one or two variables at a time may make it difficult to isolate the effects of an individual design variation.)
Accessing the Parametric Analysis Module

To access the Maxwell 2D Parametric Analysis module, open a project and run the simulator. Models created using earlier versions of Maxwell 2D can be used with the Parametric Analysis module, as can models created using the non-parametric version of the simulator.

The Parametric Analysis module is automatically loaded with the rest of the Maxwell 2D software. The same model types, field solvers, drawing package, material properties, and excitations are available for both versions of the simulator. The same field quantities can be computed, and the same post-processing functions can be used to examine solution data. However, the Parametric Analysis module contains additional functions related to performing a parametric analysis.
General Procedure

Follow the general procedure summarized below to set up a parametric model and perform a parametric sweep:

1. **Create model.** Define geometric constraints and functional material properties, boundary and source values.
2. **Set up executive parameter computations.** Identify matrix entries; define motion parameters; select post processing macros.
3. **Enter field and executive parametric solution options.** Set up parametric sweeps.
4. **Generate the nominal and/or parametric solutions.**
5. **View and plot parametric solution results.** Analyze nominal and parametric field solutions; view convergence and profile statistics.
Create the Model

First, you must create the model to which you will apply the parametric analysis.

> To create a parametric model:

1. Follow the general procedure for creating a 2D model. Identify the design parameters that serve as variables during the parametric solution as follows:
   - While drawing the model, define geometric constraints to identify the dimensions that will be varied.
   - While assigning materials, define functional material properties to identify the material properties (such as relative permittivity, conductivity, and so forth) that will be varied.
   - While setting up boundaries and sources, define functional boundaries and sources to identify the electromagnetic excitations that will be varied.
2. Set up any executive parameter computations as you would for an ordinary Maxwell 2D model, using the Setup Executive Parameters command. If you specified that a capacitance, inductance, impedance, conductance, or admittance matrix be computed, select the matrix entries that appear in the parametric spreadsheet using the Setup Executive Parameters/Select Matrix Entries command.

Set Up Solutions

Next, set up the various solutions that will be performed during the parametric sweep.

Field and Nominal Solutions

> To set up the model’s field and executive parameter solutions:

   - Enter the solution criteria — such as error tolerance for an adaptive analysis — using the Setup Solution/Options command.

These solution options control how the individual field and parameter solutions are generated during a parametric sweep. They also control how the nominal solution is generated.
Setup the Parametric Solution

Once the model and nominal solution options have been defined, specify the parametric solution options.

To set up the model’s parametric solution:
1. Choose **Setup Solution/Variables**.
2. Select the variables — frequency, geometric constraints, functional material properties, or functional boundary/source quantities — that will be assigned new values during the parametric sweep. Do so using the **Variable/Add** command.
3. Define the individual solutions that will be performed during a parametric sweep. To add solutions to the sweep, do one or both of the following:
   - Specify a range of values for the desired variables (a “data sweep”) using the **Data/Sweep** command.
   - Alternatively, add solutions using the **Edit/Insert Row** command, and then edit them individually.
4. Save your changes and exit via the **File/Exit** command.

Generate Solutions

Once the solution options have been defined, generate a solution for the model.

To generate a solution, do one or both of the following:
- Choose **Solve/Nominal Problem** to generate a solution for the model on which the parametric sweep is based. Ordinarily, you do not have to generate a nominal solution before performing a parametric analysis. However, it is required if you want to adaptively refine the finite element mesh prior to performing a parametric sweep, or if you want to define post-processing macros that will be executed during the sweep.
- Choose **Solve/Variables** to begin the parametric solution. The Maxwell 2D generates a solution for each parametric setup that’s been defined for your model.
Post Processing

When the solution process is complete, do the following to view the results:

- To analyze the field solution for the nominal problem on which the parametric sweep is based, choose Post Process/Nominal Problem.
- To view information on the various solutions computed during a parametric sweep and plot parametric solution results, choose Post Process/Variables.
- If you specified that field solutions were to be saved during the setup of a parametric sweep, choose Variables. From this window, you can select which field solution is to be viewed.
- To view complete executive parameter solution results, convergence and profile statistics for either the nominal or parametric solutions, use the Solutions, Convergence, and Profile commands.
Batch Processing

As an alternative to running Maxwell 2D interactively, use the software’s batch processing features to perform a parametric analysis. In order for batch mode to work properly, you must do the following for each model:

- Select the type of field to be computed.
- Create the geometric model, define material characteristics and set up boundaries. While doing so, define all variables (such as geometric constraints or functional boundary values) that you want to use during the parametric sweep.
- Request forces, torques and other quantities of interest.

**Note:** To use post-processing macros during a batch solution, generate a nominal solution for the problem and define the desired macros before running the batch job.

- Enter the desired solution parameters for both the nominal problem and the parametric sweep.

For an overview of the steps involved in setting up and solving a parametric model, see the General Procedure section earlier in this chapter.

Batch mode for the Parametric Analysis module operates in much the same way as batch mode for the other Maxwell 2D packages. The differences are described here.
Batch Mode for Workstations (UNIX)

To run the software in batch mode on a workstation, enter one of the following commands at the UNIX prompt:

- To generate a solution for the nominal problem, enter:
  
  \( \text{m2dfs} \ -\text{batch} \ \text{option} \ \text{projectname} \)

  where \text{option} is one of the following:

  \( \text{C} \) Solves for the model's capacitance matrix.
  
  \( \text{L} \) Solves for the model's inductance matrix.
  
  \( \text{Z} \) Solves for the model's impedance matrix.
  
  \( \text{y} \) Solves for the model's admittance matrix.
  
  \( \text{All} \) Solves for all requested circuit parameters.

- To generate a solution for the parametric sweep, enter:
  
  \( \text{m2dfs} \ -\text{batch} \ \text{Variables} \ \text{projectname} \)

  where \text{projectname} is the name and directory path of the Maxwell 2D project that you wish to solve. The \text{Variables} flag automatically executes parametric solutions for all requested circuit parameters. Use a script file to generate batch solutions for multiple projects.

Errors in Parametric Solutions

If a problem occurs during a parametric solution — such as the creation of overlapping objects when a geometric constraint is set to a new value — the solution for that parametric setup is aborted. However, Maxwell 2D continues to generate solutions for the project’s other parametric setups. Even though one or more solutions may be invalid, you will not lose the results from the rest of the parametric sweep. All solution errors are recorded in the batch log file.
Maxwell 2D — Parametric Analysis

Batch Mode for Personal Computers

Batch processing for parametric problems works differently in Microsoft Windows operating systems. To generate batch solutions for nominal models, use the Maxwell Batch Utility.

Note: You cannot directly generate batch solutions for parametric models in older Microsoft packages such as Windows for Workgroups. Instead, use a third-party batch processing utility. Enter the command to run the software as described below.

Batch Processing for Windows

To generate batch solutions in the Windows version of the software, do one or more of the following:

- Generate a solution for the nominal problem using the Microsoft Windows command shell.
- To generate a solution for the parametric problem using the Windows command shell, enter the following at the command prompt:

  ```
  path\m2dfs -batch Variables projectname
  ```

  where:

  - `path` is the drive and directory path where the Maxwell 2D executables are installed (for example, `c:\win32app\maxwell`).
  - `projectname` is the drive, directory path and name of the Maxwell 2D project that you wish to solve.

To generate solutions for multiple projects, create a batch file that can be run in the Windows command shell.
Maxwell 2D — Parametric Post Processing

Parametrics Post Processing

Maxwell 2D’s Parametric Analysis module invokes the Parametric Post Processor to analyze the solutions.

Parametric Postprocessor

The Parametric Post Processor has three commands for viewing and analyzing the completed solutions.

- To analyze the field solution for the nominal problem on which the parametric sweep is based, choose **Post Process/Nominal Problem**.
- To view information on the various solutions computed during a parametric sweep and plot parametric solution results, choose **Post Process/Variables**.
- If you specified that field solutions were to be saved during the setup of a parametric sweep, choose **Variables** to view the parametric spreadsheet. From this window, you can select which field solution is to be viewed.

This chapter describes how to access each of these post-processing functions.

Post Process/Nominal Problem

Choose **Post Process/Nominal Problem** to view, manipulate, and analyze the field solution associated with the nominal problem on which the parametric sweep is based (the functionality of this command is the same as the non-parametric **Post Process** command).
Maxwell 2D — Parametric Post Processing

Post Process/Variables

Choose Post Process/Variables to do access the parametric setup.

Parametric Setup

The parametric table allows you to do the following:

- Plot variable values against solutions, error, number of triangles, and so forth. This enables you to graphically display the results of the parametric solutions.
- View the following information for each parametric solution:
  - The parameters that were varied during the solution.
  - The executive parameters (forces, torques, matrix entries, and so forth) that were computed.
  - The energy, energy error, and the number of triangles in the finite element mesh.
  - The numeric output of any Post Processor macros that were executed during the solution process.

The following window appears when you choose this command. Each variable setup that was defined via the Setup Solution/Variables command is listed in the parametric spreadsheet (regardless of whether a solution has been generated for it).
Parametric Post Processor Tool Bar

The icons in the tool bar serve as shortcuts for executing various post processing commands. They are listed below:

- Plot/New
- Plot/Close
- Plot/Add Graphs
- Plot/Show Coordinates
- Plot/Format/Axes
- Plot/Format/Graphs
- Plot/Zoom In
- Plot/Zoom Out
- Plot/Fit All
- Data/Sort
- Edit/Deselect All
- Variables/Animate

To execute or view information on the commands in the tool bar:
- To execute a command, click on the appropriate button.
- To display a brief description of the command in the message bar, hold down the mouse button on the desired icon.
After a parametric sweep is completed, choose **Variables** from the Executive Commands window to view the following for each parametric setup:

- The field solution for a parametric setup, if it was saved during the parametric sweep.
- The parameters that were varied during the sweep.
- The solution energy, energy error and the number of triangles in the finite element mesh.
- The executive parameters (forces, torques, matrix entries, and so forth) that were computed during the sweep.
- The numeric output of any post-processor macros that were executed during the solution process.

Each variable setup that was defined via **Setup Solution/Variables** is listed in the parametric spreadsheet (regardless of whether a solution has been generated for it).
General Procedure for Viewing Field Solutions

To view a saved field solution:

1. Click on the button at the beginning of the row containing the desired parameter setup. (Those with field solutions are marked with a Y under Save Field Solution?) The row is highlighted, and a window similar to the one below appears listing the values of all variables:

2. Choose View Fields to view the field solution for the selected parametric setup. The 2D Post Processor appears.
3. Choose Return to Nominal Problem to return to the Executive Commands window.
Edit Menu

In the spreadsheet, use the Edit commands to perform the following tasks.

- Cut, copy, paste, and reset cell values.
- Deselect any selected cells.
- Insert and delete rows from the table.

The Edit menu appears as shown below:

![Edit Menu Table]

- **Edit** commands: Undo, Redo, Cut, Copy, Paste, Clear, Deselect All, Insert Row, Delete Row, Duplicate Row.
Edit Commands

The function of each Edit command is as follows.

- **Undo**: Reverses the effect of the last command.
- **Redo**: Cancels the effect of the last Undo command.
- **Cut**: Copies the values in the selected cells into the table cutbuffer, and resets the cells to their nominal values. (Their values in the nominal problem.)
- **Copy**: Copies the values in the selected cells into the table cutbuffer, but does not reset the cells.
- **Paste**: Pastes the values in the cutbuffer into the selected cells. If there is nothing in the table cutbuffer, this option is not enabled in the menu.
- **Clear**: Resets the selected cells to their nominal values.
- **Deselect All**: Deselects all selected cells.
- **Insert Row**: Adds another row above the row containing the selection, or at the bottom of the table.
- **Delete Row**: Deletes the selected row or rows.
- **Duplicate Row**: Duplicates the selected rows.

Most of these commands are executed on cells selected in the variable setup table.

**Edit/Undo**

Choose Edit/Undo to reverse the effect of the last command.

**Edit/Redo**

Choose Edit/Redo to re-perform the last action cancelled with the Edit/Undo command.
### Maxwell 2D — Edit Menu (Variable Table)

#### Edit/Cut

Choose **Edit/Cut** to copy the values of the selected cells to the internal clipboard. The cells are then reset to their nominal values, and remain selected.

#### Edit/Copy

Choose **Edit/Copy** to copy the values of the selected cells to the internal clipboard. The cells retain their values, and remain selected.

#### Edit/Paste

Choose **Edit/Paste** to paste the values from the internal clipboard into the selected cells. You must have the same number of rows and columns selected as you did when you cut or copied the data.

#### Edit/Clear

Choose **Edit/Clear** to reset all of the selected cells to their nominal values. A message appears, asking if you are sure you want to clear the cells. If you choose **OK**, the cells are cleared, and remain selected.

#### Edit/Deselect All

Choose **Edit/Deselect All** to deselect all selected cells. The cells retain their values.

#### Edit/Insert Row

Choose **Edit/Insert Row** to add a row to the table.

- If you have no cells selected, this command adds a row to the bottom of the table.
- If you have one or more cells selected, this command adds a table row above the current selection.

The cells in the inserted rows take their values from the nominal problem.
Edit/Delete Row

Choose **Edit/Delete Row** to delete the selected rows from the table.

> To do this:
> 1. Select the rows you wish to delete. To select an entire row at once, click on its **setup** heading at the left edge of the table.
> 2. Choose **Edit/Delete Row**.

The selected cells are completely removed from the table.

Edit/Duplicate Row

Choose **Edit/Duplicate Row** to duplicate rows in the parametric table.

> To duplicate table rows:
> 1. Select the row to duplicate.
> 2. Choose **Edit/Duplicate Row**. The row appears above the selected one.
Variables Menu

In the spreadsheet, use the **Variables** commands to perform the following tasks:

- Add columns to the table for defined variables.
- Delete variable columns from the table.
- View all of the defined variables, their nominal values, and what part of the software uses each.
- View an animation which uses the setups to define each frame.

The **Variables** menu appears as shown below:

Variables Commands

The function of each **Variables** command is as follows.

- **Add** — Add a project variable as a column in the data table.
- **Delete** — Delete the selected project variable column.
- **View** — View all project variables and their current values.
- **Animate** — Display the model while enforcing constraint parameters row by row.
- **Rearrange Columns** — Rearranges the columns in the parametric table.
Variables/Add

Choose **Variables/Add** to add a column to the table that represents the value of a defined variable in each of the setups. When you choose this command, the window shown here appears with a list of defined variables that do not yet have columns in the table.

Only variables that are independent of other variables and have an active effect on the model are shown in the list.

> To add a column to the table for one of the variables:
  1. Choose **Variables/Add**.
  2. Select the variable from the list.
  3. Choose **OK**.

A column is added to the left edge of the table to represent that variable. Initially, the value of all of the cells in the column is set to that variable’s value in the nominal problem.
**Variables/Delete**

Choose **Variables/Delete** to delete a variable’s column from the table. The number of set-ups does not change. If a parametric solution is executed later, the value of that variable in each setup is assumed to be equal to its value in the nominal problem.

- To delete a variable’s column from the table:
  1. Select the entire column by clicking on it’s heading.
  2. Choose **Variables/Delete**. A message appears, asking you to confirm the deletion of the selected variable columns.
  3. Choose **Yes** to delete the column or **No** to cancel the action.

The column is deleted from the table.

**Variables/View**

Choose **Variables/View** to bring up a list of all of the variables defined in the problem. It also displays which part of the simulator uses this variable, and the variable’s value in the nominal problem.

When you are finished viewing the variables:
- Choose **OK**.
Choose **Variables/Animate** to view the list of setups as frames in an animation. This animation shows changes that occur to the physical model, based on constraints.

- If you have a group of rows selected, those rows are used in the animation.
- If no rows are selected, all rows are used.

When you choose this command, the following window appears:
Maxwell 2D — Variables Menu

Animating the Model

Two checkboxes control the speed of the animation:

- Turn on the **Single Step** checkbox to have the animator pause after it displays each physical setup, and wait for you to select a button before it displays the next setup.
- Turn off the **Verify Model** checkbox to speed up the animation because the animator does not double-check the accuracy of the model.

To animate the model:
1. Set the **Single Step** and **Verify Model** checkboxes to the desired settings.
2. Select **Start**. The software is now animating the model. As each frame is displayed, the setup that it represents is listed above the **Done** button.
3. If you turned on the **Single Step** checkbox, you can step through the animation using the arrow buttons:
   - Select the left arrow (<<) to back up one frame.
   - Select the right arrows (>>) to proceed on frame.
   If you left **Single Step** off for continuous animation, you can use the arrow buttons to affect the direction of animation:
     - Select the left arrow (<<) to animate the setups in reverse order.
     - Select the right arrows (>>) to animate the setups in forward order.
4. Select **Done** when you are finished. You return to the parametric table.

Changing the View of the Geometric Model

The **Zoom In**, **Zoom Out**, **Fit All**, **Fit Drawing**, **Fill Solids/Wire Frame** commands work the same as on the **Executive Commands** menu.

Variables/Rearrange Columns

Choose this command to rearrange the order of the columns in the parametric table.

To rearrange the columns:
1. Choose **Variables/Rearrange Columns**. The **Rearrange Table** window appears.
2. Select the column name to move from the list.
3. Choose **Up** or **Down** to move the selected column in the sequence.
4. Choose **OK**. The parametric table is redrawn with the new sequence.
Data Menu

In the spreadsheet, use the commands on the Data menu to perform the tasks listed below:

- Fill a cell or group of cells with a value or series of values.
- Create rows in the table that contain a value or series of values.
- Sort rows in the table based on the values of the variables.

The Data appears as shown below:

Data Commands

The function of each Data command is as follows:

**Fill**  Change the value stored in a table cell to a certain value, or a group of cells to a series of values.

**Sweep**  Add rows to the table, filling the variable columns with a value or series of values, or replace the current table with a new set of rows.

**Sort**  Put rows of the table into ascending or descending order, indexed by the values of selected variables.

Note: The Data/Fill and Data/Sweep commands do not work in the Parametrics Post Processor and remain grayed out.
Choose **Data/Fill** to set the value of a table cell or group of cells to a value or series of values. You must have at least one cell selected to use this command. If no cells are selected, the command is inactive, and grayed out.

Once you have selected at least one cell, choose **Data/Fill**. The following window appears:

Values are generated by entering an expression using the variable \( t \) and providing starting and ending values for \( t \). Choosing **Help** brings up a window containing a list of all legal operators and functions that can be performed on \( t \).

> For example, to fill ten rows with a simple series from one to ten:

1. Select ten rows in the table by clicking in one column and dragging until ten rows in that column are selected.
2. Enter 1 in the **t_start** field.
3. Enter 10 in the **t_end** field.
4. Choose **OK**.

The ten cells are then filled with the numbers from one to ten.

> Sinusoidal values could be entered via this procedure:

1. Select a group of rows in the table by clicking in one column and dragging until the...
desired number of rows in that column are selected.
2. Enter \texttt{sin(t)} in the \texttt{val(t)} field.
3. Enter \texttt{0} in the \texttt{t_start} field.
4. Enter \texttt{90} in the \texttt{t_end} field.
5. Choose \texttt{OK}.

All trigonometric functions expect their arguments to be in degrees, and inverse trigonometric functions’ return values are in degrees. Note that “pi” is a built-in constant, and may not be reassigned. Intrinsic function names are reserved, and may not be used as variable names.
Choose **Data/Sweep** to add rows representing different setups to the table. To use this command, you must have at least one variable defined, and added to the table. When you choose this command, a window similar to the following one appears:

> To add setups to the table using this window:

1. Choose a variable from the list of variables in your table. If you need to add more variable columns to the table:
   a. Choose **Cancel**.
   b. Use the **Variables/Add** command to add a column for each variable.
   c. Choose **Data/Sweep** again.
2. Set the **val(t)** field to the function that will yield the correct values. Choosing **Help** shows you a list of the legal operators and functions. For a simple series like a sweep from one to ten, you can leave this field set to \( t \).
3. Set the **t_start** field equal to the desired initial value of \( t \).
4. Set the **t_end** field equal to the desired final value of \( t \).
5. Set the number of samples equal to the number of rows you would like the sweep to span. The actual number of rows added is equal to the product of the number of...
samples for all the variables.
6. Choose Accept.
7. Repeat steps 1 though 6 for each variable you would like to have varying value.
8. Do one of the following:
   • Select Append to table to add the rows to the existing table.
   • Deselect Append to table to replace the current table with the new rows.
9. Choose OK. If you opted to replace the table with the new sweep, a dialog box asks you if you are sure you want to lose the old table.
10. Choose OK to replace the table or Cancel to cancel the action.

If you chose to append the sweep to the current table, the setups are added to the table, below the existing setups. If you chose to replace the current table, the old setups are removed, and the new ones take their place.
Data/Sort

Choose **Data/Sort** to sort a number of rows, indexed to the values in their vertical columns.

- If no cells are selected, this command sorts the entire table based on any selection of column values.
- If a group of cells is selected, only those rows and columns represented in the selection are used by the command.

When you choose **Data/Sort**, a window similar to the following one appears:

![Sort Setup Window](image)

On the left is a list of the variables that can be used as keys for sorting. On the right is a list of the variables that are to be used as sort keys.

**Selecting Sort Keys**

1. To use a variable as a sort key:
   - Select it from the list of variables.
   - Choose **Add**.
Maxwell 2D — Data Menu

Deleting Sort Keys

> To remove a variable from the list of sort keys:
  1. Select it from the list of sort keys.
  2. Choose Remove.

Note: You may also transfer the variable name from one list to the other by double-clicking on its name in the list.

Sorting Data

> To set up and execute a sort on the selected rows:
  1. Select variables from the list of variables to use as sort keys.
  2. Do one of the following:
     • Select Ascending to sort the rows by ascending value.
     • Select Descending to sort the rows by descending value.
  3. Choose OK.

The selected rows are sorted as you specified.

Changing the Sort Priority

The software goes down the list of sort keys, and sorts each row in order. Therefore, if several sort keys have been chosen, those lower on the list have greater effect on the order of the rows than those further up on the list.

> To reorder the list:
  • Select the item you want moved to the top of the Sort Keys list.

The list of sort keys is reordered to have that item at the top. In order to set up a specific sort order, select keys in inverse order of their desired priority.
Maxwell 2D — Plot Menu (Parametrics)

Plot Menu

Use the **Plot** commands in the Maxwell 2D Parametric Post Processor to do the following:

- Plot the results of a parametric sweep.
- Add new data to an existing plot.
- Delete plots from the screen.
- Change the line color, line thickness, or other attributes of a plot.
- Change the scale, labels, number of tick marks, and other plot axes parameters.
- Zoom into or out of a plot, or redraw the plot to fill the plot window.
- Show the coordinates of points you select.

When you choose **Plot** from the Parametric Post Processor menu bar, the following menu appears:
Plot Commands

The commands on the Plot menu are:

- **New** Draws a new plot.
- **Open** Reads in a previously-saved plot from a disk file.
- **Close** Removes an existing plot from the screen.
- **Save As** Saves a plot to a disk file.
- **Create Composite Plot** Creates composite plots.
- **Add Graphs** Adds another graph to an existing plot.
- **Show Coordinates** Displays the x- and y-coordinates of points on the plot.
- **Format** Changes the appearance of the following parts of a plot:
  - **Axes** Specifies axis scales, labels, plot headings, and the minimum and maximum values to be plotted. Also determines the axis tick mark and grid settings.
  - **Graphs** Specifies the color, line thickness, and line style of the graphs on a plot. Also determines the type of markers displayed at data points, and whether the curve is visible on the plot.
- **Zoom In** Zooms into an area of the plot, rescaling the axes to magnify the view.
- **Zoom Out** Zooms out of an area of the plot, rescaling the axes to shrink the view.
- **Fit All** Redraws and rescales the plot so that it fills the entire plot window.
Choose **Plot/New** to draw a plot of parametric solution results. The following information can be plotted:

- The value of any variable in the sweep.
- Solution information, such as energy, error energy, and number of triangles in the mesh.
- Results of parametric solutions, such as force, torque, capacitance, and so forth.

Any set of parametric solution data can be plotted against any other set of data. You can create as many plots as you like. Each plot is completely independent of other plots, and appears in its own plot window.

> To draw a new plot:

1. Highlight the rows in the parametric spreadsheet that contain the data to plot. If no rows are highlighted, the solution data in all rows is plotted.
2. Choose **Plot/New**. The following window appears:

The information that can be plotted (variables, solution information, and solution results) is listed under **Abscissa** and **Ordinate**.
3. Under **Abscissa**, highlight the data that serves as the x-axis of the plot.
4. Under **Ordinate**, highlight the data that serves as the y-axis of the plot. You can select more than one type of data to be plotted against the abscissa.
5. Select the plot type (**Cartesian** or **Polar**).
6. Do one of the following:
   - For a cartesian plot, set the scale of the x- and y-axes (**X Scale** and **Y Scale**):
     - **Linear** Specifies a linear scale for the x- or y-axis.
     - **Log** Specifies a logarithmic scale for the x- or y-axis.
   - For a polar plot, set the units of angular measurement. Click on the field next to **Angle Units** to display the units of angular measurement (**degrees** and **radians**), then choose the desired unit.
7. To smooth the curve, enter the degree of interpolation in the **Interpolation Order** field. Specify an interpolation order between -1 and 12. (Higher order interpolants tend to oscillate in an uncontrolled manner and are not supported in the Maxwell software.) By default, this field is set to -1, which does not smooth the curve.
8. Choose **Family of Curves** to cause the curve for each setup to be individually identified on the plot.
9. Choose **OK**.

The desired information is then plotted. For example, in the plot shown on the next page, the following settings were used:

- **Abscissa**: The force on the plunger in Newtons, labeled **Force(N)** on the plot.
- **Ordinate**: The variable **stroke**, a geometric constraint representing the distance between the plunger and the plug nut in a solenoid.
- **Plot Type**: **Cartesian**
- **X Scale, Y Scale**: **Linear**
- **Interpolation order**: -1

As you can see, the force on the plunger decreases as the distance increases. However, this relationship is not linear — the force levels off when **stroke** is between 0.047 and
Maxwell 2D automatically bases the plot axis settings on the maximum and minimum values of the plotted data. These settings may change as you plot other data with different values. To change the axis settings, choose **Plot/Format/Axes**.
Plot Windows

Each new plot displays in its own plot window, similar to the one shown above. Like Maxwell 2D project windows, plot windows are implemented using Motif window functions. They can be resized, expanded, or iconified using the window frame functions.

Plot/Open

Choose Plot/Open to open and view plots that have been saved to disk files. This command operates the same way as the File/Open command in the 2D Modeler, except that only plot files (*.dat) are listed.

Plot/Close

Choose Plot/Close to close an open plot.

> To do this:
1. Select the desired plot window as the active plot window.
2. Choose Plot/Close.

Plot/Create Composite Plot

Choose Plot/Create Composite Plot to plot a series of values on the same plot.

> To plot a series of values:
1. Choose Plot/New or Plot/Open to create the base plot.
2. Choose Plot/Create Composite Plot. A file browser appears.
3. Use the browser to select the plot to incorporate into the composite plot and choose OK.

The plots appear in the window.
Plot/Save As

Choose Plot/Save As to save a plot to a disk file. After you save a plot, you can later read the plot file back into the Parametrics post processor or into the PlotData utility in the Utilities panel.

This command operates the same way as the File/Save As command in the 2D Modeler, except that the software automatically appends a .dat extension to the file name you specify.
Plot/Add Graphs

Choose **Plot/Add Graphs** to do the following:

- Add new data to an existing plot. New data is automatically plotted against the abscissa you selected when creating the plot.
- Change the title of an existing plot.
- Specify how the newly plotted data is to be interpolated, or “smoothed.”

To do this:
1. Highlight the rows in the parametric spreadsheet that contain the data you’d like to plot. If no rows are highlighted, the solution data in all rows is plotted.
2. Select the desired plot window as the active plot window.
3. Choose **Plot/Add Graphs**. A window similar to the following one appears:

   ![Select Data to Plot Window]

4. Optionally, enter a new title for the plot.
5. Select one or more types of data to be added to the plot under **Ordinate**.
6. To smooth the curve, enter the degree of interpolation in the **Interpolation Order** field. Specify an interpolation order between -1 and 12. (Higher order interpolants tend to oscillate in an uncontrolled manner and are not supported in the Maxwell software.) By default, this field is set to -1, which does not smooth the curve.
7. Choose **Family of Curves** to cause the curve for each setup to be individually identified on the plot.
8. To plot the desired data, choose **OK**.

**Note:** The Maxwell 2D automatically bases the plot axis settings on the maximum and minimum values of the plotted data. These settings may change as you plot other data with different values. To change the axis settings, use the **Plot/Format/Axes** command.

**Plot/Show Coordinates**

Choose **Plot/Show Coordinates** to display the coordinates of points on the plot.

> To view the $x$- and $y$-coordinates of points on the plot:
1. Plot the desired data. To get a closer view of a graph to more accurately determine its coordinates, use the **Plot/Zoom In** command to zoom into that part of the plot.
2. Choose **Plot/Show Coordinates**.
3. Move the mouse to the desired point on the plot.
4. Click the left mouse button. A window appears showing the coordinates of the selected point. The point is marked with a cross.
5. To view the coordinates of additional points, repeat steps 3 and 4.
6. To exit the **Plot/Show Coordinates** command, click the right mouse button.

**Plot/Format**

Use the **Plot/Format** commands to modify the appearance of the plot axes and the plotted data:

- **Axes**
  Specifies axis scales, labels, plot headings, and minimum and maximum signal values to be plotted along with axis tick mark and grid settings.

- **Graphs**
  Specifies the color, line thickness, and line style of a previously plotted line. Also determines the type of markers displayed at solution data points, and whether the graph is visible on the plot.
Choose **Plot/Format/Axes** to change the following information associated with a plot:

- The scale of the x- and y-axes.
- The number and type of tick marks (if any) to be drawn on the x- and y-axes.
- Whether a grid is drawn on the major tick marks, minor tick marks, or both.
- The minimum and maximum data values that are displayed on the axes.
- The plot heading and labels on the x- and y-axes.

The following window appears when this command is selected. This command is identical to the **Plot/Format Axes** command in the **PlotData** utility.

**Note:** As an alternative to executing **Plot/Format/Axes**, you can double-click the right mouse button in the window containing your plot.
Choose **Plot/Format/Graphs** to change how graphs are drawn on a plot. The following attributes may be changed:

- Line color, width, and style.
- Whether markers are used to indicate data points.
- The type of symbol that marks data points.
- Whether the graph is visible on the plot.

The following window appears when this command is selected:

This command is identical to **Plot/Format Graphs** in the **PlotData** utility.
**Plot/Zoom In**

Choose **Plot/Zoom In** to zoom in on a region of the active plot window, magnifying the view.

> To zoom in on part of a plot:
1. Select the desired plot as the active plot window.
2. Choose **Plot/Zoom In**.
3. Select a point at one corner of the region on the plot that is to be zoomed.
4. Select the point in the diagonal corner.

The system then redraws and rescales the plot, expanding the selected area to fill the plot window.

**Plot/Zoom Out**

Choose **Plot/Zoom Out** to zoom out on the field of view in the active plot window.

> To zoom out of a plot:
1. Select the desired plot as the active plot window.
2. Choose **Plot/Zoom Out**.
3. Select a point at one corner of the region that is to be zoomed out.
4. Select the point in the diagonal corner.

The system redraws and rescales the plot, shrinking it to display in the selected area.

**Plot/Fit All**

Choose **Plot/Fit All** to redraw the plot so that the entire plot fills the plot window.

To redraw a plot to fit the plot window:

1. Select the desired plot window as the active plot window.
2. Choose **Plot/Fit All**.

The system then redraws and rescales the plot to fill the plot window.

Note that only the visible graphs are fitted to the plot window — the Maxwell 2D does not take invisible lines into account when changing the scale of the plot window with **Plot/Fit All**. You can change how graphs are shown on the plot.
Plotting Vector Data

As shown below, both components of force in X-Y problems are displayed in the parametrics table through the use of a variable type named vector.

<table>
<thead>
<tr>
<th>Triangles</th>
<th>Energy(J)</th>
<th>Error(%)</th>
<th>Force(N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>129</td>
<td>86.6331</td>
<td>0.516722</td>
<td>(462.199, -866.47)</td>
</tr>
<tr>
<td>121</td>
<td>86.6577</td>
<td>0.762956</td>
<td>(526.81, -835.88)</td>
</tr>
<tr>
<td>125</td>
<td>86.0533</td>
<td>0.768846</td>
<td>(371.58, -749.681)</td>
</tr>
<tr>
<td>134</td>
<td>85.3335</td>
<td>0.766864</td>
<td>(248.967, -965.395)</td>
</tr>
<tr>
<td>121</td>
<td>84.0991</td>
<td>0.765632</td>
<td>(227.301, -1051.85)</td>
</tr>
<tr>
<td>141</td>
<td>84.8268</td>
<td>0.764298</td>
<td>(177.983, -1866.99)</td>
</tr>
</tbody>
</table>

If you select vector type data for plotting, the following window appears, allowing you to select the components to plot.

To plot components of vector data:
1. Select any or all of the four vector components for plotting.
2. Choose OK to execute the plots.
Each component provides unique information, and your selections will be determined by your information needs. The selections are the following:

**Magnitude**  Plots the resultant of the X and Y forces, where:

\[
\text{Magnitude} = \sqrt{x^2 + y^2}
\]

**Angle**  Plots the angle, \( \theta \), of the resultant force with respect to the x-axis, where

\[
\theta = \arctan \left( \frac{y}{x} \right).
\]

- **Degrees**  Plots \( \theta \) in degrees, if **Angle** is selected.
- **Radians**  Plots \( \theta \) in radians, if **Angle** is selected.

**X**  Plots the force exerted in the x direction.

**Y**  Plots the force exerted in the y direction.

Choose **degrees** (the default) or **radians** for the angular units by clicking on the box labeled **degrees** to pull down the submenu.
The following figure shows a multiple plot graph with three components selected.

The customized appearance (headings, etc.) of this graph was achieved by using the formatting options that are available through the Plot/Format/Axes and Plot/Format/Graphs commands.
Plotting Complex Data

If you select a complex variable for plotting, the following window appears:

To plot components of complex values that are in the form \( a + bj \):
1. Select any or all of the four complex components.
2. Select **Use Frequency** to use the frequency in the plot.
3. Choose **OK** to execute the plot.

The selections and options are the following:

- **Magnitude**: Plots magnitude in the form \( \sqrt{a^2 + b^2} \).
- **Phase**: Plots phase angle in the form \( \tan^{-1}(\frac{b}{a}) \).
- **Degrees**: Plots phase angle in degrees, if **Phase** is selected.
- **Radians**: Plots phase angle in radians, if **Phase** is selected.
| **Real**  | Plots the real component, $a$. |
| **Imaginary**  | Plots the imaginary component, $b$. |
| **Use**  | The $b$ part of the complex expression is multiplied by $2\pi f$, where $f$ is the frequency. For example, a complex variable in the form $a + bj$ becomes $a + 2\pi fbj$. |
| **Frequency**  | Choose **degrees** (the default) or **radians** for the angular units by clicking on the box labeled **degrees** to pull down the submenu. |
Choose **Post Process/Transient Data** to:

- Plot the results of the transient motion problem.
- View the torque, power loss, position, and speed as functions of time.
- Calculate the fields and functions of the solution.

When you choose **Post Processor/Transient Data**, the PlotData utility appears, displaying the **Edit Plot** window:

While you are accessing PlotData, the Maxwell 2D Executive Commands menu remains active. This allows you to inspect material properties, boundary conditions, convergence information, and so forth while viewing the field solutions.
Plotting the Transient Solutions

The Edit Plot window appears by default when you first access the Transient Data Post Processor to allow you to select the type of plot to display.

1. Select the type of plot to display from the Loaded Signals list.
2. Select the Type of plot:
   - Cartesian plots display the data on x- and y-axes.
   - Polar plots display the data using polar coordinates.
   - Smith Charts display the data using the standard Smith chart coordinates and layout.
3. For Cartesian plots:
   a. Select Linear or Log to plot the data on linear or logarithmic for the X and Y scales.
   b. Optionally, select Separate Y Scales to plot the data on separate y axes.
4. For Polar plots:
   a. Select Degrees or Radians to define the units of the plot.
   b. Optionally, select Far-Field Pattern to plot the far fields in the polar plot.
5. For Smith Charts, select Degrees or Radians to define the units of the plot.
6. Choose OK.

A plot of the selected fields appears.

Once the window closes, use the PlotData utility to analyze the fields.

Exiting PlotData

When you have finished using PlotData, choose File/Exit from the PlotData menu bar to return to the Maxwell 2D Executive Commands window.
These technical notes contain background information on the theory behind the Maxwell 2D, including:

- A list of all Maxwell 2D software modules.
- Discussion of: background theory; capacitance, inductance, impedance, conductance, and admittance matrices; virtual force and torque; flux linkage; and current flow for the following field solvers:
  - Electrostatic
  - Magnetostatic
  - Eddy current
  - DC conduction
  - AC conduction
  - Eddy axial
  - Transient
- The differences between field solutions in cartesian (XY) and axisymmetric (RZ) models.
- An overview of the phasor notation used in this guide to represent complex field quantities.
Maxwell 2D consists of the following modules:

**Executive**
Controls the screen that contains the Executive Commands menu for Maxwell 2D. Also performs administrative tasks such as maintaining the material database and coordinating sources and boundary conditions.

**2D Modeler**
A drawing package that allows a user to draw a geometric model of the cross-section being modeled.

**Material Manager**
Defines materials and enter their electromagnetic characteristics. These materials can then be assigned to objects in your models. The Material Manager also contains a database of pre-defined materials that are available to all models.

**Boundary Manager**
Defines electromagnetic sources (such as currents, voltages, and charges) and boundary conditions modeling the field behavior at object interfaces and the edges of the solution region.

**2D Meshmaker**
Creates or refines a finite element mesh for cross-sectional models, preparing them for field simulations.

**Electrostatic Field Solver**
Computes the static electric field that exists in a structure given a distribution of DC voltages and static charges. A capacitance matrix, force, torque, and flux linkage may also be computed from the energy stored in the electric field.

**Magnetostatic Field Solver**
Computes the static magnetic field that exists in a structure given a distribution of DC currents. The magnetic field may be computed in structures with both non-linear and linear materials. An inductance matrix, force, torque, and flux linkage may also be computed from the energy stored in the magnetic field.

**AC Conduction Field Solver**
Computes the AC currents that flow in a dielectric given a distribution of AC voltages. An admittance matrix and current flow may also be computed from the energy losses in dielectrics.

**DC Conduction Field Solver**
Computes the DC currents that flow in a lossy dielectric given a distribution of DC voltages. A conductance matrix and current flow may also be computed from the energy stored in the electric field and from energy losses in dielectrics.
After you access Maxwell 2D, these software modules are invoked as necessary.

**Eddy Current Field Solver**
Computes the oscillating magnetic field that exists in a structure given a distribution of AC currents. Also computes current densities, taking into account all eddy current effects (including skin effects). An impedance matrix, force, torque, and current flow may also be computed from the energy stored in the magnetic field and from energy losses in conductors.

**Eddy Axial Field Solver**
Computes the oscillating magnetic field that is induced in a structure by external oscillating magnetic fields. Current flow may also be computed from the energy stored in the magnetic field and from energy losses in conductors.

**Transient Solver (EMpulse)**
Computes the forces, torques, speeds, and energy on moving objects.

**Thermal Solver**
Computes the temperature on the system.

**Post Processor**
Manipulates and displays quantities such as $\phi$, $E$, $D$, $B$, and $H$.

**Parametric Analysis**
Enables you to define and perform parametric sweeps — solutions in which you specify a range of values for one or more variables in your model (such as a geometric dimension, material property, source, and so forth).

**Note:**
The specific field solvers that are available to you depend on which software package you purchased — *Electric Fields*, *DC Magnetics*, *AC Magnetics*, or the complete Maxwell 2D package. The *Parametric Analysis* and *EMpulse* modules are available only if you purchased those software packages. All field solvers are distributed with them.
Electrostatic Field Simulation

The electrostatic field simulator computes static electric fields arising from potential differences and charge distributions.

Theory

The electrostatic field simulator solves for the electric potential, \( \phi(x, y) \), in this field equation:

\[
\nabla \cdot (\varepsilon_r \varepsilon_0 \nabla \phi(x, y)) = -\rho
\]

where:

- \( \phi(x, y) \) is the electric potential.
- \( \varepsilon_r \) is the relative permittivity. It can be different for each material.
- \( \varepsilon_0 \) is the permittivity of free space, \( 8.854 \times 10^{-12} \text{ F/m} \).
- \( \rho(x,y) \) is the charge density.

This equation is derived from Gauss’s Law, which indicates that the net electric flux passing through any closed surface is equal to the net positive charge enclosed by that surface. In differential form, Gauss’s Law is:

\[
\nabla \cdot \mathbf{D} = \rho
\]

where \( \mathbf{D}(x,y) \) is the electric flux density. Since \( \mathbf{D} = \varepsilon_r \varepsilon_0 \mathbf{E} \), then:

\[
\nabla \cdot (\varepsilon_r \varepsilon_0 \mathbf{E}(x, y)) = \rho
\]

In a static field, \( \mathbf{E} = -\nabla \phi \). Therefore,

\[
\nabla \cdot (\varepsilon_r \varepsilon_0 \nabla \phi(x, y)) = -\rho
\]

which is the equation that the electrostatic field simulator solves using the finite element method.

After the solution for the potential is generated, the system automatically computes the E-field and D-field using the relations \( \mathbf{E} = -\nabla \phi \) and \( \mathbf{D} = \varepsilon_r \varepsilon_0 \mathbf{E} \).

An arrow plot of an electric field generated by the electrostatic field simulator is shown.
Electrostatic Field Simulation
Theory
- Capacitance
  - Capacitance in Terms of Charges and Voltages
  - Capacitance in Terms of Currents and Time Varying Voltages
- Computing Capacitance
- Virtual Forces
- Virtual Torques
- Flux Linkage

Magnetostatic Field Simulation
Theory
- Inductance
- Virtual Forces
- Virtual Torques
- Flux Linkage
Capacitance

At the simplest level, capacitance represents the amount of energy stored in the electric field in and surrounding a structure. In a single circuit, the capacitance represents the amount of energy stored in the electric field that arises due to a voltage differential across a dielectric.

\[ U_e = \frac{1}{2} C v^2 \]

where \( U_e \) is the energy stored in the electric field, \( C \) is the capacitance, and \( v \) is the voltage across the dielectric.

The Maxwell 2D computes the capacitance between two conductors by simulating the electric field that arises when a voltage differential is applied. By computing the energy stored in the field, the corresponding capacitance can be computed.

\[ C = \frac{2U_e}{v^2} \]

To compute capacitances using this method, the E-field and D-field associated with a given distribution of voltages must first be computed. The electrostatic field simulator, which computes the electric potential at all points in the problem region, does this.
Capacitance in Terms of Charges and Voltages

A capacitance matrix represents the charge coupling within a group of conductors — that is, the relationship between charges and voltages for the conductors. Given the three conductors shown below, with the outside boundary taken as a reference, the net charge on each object will be:

\[
Q_1 = C_{10} V_1 + C_{12} (V_1 - V_2) + C_{13} (V_1 - V_3)
\]
\[
Q_2 = C_{20} V_2 + C_{12} (V_2 - V_1) + C_{23} (V_2 - V_3)
\]
\[
Q_3 = C_{30} V_3 + C_{13} (V_3 - V_1) + C_{23} (V_3 - V_2)
\]

This can be expressed in matrix form as:

\[
\begin{bmatrix}
Q_1 \\
Q_2 \\
Q_3
\end{bmatrix} =
\begin{bmatrix}
C_{10} + C_{12} + C_{13} & -C_{12} & -C_{13} \\
-C_{12} & C_{20} + C_{12} + C_{23} & -C_{23} \\
-C_{13} & -C_{23} & C_{30} + C_{13} + C_{23}
\end{bmatrix}
\begin{bmatrix}
V_1 \\
V_2 \\
V_3
\end{bmatrix}
\]

The capacitance matrix above gives the relationship between \(Q\) and \(V\) for the three conductors and ground. In a device with \(n\) conductors, this relationship would be expressed by an \(n \times n\) capacitance matrix. Capacitance matrix values are specified in farads (coul-
If one volt is applied to Conductor 1 and zero volts is applied to the other two conductors, the capacitance matrix becomes:

\[
\begin{bmatrix}
Q_1 \\
Q_2 \\
Q_3
\end{bmatrix} = \begin{bmatrix} 1 & 0 \\ 0 & 0 \end{bmatrix} = \begin{bmatrix}
C_{10} + C_{12} + C_{13} \\
-C_{12} \\
-C_{13}
\end{bmatrix}
\]

The diagonal elements in the matrix (such as \( C_{(1,1)} \)) are the sum of all capacitances from one conductor to all other conductors. These terms represent the self-capacitance of the conductors. Each is numerically equal to the charge on a conductor when one volt is applied to that conductor and the other conductors (including ground) are set to zero volts. For instance,

\[
C_{(1,1)} = C_{10} + C_{12} + C_{13}
\]

The off-diagonal terms in each column (such as \( C_{(1,2)} \), \( C_{(1,3)} \)) are numerically equal to the charges induced on other conductors in the system when one volt is applied to that conductor. For instance, in column one of the example capacitance matrix, \( C_{(1,2)} \) is equal to -\( C_{12} \). This is equal to the charge induced on Conductor 2 when one volt is applied to Conductor 1 and zero volts are applied to Conductor 2.

The off-diagonal terms are simply the negative values of the capacitances between the corresponding conductors (the mutual capacitances). In column one of the example capacitance matrix, the off-diagonal terms represent the capacitances between Conductor 1 and the other two conductors; in column two, the terms represent the capacitance between Conductor 2 and the other conductors; and so forth.

Note that the capacitance matrix is symmetric about the diagonal. This indicates that the mutual effects between any two objects are identical. For instance, \( C_{(1,3)} \), the capacitance between Conductor 1 and Conductor 3 (-\( C_{13} \)), is equal to \( C_{(3,1)} \), the capacitance between Conductor 3 and Conductor 1.
Capacitance in Terms of Currents and Time Varying Voltages

A capacitance matrix can also represent the relationship between currents and time varying voltages in a system of conductors.

Given the three transmission lines shown here,

\[
\begin{align*}
    &\frac{dV_1}{dt} \\
    &\frac{dV_2}{dt} \\
    &\frac{dV_3}{dt}
\end{align*}
\]

the currents caused by the time varying voltage source on each line are given by the following relationship:

\[
\begin{bmatrix}
    i_1 \\
    i_2 \\
    i_3
\end{bmatrix} =
\begin{bmatrix}
    C_{10} + C_{12} + C_{13} & -C_{12} & -C_{13} \\
    -C_{12} & C_{20} + C_{12} + C_{23} & -C_{23} \\
    -C_{13} & -C_{23} & C_{30} + C_{13} + C_{23}
\end{bmatrix}
\begin{bmatrix}
    \frac{dV_1}{dt} \\
    \frac{dV_2}{dt} \\
    \frac{dV_3}{dt}
\end{bmatrix}
\]
If \( \frac{dV_2}{dt} \) and \( \frac{dV_3}{dt} \) are set to zero, this relationship becomes:

\[
\begin{bmatrix}
i_1 \\
i_2 \\
i_3
\end{bmatrix} = \begin{bmatrix}
C_{10} + C_{12} + C_{13} \\
0 \\
0
\end{bmatrix} \begin{bmatrix}
\frac{dV_1}{dt} \\
0 \\
0
\end{bmatrix} = \begin{bmatrix}
C_{10} + C_{12} + C_{13} \\
-C_{12} \\
-C_{13}
\end{bmatrix} (\frac{dV_1}{dt})
\]

This gives the currents that are induced on Line 2 and Line 3 when a time varying voltage source is applied to Line 1 — that is, the capacitive coupling between the three lines, or the short circuit capacitance.
Computing Capacitance

To compute a capacitance matrix for a structure, the Maxwell 2D performs a sequence of electrostatic field simulations. In each field simulation, one volt is applied to a single conductor and zero volts is applied to all other conductors. Therefore, for an \( n \)-conductor system, \( n \) field simulations are automatically performed.

The energy stored in the electric field associated with the capacitance between two conductors is given by the following relation:

\[
U_{ij} = \frac{1}{2} \int_{\Omega} D_i \cdot E_j d\Omega
\]

where:

- \( U_{ij} \) is the energy in the electric field associated with flux lines that connect charges on conductor \( i \) to those on conductor \( j \).
- \( D_i \) is the electric flux density associated with the case in which one volt is placed on conductor \( i \).
- \( E_j \) is the electric field associated with the case in which one volt is placed on conductor \( j \).

The capacitance between conductors \( i \) and \( j \) is therefore:

\[
C = \frac{2U_{ij}}{v^2} = \int_{\Omega} D_i \cdot E_j d\Omega
\]
Virtual Forces (Electrostatic)

To compute the virtual force on an object, the electrostatic field simulator uses the principle of virtual work. In the structure shown below, the force on the bottom plate (plate B) in the direction of the displacement, \( x \), is given by the following relationship:

\[
F_B = \frac{dW(v, x)}{dx} \bigg|_{V = \text{Constant}}
\]

where \( W \) is the stored energy of the system,

\[
W(v, x) = \frac{1}{2} \int_{V=0}^{V=E} E \cdot D dVol
\]

Unlike the classical virtual work method, the plate is not actually moved during the numerical process of the force computation. Instead, only the tetrahedra that lie along the outside surface of the object are virtually distorted. \( W \) and its derivative, \( dW/dx \), are calculated from a single field solution using finite element interpolation functions.
Virtual Torques (Electrostatic)

Similar to the virtual force calculation, the system uses virtual work principles to compute the torque on an object. In the structure shown below, the virtual torque on the bottom plate (plate B) about the axis of rotation is given by the following relationship:

\[ T_B = \frac{dW(v, \theta)}{d\theta} \bigg|_{V = \text{Constant}} \]

where \( W \) is the stored energy of the system,

\[ W(v, \theta) = \frac{1}{2} \int_V E \cdot D dV \]

Unlike the classical virtual work method, the plate is not actually rotated during the numerical process of the torque computation. Instead, only the tetrahedra that lie along the outside surface of the object are virtually distorted. \( W \) and its derivative, \( dW/d\theta \), are calculated from a single field solution using finite element interpolation functions.
Flux Linkage (Electrostatic)

To compute the electric flux linkage, the electrostatic field solver uses the following relationship:

$$\Psi = \int E \cdot dA$$

where $E$ is the electric field and $A$ is the area over which flux density is computed.

- In cartesian (XY) models, the area is found by sweeping the flux line you’ve drawn in the xy-plane into the z direction — forming a 3D surface. The electric flux value computed is the flux per meter depth in the z direction.
- In axisymmetric (RZ) models, the area is found by rotating the flux line you’ve drawn in the rz-plane 360 degrees about the z axis. The electric flux computed is the total flux that passes through this surface.

A separate flux linkage value is computed for each line you draw.
Magnetostatic Field Simulation

The magnetostatic field simulator lets you compute static magnetic fields arising from DC currents and other sources like permanent magnets and external magnetic fields. Magnetic fields in both linear and nonlinear materials can be simulated.

Theory

The magnetostatic field simulator solves for the magnetic vector potential, \( A_z(x,y) \) in this field equation:

\[
J_z(x, y) = \nabla \times \left( \frac{1}{\mu_r \mu_0} \left( \nabla \times A_z(x, y) \right) \right)
\]

where:

- \( A_z(x,y) \) is the z component of the magnetic vector potential.
- \( J_z(x,y) \) is the DC current density field flowing in the direction of transmission.
- \( \mu_r \) is the relative permeability of each material.
- \( \mu_0 \) is the permeability of free space.

Given \( J_z(x,y) \) as an excitation, the magnetostatic field simulator computes the magnetic vector potential at all points in space.

Note:

In general, both \( J \) and \( A \) are vectors. However, \( J \) is assumed to only have a z-component. A consequence of this is that \( A \) only has a z-component as well. Both quantities can therefore be treated as scalars.

The equation that the magnetostatic field solver computes is derived from Ampere's law, which is:

\[
\nabla \times H = J
\]

Since \( H = \frac{B}{\mu_r \mu_0} \), then:

\[
\nabla \times \left( \frac{B}{\mu_r \mu_0} \right) = J
\]

Since \( B = \nabla \times A \), then:
The magnetostatic field simulator solves this equation using the finite element method. After \( A(x,y) \) is computed, the magnetic flux density, \( B \), and the magnetic field, \( H \), can then be computed using the relationships:

\[
\nabla \times \left( \frac{1}{\mu_r \mu_0} \nabla \times A \right) = J
\]

The magnetic flux density, \( B \), and the magnetic field, \( H \), can then be computed using the relationships:

\[
B = \nabla \times A
\]

\[
H = \frac{B}{\mu_r \mu_0}
\]

Both \( B \) and \( H \) lie in the xy cross-section being analyzed. An arrow plot of a B-field generated by the magnetostatic field simulator is shown below:
Inductance

At the simplest level, inductance represents how much energy is stored in the magnetic field when current flows.

\[
U_m = \frac{1}{2}Li^2
\]

where:
- \(U_m\) is the energy stored in the magnetic field
- \(L\) is the inductance
- \(i\) is the current flowing in the circuit

The Maxwell 2D computes inductances associated with a structure by simulating the magnetic field that arises when various voltages and currents are applied. Then, by computing the energy stored in those fields, it can then compute the necessary inductances:

\[
L = \frac{2U_m}{i^2}
\]

To compute inductances using this method, the B-field and H-field associated with a distribution of currents must first be computed. The magnetostatic field simulator, which computes the magnetic vector potential at all points in the problem region, performs this task.
**Inductance in Terms of Flux Linkage and Currents**

An inductance matrix represents the magnetic flux linkage between the current loops in a system. Given the three current loops below, the relationship between induced flux and currents is as follows:

\[
\begin{align*}
\lambda_1 &= L_{11}i_1 + L_{12}i_2 + L_{13}i_3 \\
\lambda_2 &= L_{12}i_1 + L_{22}i_2 + L_{23}i_3 \\
\lambda_3 &= L_{13}i_1 + L_{23}i_2 + L_{33}i_3
\end{align*}
\]

This can be expressed in matrix form as:

\[
\begin{bmatrix}
\lambda_1 \\
\lambda_2 \\
\lambda_3
\end{bmatrix} =
\begin{bmatrix}
L_{11} & L_{12} & L_{13} \\
L_{12} & L_{22} & L_{23} \\
L_{13} & L_{23} & L_{33}
\end{bmatrix}
\begin{bmatrix}
i_1 \\
i_2 \\
i_3
\end{bmatrix}
\]

The inductance matrix above gives the relationship between \( \lambda \) and \( i \) for the three independent current loops. In a device with \( n \) current loops, this relationship would be expressed by an \( n \times n \) inductance matrix. Inductance matrix values are specified in henries.

If one ampere is applied to Current Loop 1 and zero amperes are applied to the other two loops, the inductance matrix becomes:
The diagonal terms in the matrix (such as \( L_{11} \)) represent the self-inductance of each current loop. Self-inductance is numerically equal to the flux linkage in a current loop when one ampere is flowing in it, and no current is flowing in the other loops. For example, \( L_{11} \) is equal to the flux in Current Loop 1 when one ampere is flowing in that current loop, and no current is flowing in the other loops.

The off-diagonal terms (such as \( L_{12}, L_{13} \)) represent the mutual inductances between the current loops. Mutual inductance is numerically equal to the flux linkage in a current loop when one ampere is flowing through another loop, and no current is flowing anywhere else. For example, \( L_{12} \) is equal to the flux linkage in Loop 1 when one ampere is applied to Loop 2 and no current is flowing in the other loops.

Note that the inductance matrix is symmetric about the diagonal. This indicates that the mutual effects between any two loops are identical. For instance, \( L_{13} \), the inductance between Current Loop 1 and Current Loop 3, is equal to the inductance between Current Loop 3 and Current Loop 1.
Inductance in Terms of Voltages and Time Varying Currents

An inductance matrix can also represent the relationship between voltage and current fluctuations in a system. Given the three transmission lines shown below, the voltage changes caused by the time varying current source on each line are given by:

\[
\begin{bmatrix}
\Delta V_1 \\
\Delta V_2 \\
\Delta V_3
\end{bmatrix} =
\begin{bmatrix}
L_{11} & L_{12} & L_{13} \\
L_{12} & L_{22} & L_{23} \\
L_{13} & L_{23} & L_{33}
\end{bmatrix}
\begin{bmatrix}
(di_1)/(dt) \\
(di_2)/(dt) \\
(di_3)/(dt)
\end{bmatrix}
\]

The inductance matrix above gives the relationship between $\Delta V$ and $di/dt$ for the three independent transmission lines.

If $di_2/dt$ and $di_3/dt$ are set to zero, this relationship becomes

\[
\begin{bmatrix}
\Delta V_1 \\
\Delta V_2 \\
\Delta V_3
\end{bmatrix} =
\begin{bmatrix}
0 & 0 & L_{13} \\
0 & L_{12} & L_{23} \\
0 & L_{13} & L_{33}
\end{bmatrix}
\begin{bmatrix}
(di_1)/(dt) \\
(di_2)/(dt) \\
(di_3)/(dt)
\end{bmatrix}
\]

This gives the voltage changes that are induced on Lines 2 and 3 when a time-varying current source is applied to Line 1 — that is, the inductive coupling between all the loops.
Computing an Inductance Matrix

To compute an inductance matrix, the software performs a sequence of magnetostatic field simulations. In each field simulation, one ampere is allowed to flow in a single conductor. The current returns as defined under Setup Executive Parameters — either in the conductor you identified as the return path, or along outside balloon, value (Dirichlet) or odd symmetry boundaries. No current flows in any other conductor.

For an n-conductor system, n field simulations are automatically performed. The energy stored in the magnetic field that couples two conductors is:

\[
U_{ij} = \frac{1}{2} L I^2 = \frac{1}{2} \int B_i \cdot H_j d\Omega
\]

where:

- \(U_{ij}\) is the energy stored in the magnetic field linking conductor \(i\) with conductor \(j\).
- \(I\) is the current in conductor \(i\).
- \(B_i\) is the magnetic flux density where one ampere is allowed to flow through conductor \(i\).
- \(H_j\) is the magnetic field where one ampere is allowed to flow through conductor \(j\).

The inductance coupling conductors \(i\) and \(j\) is therefore:

\[
L_{ij} = \frac{2 U_{ij}}{I^2} = \int B_i \cdot H_j d\Omega
\]

For multiturn conductors, the net value of inductance is the value given by:

\[
L_{net} = N^2 L_{matrix}
\]

where \(N\) is the number of turns in the coil.
Virtual Forces (Magnetostatic)

To compute the force on an object, the system uses the principle of virtual work. In the structure shown below, the force on the plate in the direction of the displacement, \( s \), is given by the following relationship:

\[
F_{\text{plate}} = \left. \frac{dW(s, i)}{ds} \right|_{i = \text{const}} = \frac{\partial}{\partial s} \int_{V} (B \cdot dH) dV
\]

where \( W(s, i) \) is the magnetic coenergy of the system. The current, \( i \), is held constant.

Unlike the classical virtual work method, the plate is not actually moved during the force computation. Instead, only the tetrahedra that lie along the outside surface of the object are virtually distorted. Thus, the force computation only requires one field solution.
Virtual Torques (Magnetostatic)

Similar to the virtual force calculation, the system uses virtual work principles to compute the torque on an object. In the structure shown below, the torque on Object B about the axis of rotation is given by the following relationship:

\[ T_B = \left. \frac{dW(\theta, i)}{d\theta} \right|_{i = \text{const}} = \frac{\partial}{\partial \theta} \left[ \int_V (\int_0^H B \cdot dH) dV \right] \]

where \( W(\theta, i) \) is the magnetic coenergy of the system. The current, \( i \), is held constant.

Unlike the classical virtual work method, Object B is not actually rotated during the force computation. Instead, only the tetrahedra that lie along the outside surface of the object are virtually distorted. Thus, the change in the system's coenergy (and therefore the virtual torque) is given by the change in the coenergy of these tetrahedra.
Flux Linkage (Magnetostatic)

To compute the magnetic flux linkage, the magnetostatic field solver uses the following relationship:

$$ \Phi = \int B \cdot dA $$

where $B$ is the magnetic flux density and $A$ is the area over which flux density is computed.

- In cartesian (XY) models, the area is found by sweeping the flux line you've drawn in the xy-plane into the z direction — forming a 3D surface. The magnetic flux value computed is the flux per meter depth in the z direction.
- In axisymmetric (RZ) models, the area is found by revolving the flux line you’ve drawn in the rz-plane 360 degrees around the z-axis — forming a 3D surface. The magnetic flux value computed is the total flux that passes through this surface.

A separate flux linkage value is computed for each line you draw.
Eddy Current Field Simulation

The eddy current field simulator allows you to simulate the effects of time-varying currents in parallel-conductor structures — including eddy current effects in conductors.

Theory

Time-varying currents flowing in a conductor produce a time-varying magnetic field in planes perpendicular to the conductor. In turn, this magnetic field induces eddy currents in the source conductor and in any other conductor parallel to it. The eddy current field solver calculates the eddy currents by solving for $A$ and $\phi$ in the field equation:

$$\nabla \times \frac{1}{\mu} (\nabla \times A) = (\sigma + j \omega \epsilon) (\omega A + \nabla \phi)$$

where:

- $A$ is the magnetic vector potential.
- $\phi$ is the electric scalar potential.
- $\mu$ is the relative magnetic permeability.
- $\omega$ is the angular frequency at which all quantities are oscillating.
- $\sigma$ is the conductivity.
- $\epsilon$ is the relative permittivity.

Note: The eddy current equation is derived from Maxwell’s equations. Phasor notation is used to represent complex quantities.

A plot of flux lines produced by eddy currents that were computed in a structure by the
The eddy current solver is shown below:

Components of Current Density

Notice that the right side of the equation:

\[
\nabla \times \left( \frac{1}{\mu} \nabla \times A \right) = (\sigma + j\omega\varepsilon)(-j\omega A - \nabla\phi)
\]

consists of a complex conductivity multiplied by the complex value of \(E\) \((-j\omega A - \nabla\phi\). It is therefore equal to the complex current density, \(J\), which has three components:

- \(J_s\), the source current density due to differences in electric potential, \(-\sigma\nabla\phi\).
- \(J_e\), the induced eddy current density due to time-varying magnetic fields, \(-j\omega\sigma A\).
- \(J_d\), the displacement current density (time-varying electric fields), \(j\omega\varepsilon(-j\omega A - \nabla\phi)\).

The total current density is the sum of these three components. The \(j\omega\) term in the eddy and displacement components indicate that they are a function of frequency and become increasingly significant as the frequency increases.
Integrating the Current Density

When setting up a problem, you specify the total current flowing in any conductor that is connected to an external source. Therefore, the eddy current module is able to make use of a second equation:

\[ I_T = \int \frac{1}{\mu} (\sigma + j\omega\varepsilon)(-j\omega A - \nabla\phi) d\omega \]

which simply reflects the fact the total current in a conductor equals the integral of the current density over the cross-section of the conductor.

Assumptions

The eddy current field solver makes the following assumptions about the field quantities for which it solves:

- Time-varying electromagnetic quantities are assumed to have the periodic waveform:
  \[ F(t) = F_m \cos(\omega t + \theta) \]

- All quantities must have the same value of \( \omega \), but can have different phase angles (\( \theta \)). If a current is not a pure sinusoid, decompose it into sinusoidal harmonics, and solve separately at each frequency.

- All currents (source, eddy, and displacement) are assumed to flow perpendicular to the plane being studied (that is, in the z-direction). Therefore, the magnetic fields associated with these currents lie within the xy-plane. As a result, \( A \), the magnetic vector potential, has a z-component only.

- Because no currents flow in the xy-plane, the electric field, \( E \), has a z-component only. It follows that \( \phi \) is constant over the cross section of each conductor in the problem.
Deriving the Eddy Current Equation

The eddy current field solver uses the finite element method to compute \( A \) and \( \phi \) using these two relationships:

\[
\nabla \times \frac{1}{\mu_r} ( \nabla \times A ) = ( \sigma + j \omega \varepsilon ) (- j \omega A - \nabla \phi)
\]

\[
I_T = \int \frac{1}{\mu} (\sigma + j \omega \varepsilon )(- j \omega A - \nabla \phi) d\omega
\]

where:

- \( A \) is the magnetic vector potential.
- \( \phi \) is the electric scalar potential.
- \( \mu \) is the magnetic permeability.
- \( \omega \) is the angular frequency at which all quantities are oscillating.
- \( \sigma \) is the conductivity.
- \( \varepsilon \) is the permittivity.
- \( I_T \) is the total current flowing in conductors.

The following section shows how these equations are derived from Maxwell’s equations.
Maxwell's Equations

The eddy current field simulator solves for time harmonic electromagnetic fields governed by Maxwell's equations:

\[ \nabla \times \mathbf{H} = \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} \]

\[ \nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t} \]

\[ \nabla \cdot \mathbf{D} = \rho \]

\[ \nabla \cdot \mathbf{B} = 0 \]

where:

- \( \mathbf{E} \) is the electric field.
- \( \mathbf{D} \) is the electric displacement, \( \varepsilon \mathbf{E} \).
- \( \mathbf{B} \) is the magnetic flux density.
- \( \mathbf{H} \) is the magnetic field intensity, \( \mu \mathbf{B} \).
- \( \mathbf{J} \) is the current density, \( \sigma \mathbf{E} \).
- \( \rho \) is the charge density.

The eddy current solver assumes that all time-varying electromagnetic quantities in the problem have the form:

\[ F(t) = F_m \cos(\omega t + \theta) \]

Using Euler's formula:

\[ e^{j\alpha} = \cos \alpha + j \sin \alpha \]

If \( \alpha = \omega t + \theta \), \( F(t) \) equals the real portion of \( e^{j(\omega t + \theta)} \):

\[ F(t) = \Re[F_m e^{j(\omega t + \theta)}] = \Re[F_m (\cos(\omega t + \theta) + j \sin(\omega t + \theta))]] = F_m \cos(\omega t + \theta) \]

Now, because each time-varying quantity has the form \( F_m e^{j\theta} e^{j\omega t}, \frac{\partial \mathbf{D}}{\partial t} \) and \( \frac{\partial \mathbf{B}}{\partial t} \) are equal to \( j\omega \mathbf{D} \) and \( j\omega \mathbf{B} \).

Therefore, with this simplification and the relations \( \mathbf{H} = \mu \mathbf{B}, \mathbf{D} = \varepsilon \mathbf{E}, \) and \( \mathbf{J} = \sigma \mathbf{E} \), Maxwell's
equations reduce to:

\[
\nabla \times \frac{1}{\mu} B = (\sigma E + j\omega\epsilon E) \\
\n\nabla \times E = -j\omega B \\
\n\n\nabla \cdot \epsilon E = \rho \\
\n\n\nabla \cdot B = 0
\]

Relationship of Magnetic and Electric Field

The quantity that the eddy current field simulator actually solves for is \( A \), the magnetic vector potential. It is given by:

\[
\nabla \times A = B
\]

Substituting this into the first of Maxwell's equations, the result is:

\[
\nabla \times \left( \nabla \frac{1}{\mu} A \right) = (\sigma E + j\omega\epsilon E)
\]

A solution for \( E \) in terms of \( A \) is given by:

\[
E = -j\omega A - \nabla \phi
\]

where \( \phi \) is the electric potential. Substituting the right side of this relationship for \( E \) into the previous equation results in:

\[
\nabla \times \frac{1}{\mu} \left( \nabla \frac{1}{\mu} A \right) = (-j\omega A - \nabla \phi)(\sigma + j\omega\epsilon)
\]

This equation is one of the two used by the eddy current solver to compute \( A \) and \( \phi \).
Notice that the previous equation is in the form of a complex conductivity, \((\sigma + j\omega\varepsilon)\), times the complex value of \(E\). The result is the complex current density, \(J\). Therefore, the integral of this expression over the cross-section of a conductor is constrained to equal the total current that is specified as flowing in that conductor:

\[
I_T = \int_{\Omega} J d\Omega = \int_{\omega}^{\mu} (\sigma + j\omega\varepsilon)(-j\omega A - \nabla \phi) d\omega
\]

This is the second equation that the eddy current solver uses in computing \(A\) and \(\phi\).

The total current, \(I_t\), in this equation is the quantity you specify when setting up a problem. It is the total current flowing in a conductor, and includes:

- \(I_s\), the current from an external source, \(-\int_{\omega}^{\mu} \frac{1}{\mu} \sigma \nabla \phi d\omega\).
- \(I_e\), the induced eddy current, \(-\int_{\omega}^{\mu} \frac{1}{\mu} j\omega \varepsilon A d\omega\).
- \(I_d\), the displacement current, \(\int_{\omega}^{\mu} \frac{1}{\mu} j\omega \varepsilon (-j\omega A - \nabla \phi) d\omega\).

For problems solved by the eddy current solver, \(B\) is assumed to lie in the \(xy\) plane. Therefore, \(A\) can only have a component in the \(z\)-direction. The simulator does not have to solve for the \(x\) and \(y\) components of \(A\). Since \(E\) only has a \(z\)-component, \(\phi\) is a constant for each cross section of a conductor. Therefore, Maxwell does not have to solve for \(\phi\) at every node.

For a more detailed discussion of eddy current and skin effect problems in multiconductor systems, see:

Eddy Currents and Skin Depth

Induced currents allow magnetic fields to penetrate conductors only to a certain depth, which is approximated by the formula:

\[ \delta = \sqrt{\frac{2}{\omega \sigma \mu_0 \mu_r}} \]  

(in meters)

where:

- \( \omega \) is the angular frequency, which is equal to \( 2\pi f \). \( f \) is the frequency at which source currents and voltages oscillate during the solution.
- \( \sigma \) is the conductor’s conductivity, in siemens/meter.
- \( \mu_r \) is the conductor’s relative permeability, in amperes/meter.
- \( \mu_0 \) is the permeability of free space, which is equal to \( 4\pi \times 10^{-7} \text{ A/m} \).

Currents will be concentrated near the surface of the conductor, decaying rapidly past the skin depth. As the formula above indicates, the skin depth gets smaller as the frequency increases.
Impedance Matrix

An impedance matrix summarizes the relationship between AC voltages and AC currents in multi-conductor systems. Given the two current loops below, the relationships between voltages and currents in each loop is as follows:

\[ \Delta V_1 = I_1 R_{11} + I_2 R_{12} + I_1 j\omega L_{11} + I_2 j\omega L_{12} \]

\[ \Delta V_2 = I_2 R_{22} + I_1 R_{12} + I_2 j\omega L_{22} + I_1 j\omega L_{12} \]

This can be expressed in matrix form as:

\[
\begin{bmatrix}
\Delta V_1 \\
\Delta V_2
\end{bmatrix} =
\begin{bmatrix}
Z_{11} & Z_{12} \\
Z_{12} & Z_{22}
\end{bmatrix}
\begin{bmatrix}
I_1 \\
I_2
\end{bmatrix}
\]

where:

- \( V_i \) and \( I_i \) are phasors.
- \( Z_{11} = R_{11} + j\omega L_{11} \) (the self-impedance of Loop 1).
- \( Z_{12} = R_{12} + j\omega L_{12} \) (the mutual impedance between Loops 1 and 2).
- \( Z_{22} = R_{22} + j\omega L_{22} \) (the self-impedance of Loop 2).

The impedance matrix above gives the relationship between \( V \) and \( I \) for the two current loops. In a device with \( n \) current loops, this relationship would be expressed by an \( n \times n \) matrix.
impedance matrix. The matrix values displayed by the software are resistance and inductance (not resistance and reactance) and therefore do not include $j\omega$.

**Note:**

All impedances are complex numbers in the form:

$$Z = R + j\omega L$$

where:
- $\omega$ is equal to $2\pi f$, where $f$ is the frequency of the AC current source.
- $R$ is the AC resistance, given in ohms/meter (XY) or ohms (RZ).
- $L$ is the AC inductance, given in henries/meter (XY) or henries (RZ).

**Computing an Impedance Matrix**

Maxwell 2D breaks down the impedance matrix computation into two parts. First, it solves for the inductance matrix (L-matrix) associated with the model. It then solves for the resistance matrix (R-matrix). When it finishes solving for these matrices, the simulator combines them to form the impedance matrix, using the relationship $Z_i = R_i + j\omega L_i$.

To compute the inductance and resistance matrices for the impedance solution, the simulator generates an eddy-current field solution for each conductor in the matrix. In the first solution, the current in the first conductor is set to one ampere; currents in the other conductors are set to zero. This is done by imposing current sources on the conductors. In the second solution, the current in the second conductor is set to one ampere and all other conductors are set to zero amperes, and so forth. Objects that are not included in the impedance matrix are not affected.
### Inductance

To compute the inductance of the current loop, the simulator calculates the average energy, $U_{AV}$, of the system after a field solution is computed:

$$U_{AV} = \frac{1}{4} \int_{V} B \cdot H^* dV$$

Since the instantaneous energy of the system is equal to:

$$U_{Inst} = \frac{1}{2} I^2$$

where the instantaneous value of the current is related to the peak value of the current by $i = I_{Peak} \cos(\omega t + \theta)$. The average value for the energy can then be found by integrating the instantaneous energy:

$$U_{AV} = \frac{1}{2\pi} \int_{0}^{2\pi} U_{Inst} d\omega t = \left(\frac{L}{2}\right) \left(\frac{1}{2\pi}\right)^2 \int_{0}^{\frac{2\pi}{\omega}} I_{Peak}^2 \cos(\omega t + \theta)^2 d\omega t$$

From this, the average energy of the system is equal to:

$$U_{AV} = \left(\frac{L}{2}\right) I_{RMS}^2 = \left(\frac{L}{2}\right) \left(\frac{I_{Peak}}{\sqrt{2}}\right)^2 = \left(\frac{L}{4}\right) I_{Peak}^2$$

The inductance, therefore is:

$$L = \frac{4U_{AV}}{I_{Peak}^2}$$

The software assumes that the object for which impedance is being computed has a peak current of one ampere per coil turn flowing through it. Thus, the inductance is simply $4U_{AV}$.

Note that the inductance computed for an eddy current model does not necessarily equal the inductance computed for an equivalent magnetostatic model. The eddy current simulator partially includes the eddy effect in conductors when it computes inductance during an impedance solution.
Resistance

To compute the resistance, the simulator calculates the ohmic loss, \( P \), after a field solution has been computed:

\[
P = \frac{1}{2\sigma} \int_{V} J \cdot J^* dV
\]

The ohmic loss is related to the resistance by:

\[
P = RI_{RMS}^2
\]

The resistance is therefore:

\[
R = \frac{P}{I_{RMS}^2} = \frac{2P}{I_{Peak}^2}
\]

The system assumes that the object for which impedance is being computed has a peak current of one ampere per coil turn flowing through it. Therefore, the resistance is simply 2P.

Note that the resistance for an eddy current problem will be higher than the equivalent DC resistance, due to the skin concentration of currents.
Inductance and Resistance in Impedance Computations

The inductances and resistances computed during an impedance matrix solution are different from those computed for the equivalent DC case. This figure shows how they differ:

In the DC example, no eddy currents occur. The magnetic field created by the current flowing through the conductor is static. In the AC example, the oscillating magnetic field induces currents in conductors in the model. These induced currents affect the computation of inductance for the impedance matrix, causing it to be slightly different from the equivalent DC computation of inductance.
Virtual Forces (Eddy Current)

Virtual force in an eddy current problem is computed the same way as virtual force in a magnetostatic problem. The only difference is that the average value of force over time is computed — not the instantaneous force at a given time.

The difference between the time-averaged (or DC) force, AC force, and instantaneous force is shown below:

Force oscillates at twice the frequency of the source current and magnetic field:

\[
f_F = \frac{1}{T_F} = 2f_S
\]

where:

- \(f_F\) is the frequency of the force.
- \(f_S\) is the frequency of the source current and magnetic field.
- \(T_F\) is the period of the force.

The time-averaged (or DC) force, AC force, and instantaneous force can be determined...
The AC force, $F_{AC}$, must be evaluated at a particular phase ($=\omega t$) in order to determine its magnitude at an instant in time. However, the peak value of the AC force is reported as the “AC Fluctuation” in the force and torque solution panel for Maxwell 2D.

\[
F_{DC} = \frac{1}{2} \int Re |\mathbf{J} \times \mathbf{B}^*| \, dV
\]

\[
F_{AC} = \frac{1}{2} \int Re |\mathbf{J} \times \mathbf{B}| \, dV
\]

\[
F_{INST} = F_{DC} + F_{AC}
\]
Virtual Torques (Eddy Current)

Virtual torque in an eddy current problem is computed the same way as virtual torque in a magnetostatic problem. The only difference is that the average value of the torque over time is computed, not the net torque at a given time.

Current Flow (Eddy Current)

To compute the current flow, the eddy current field solver uses the following relationship:

\[ I = \int J \cdot dA \]

where:

- \( I \) is the current.
- \( J \) is the current density, given by:

\[ J = (\sigma + j\omega \varepsilon_r)(-j\omega A - \nabla \phi) \]

- \( A \) is the area over which the current flow is computed.
  - In cartesian (XY) models, the area is found by sweeping the current flow line you’ve drawn in the xy-plane into the z direction — forming a 3D surface. The current flow computed is the current per meter depth in the z-direction.
  - In axisymmetric (RZ) models, the area is found by revolving the flux line you’ve drawn in the rz-plane 360 degrees around the z-axis, forming a 3D surface. The current flow computed is the total current that passes through this surface.

A separate current flow value is computed for each line you draw.
Nonlinear Eddy Current Field Simulation

The eddy current solver for problems with nonlinear materials allows you to analyze the fundamental components of B and H at a specified frequency.

Theory

Unlike linear problems in which the energy is assumed to be based on the sinusoidal complex peak field values in a steady state, nonlinear problems are usually based on all harmonic components of the fields.

That is to say for nonlinear problems, though the current is sinusoidal at its input, the resulting fields are “harmonically rich.” Although the B- and H-fields are not sinusoidal at output, the Maxwell 2D Eddy Current Solver assumes that the fundamental components of B and H are sufficient for describing BH-curve and the associated field behavior.

Sinusoidal B

Assuming both that B is sinusoidal with time, t, and that the value of H is derived from the original BH-curve, introduce a new value \( H_e \) to represent the effective magnetic field. Using the average energy equation to define the time averaged coenergy density:

\[
\langle w_{co}(t) \rangle' = \frac{4}{T} \int_0^T \int_{H(0)}^{H(t)} BdH \ dt
\]

and the simple energy method in which:

\[
B_e = \mu_{eff} \dot{H}
\]

where:

\[
\mu_{eff} = \frac{4 \langle w_{co}(t) \rangle'}{\dot{H}^2}
\]
then:

\[
\frac{4}{T} \int_0^T H_e \sin(\omega t) B_m \sin(\omega t) dt = \frac{4}{T} \int_0^T (H \cdot B) dt
\]

and:

\[
T = \frac{2\pi}{\omega}
\]

or:

\[
\int_0^{\frac{\pi}{2}} H_e B_m \sin^2(\omega t) d\omega t = \int_0^{\frac{\pi}{2}} (H \cdot B) dt
\]

Allowing \( \alpha = \omega t \) and extracting \( H_e \), simplifying yields:

\[
H_e B_m \int_0^{\frac{\pi}{2}} \left( 1 - \cos^2 \alpha \right) d\alpha = \int_0^{\frac{\pi}{2}} (H \cdot B) d\alpha
\]

or:

\[
H_e = \frac{4}{\pi} \int_0^{\frac{\pi}{2}} \frac{(H \cdot B) d\alpha}{B_m} = \frac{4}{\pi} \sum_{i=1}^{n} \frac{\alpha_i \cdot (H \cdot B)}{B_m} d\alpha_i
\]

where:

- \( B_m \) relates to \( 2/\pi \).
- \( \alpha_i = \arcsin \left( \frac{B_i}{B_m} \right) \)

Now define the function \( f(\alpha_i) \) to be represented by:

\[
f(\alpha_i) = \frac{H_i B_i}{B_m}
\]
$H_e$ equates to:

$$H_e = \frac{2}{3\pi} \left( \sum_{i=1}^{n} f(\alpha_i) + 4 f\left(\frac{\alpha_i + \alpha_{i+1}}{2}\right) + f(\alpha_{i+1}) \right)$$

which represents the effective H for the BH-curve.
Sinusoidal H

Assuming both that $H$ is sinusoidal with time, $t$, and that the value of $B$ is derived from the original BH-curve, introduce a new value $B_e$ to represent the effective magnetic flux density.

Using the simple energy method equation and following the previous assumptions:

$$
\int_{0}^{\frac{T}{4}} B_e \sin(\omega t) H_B \sin(\omega t) dt = \int_{0}^{\frac{T}{4}} (H \cdot B) dt
$$

and allowing $\alpha = \omega t$, extracting $B_e$ and simplifying yields:

$$
B_e H_B \int_{0}^{\frac{\pi}{2}} \left( \frac{1 - \cos 2\alpha}{2} \right) d\alpha = \frac{\pi}{2} (H \cdot B) d\alpha
$$

Now define the function $f(\alpha_i)$ to be represented by:

$$
f(\alpha_i) = \frac{H_i B_i}{B_m}
$$

Solving for $B_e$ yields:

$$
B_e = \frac{4 \pi}{\pi H_B} \int_{0}^{\pi} (H \cdot B) d\alpha = \frac{4}{\pi} \sum_{i=1}^{n} \int_{\alpha_i}^{\alpha_{i+1}} \frac{(H \cdot B)}{H_B} d\alpha_i
$$

or:

$$
B_e = \frac{2}{3\pi} \sum_{i=1}^{n} f(\alpha_i) + f\left(\frac{\alpha_i + \alpha_{i+1}}{2}\right) + f(\alpha_{i+1})
$$

which represents the effective $B$ for the BH-curve.
Permeability

Nonlinear eddy current problems calculate the permeability based on the following equations:

\[ \mu_{\text{eff}} = \frac{B_e}{H_e} \]

where:

- \( B_e \) is the effective value of B in the BH-curve.
- \( H_e \) is the effective value of H in the BH-curve.
DC Conduction Field Simulation

The DC conduction field simulator allows you to analyze conduction currents due to static electric fields in conductors and lossy dielectrics.

Theory

When a material with a non-zero conductivity is subject to a potential difference, conduction current flows in the material. At all points in the problem space, the current density \( \mathbf{J} \) will be proportional to the electric field \( \mathbf{E} \) that is established due to the potential difference.

\[
\mathbf{J}(x, y) = \sigma \mathbf{E}(x, y) = -\sigma \nabla \phi(x, y)
\]

where:

- \( \mathbf{J}(x,y) \) is the current density.
- \( \mathbf{E}(x,y) \) is the electric field.
- \( \sigma \) is the conductivity of the material in MKS units (mhos/meter).
- \( \phi(x,y) \) is the electric potential.

The equation that the DC conduction field simulator solves is based on the fact that, under steady state conditions, the amount of charge, \( \rho \), leaving any infinitesimally small region must equal the charge flowing into that region.

\[
\nabla \cdot \mathbf{J} = \frac{\partial \rho}{\partial t} = 0
\]

The field quantity that DC conduction actually solves for is the electric potential, \( \phi \), in the following equation:

\[
\nabla \cdot (\sigma \nabla \phi) = 0
\]

Note that \( -\sigma \nabla \phi = \mathbf{J} \) and therefore these two equations are equivalent.

A plot of electric potential that was computed by the DC conduction solver is shown.
Steady-state Conditions

The DC conduction solver assumes that current flow in the conducting material has already reached steady state conditions. The implication of this assumption becomes clear when you compare the initial and steady state conditions of a simple problem such as that shown here:

First, consider the interval before steady state is reached. Assume that the switch establishing the potential across the parallel plates in the figure above closes at \( t=0 \). Also assume that the current required to deposit charges on the parallel plates (so that the voltage difference can be supported) occurs instantaneously.

However, it will take a while for current to start flowing in the dielectric. The time it takes for current to flow is determined by the time constant of the material. Therefore, at \( t=0 \), there will be an electric field in the dielectric, but no current and no free charges. Conse-
quently, the relationship that must be satisfied at the interface of the two dielectrics is:
\[ \nabla \cdot \mathbf{D} = \rho \]
or, equivalently:
\[ \nabla \cdot \varepsilon \mathbf{E} = \rho \]

At the interface between the two dielectrics, this relationship implies that:
\[ \varepsilon_1 E_{n1} = \varepsilon_2 E_{n2} \Rightarrow E_{n1} = \frac{\varepsilon_2}{\varepsilon_1} E_{n2} \]

where \( E_n \) is the normal component of \( \mathbf{E} \). Therefore, at \( t=0 \), before current starts to flow in the two dielectrics, \( \phi(x,y) \) is determined entirely by the permittivity of the dielectrics. Use the electrostatic field solver to solve for \( \phi(x,y) \) in such a case.

After current starts to flow in the dielectric and steady state is reached, free charges are able to accumulate at the boundaries of the two dielectrics. The free charge, \( \rho \), is no longer zero.

The relationship that must be satisfied now is:
\[ \nabla \cdot \mathbf{J} = \frac{\partial \rho}{\partial t} = 0 \]
or, equivalently,
\[ \nabla \cdot \sigma \mathbf{E} = 0 \]

This relationship implies that:
\[ \sigma_1 E_{n1} = \sigma_2 E_{n2} \Rightarrow E_{n1} = \frac{\sigma_2}{\sigma_1} E_{n2} \]

In this case, the solution depends on the conductivity (\( \sigma \)) of the materials rather than on their dielectric constant (\( \varepsilon \)). The DC conduction solver analyzes the steady state condition. Therefore, use the DC conduction field solver when steady state conditions have been reached.
Relevant Time Constant

The time constant that determines how long it will take the current flowing in the conductive material to reach steady state is given by \( \tau = \frac{\varepsilon}{\sigma} \). In a good conductor such as steel \( (\varepsilon = 9 \times 10^{-12}, \sigma = 1 \times 10^7) \), \( \tau = 9 \times 10^{-19} \) seconds. Steady state conditions are reached almost instantaneously and you should use the DC conduction solver to analyze the potential field.

On the other hand, for a good dielectric such as fused quartz \( (\sigma = 1 \times 10^{-17}) \), the time constant is on the order of \( 10^5 \) seconds. There will be a significant period before steady state currents start to flow and you should use the electrostatic solver to analyze the field. For many materials in between, a complete analysis may require both an electrostatic and a DC conduction solution.
Conductance

A conductance matrix gives the relationship between currents and voltage drops. In the figure below, two conductors at voltages $V_1$ and $V_2$ are touching a bar with a conductance, $G$.

Because the bar is not a perfect insulator, charge will flow between the two conductors and cause conduction currents. Given the conductors shown on the previous page, the relationship between the conduction current and the voltage drop, $V$, in each conductor is:

$$I = GV$$

where:

- $I$ is the conduction current.
- $G$ is the conductance, measured in mhos.
- $V$ is the voltage drop given by $V_1 - V_2$.

Conductance, as evident from the equation, is the inverse of resistance. Therefore, if a material has a large conductance, it will be a better conductor, and if it has a low conductance, it will be a better resistor.
To compute current flow, the DC conduction field solver uses the following relationship:

\[ I = \int J \cdot dA \]

where:

- \( I \) is the current.
- \( J \) is the current density, given by: \( J = \sigma E = -\sigma \nabla \phi \)
- \( A \) is the area over which the current flow is computed.

- In cartesian (XY) models, the area is found by sweeping the current flow line you’ve drawn in the xy-plane into the z direction — forming a 3D surface. The current flow computed is the current per meter depth in the z direction.
- In axisymmetric (RZ) models, the area is found by revolving the flux line you’ve drawn in the rz-plane 360 degrees around the z-axis, forming a 3D surface. The current flow computed is the total current that passes through this surface.

A separate current flow value is computed for each line you draw.
AC Conduction Field Simulation

The AC conduction field solver allows you to analyze conduction currents caused by time-varying electric fields in conductors and lossy dielectrics.

Theory

The AC conduction field simulator solves for $\phi$ in the following equation:

$$\nabla \cdot [\sigma E + j \omega \epsilon \nabla \phi(x, y)] = 0$$

where:

- $\phi(x, y)$ is the magnitude and phase of the electric potential at each value of $x$ and $y$.
- $\omega$ is the angular frequency at which the potential is oscillating.
- $\sigma$ is the conductivity.
- $\epsilon$ is the permittivity.

This equation can be expanded to:

$$\nabla \cdot (J + j \omega D) = 0$$

where:

- $J$ is the current density, $\sigma E$.
- $D$ is the electric flux density, $\epsilon E$.
- $E$ is the electric field, $-\nabla \phi$.

Complex quantities are represented using phasor notation.

Assumptions

The AC conduction field solver assumes the following conditions about field quantities:

- All time-varying electromagnetic quantities have the periodic waveform:

  $$F(t) = F_m \cos(\omega t + \theta)$$

  where all quantities must have the same value of $\omega$, but can have different phase angles ($\theta$).

- If a current is not a pure sinusoid, it is decompose into sinusoidal harmonics, and solved separately at each frequency.

- The component of $E$ due to time-varying magnetic fields caused by conduction currents can be neglected.
Admittance

Admittance can best be explained as the inverse of impedance, and is expressed by:

\[ Y = G - j\omega C \]

where:
- \( \omega \) is equal to \( 2\pi f \), where \( f \) is the frequency of the AC voltage source.
- \( Y \) is the admittance in mhos/meter.
- \( G \) is the conductance in mhos/meter.
- \( j\omega C \) is the susceptance in mhos/meter.

Note: The matrix values displayed by the software are conductance in mhos/meter and capacitance in farads/meter (not conductance and susceptance) and therefore do not include \( j\omega \).

Current Flow

To compute current flow, the AC conduction field solver uses the following relationship:

\[ I = \int J \cdot dA \]

where:
- \( I \) is the current.
- \( J \) is the current density, given by \( J = \sigma E \).
- \( A \) is the area over which the current flow is computed. It is found by sweeping the current flow line you've drawn in the xy-plane into the z direction, forming a surface. The current flow computed is the current per meter depth in the z direction.

A separate current flow value is computed for each line you draw.
**Eddy Axial Field Simulation**

The eddy axial field simulator allows you to analyze axial eddy currents in devices subject to time-varying magnetic fields.

**Theory**

When a conducting material is placed in a time-varying magnetic field, eddy currents are induced in the material. The currents circulate in the plane perpendicular to the magnetic field.

Given “sources” of $H$ in the form of boundary conditions, the eddy axial field solver uses the finite element method to calculate $H$ by solving this equation:

$$\nabla \times \left( \frac{1}{\sigma + j\omega\varepsilon} \nabla \times H \right) + j\omega\mu H = 0$$

where:

- $H$ is the magnetic field intensity.
- $\omega$ is the angular frequency at which all quantities are oscillating.
- $\mu$ is permeability.
- $\sigma$ is conductivity.
- $\varepsilon$ is permittivity.

This equation is derived from Maxwell’s equations. Complex quantities are represented using phasor notation.

Once the solution for $H$ at all points is obtained, induced currents are obtained by solving for $\sigma E$ and $j\omega\varepsilon E$ in the following relationship:

$$\nabla \times H = J + j\omega\varepsilon E = \sigma E + j\omega\varepsilon E = J_c + J_d$$

where:

- $J_c$ is the conduction current, $\sigma E$.
- $J_d$ is the displacement current, $j\omega\varepsilon E$.

Note that the eddy axial field solver assumes that there is no applied potential ($\phi$) in the...
problem. Therefore, the conduction current, which is usually given by:

\[ J_c = \sigma E = \sigma (-j \omega A - \nabla \phi) = -\sigma j \omega A - \sigma \nabla \phi = J_{Eddy} + J_{Source} \]

has no source component of current (-\sigma \nabla \phi). Hence the conduction current equals the eddy current.

A plot of the H-field induced by an oscillating external magnetic field is shown below:
Electromagnetic Sources

The only sources in the problem are established by specifying values of $H$ at boundaries. For example, when an alternating field is created by a large solenoid (one in which the length, $l$, is much larger than the diameter), the H-field inside the solenoid is uniform and given by:

$$H = \frac{nI}{l}$$

where:

- $n$ is the number of coil turns.
- $I$ is the current.
- $l$ is the length of the solenoid.

Only in regions where a conductor is present will the H-field be distorted by eddy currents. At all other points inside the solenoid, including the boundary around the conductor, $H$ will abide by the previous equation.

Assumptions

The eddy axial field simulator assumes the following conditions about the field quantities:

- All time-varying electromagnetic quantities are assumed to have the form:
  $$F(t) = F_m \cos(\omega t + \theta)$$
- All quantities must have the same value of $\omega$, but can have different phase angles ($\theta$).
- If the source waveform associated with the magnetic field intensity is not a pure sinusoid, decompose it into sinusoidal harmonics, and solve separately at each frequency. (This is only possible in linear systems where superposition is valid.)
- The magnetic field, $H$, has a z-component only, and is uniform in the z direction:
  $$H = 0\hat{x} + 0\hat{y} + H_z(x, y, t)\hat{z}$$
  However, $H$ can vary in the x- and y-directions.
- Because no currents flow in the z-direction, the electric field, $E$, has x- and y-components only:
  $$E = E_x(x, y, t)\hat{x} + E_y(x, y, t)\hat{y} + 0\hat{z}$$
Deriving the Eddy Axial Field Equation

The main equation (derived from Maxwell’s equations) that the eddy axial field simulator solves is:

\[ \nabla \times \left( \frac{1}{\sigma + j\omega \varepsilon} \nabla \times \mathbf{H} \right) + j\omega \mu \mathbf{H} = 0 \]

where:
- \( \mathbf{H} \) is the magnetic field intensity.
- \( \phi \) is the electric scalar potential.
- \( \mu \) is the magnetic permeability.
- \( \omega \) is the angular frequency at which all quantities are oscillating.
- \( \sigma \) is the conductivity.
- \( \varepsilon \) is the permittivity.
Obtaining Maxwell’s Equations in Terms of H

The eddy axial field solver solves for time harmonic electromagnetic fields governed by Maxwell’s equations:

\[
\begin{align*}
\nabla \cdot \mathbf{D} &= \rho \\
\nabla \cdot \mathbf{B} &= 0 \\
\n\nabla \times \mathbf{H} &= \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} \\
\n\nabla \times \mathbf{E} &= \frac{\partial \mathbf{B}}{\partial t} 
\end{align*}
\]

where:

- \( \mathbf{E} \) is the electric field.
- \( \mathbf{D} \) is the electric displacement, \( \varepsilon \mathbf{E} \).
- \( \mathbf{B} \) is the magnetic flux density.
- \( \mathbf{H} \) is the magnetic field intensity, \( \mu \mathbf{B} \).
- \( \mathbf{J} \) is the current density, \( \sigma \mathbf{E} \).
- \( \rho \) is the charge density.

The eddy axial solver assumes that all time-varying electromagnetic quantities in the problem have the form:

\[ F(t) = F_m \cos(\omega t + \theta) \]

Using Euler's formula:

\[ e^{j\alpha} = \cos \alpha + j \sin \alpha \]

If \( \alpha = \omega t + \theta \), \( F(t) \) equals the real portion of \( e^{j(\omega t + \theta)} \):

\[ F(t) = \Re \{ F_m e^{j(\omega t + \theta)} \} = \Re \{ F_m (\cos (\omega t + \theta) + j \sin (\omega t + \theta)) \} = F_m \cos (\omega t + \theta) \]

Each time-varying quantity has the form \( F_m e^{j\theta} e^{j\omega t} \). Using this relationship and the one given for \( \mathbf{D} \) above:
Substituting this into the first of Maxwell’s equations and solving for $E$:

$$\frac{1}{\sigma + j\omega\epsilon} (\nabla \times H) = E$$

Taking the curl of both sides, this equation becomes:

$$\nabla \times \left[ \frac{1}{\sigma + j\omega\epsilon} (\nabla \times H) \right] = \nabla \times E$$

Now, the curl of $E$ is given by:

$$\nabla \times E = -j\omega\sigma\mu H$$

Substituting this into the previous equation gives the following relationship:

$$\nabla \times \left[ \frac{1}{\sigma + j\omega\epsilon} (\nabla \times H) \right] = -j\omega\sigma\mu H$$

Or, in the form of a wave equation:

$$\nabla \times \left( \frac{1}{\sigma + j\omega\epsilon} \nabla \times H \right) + j\omega\mu H = 0$$

This equation is the one that the eddy axial field solver uses to calculate $H$. 
Obtaining Currents

Once the eddy axial field simulator has solved for the vector field $\mathbf{H}$, the current can be obtained by:

$$
\mathbf{J} = \nabla \times \mathbf{H} = \sigma \mathbf{E} + j \omega \varepsilon \mathbf{E} = \mathbf{J}_c + \mathbf{J}_d
$$

where:

- $\mathbf{J}_c$ is the conduction current, $\sigma \mathbf{E}$.
- $\mathbf{J}_d$ is the displacement current, $j \omega \varepsilon \mathbf{E}$.

Note that since $\sigma$, $\varepsilon$, and $\omega$ are known quantities, $\mathbf{E}$ can also be calculated from this equation.

Current Flow (Eddy Axial)

To compute the current flow, the eddy axial field solver uses the following relationship:

$$
I = \int \mathbf{J} \cdot d\mathbf{A}
$$

where:

- $I$ is the current.
- $\mathbf{J}$ is the current density, given by:

$$
\mathbf{J} = \sigma \mathbf{E} + j \omega \varepsilon \mathbf{E}
$$

$A$ is the area over which the current flow is computed. It is found by sweeping the current flow line drawn in the $xy$-plane into the $z$ direction — forming a 3D surface. The current flow computed is the current per meter depth in the $z$ direction.

A separate current flow value is computed for each line you draw.
Axisymmetric Field Simulation

If you selected **RZ Plane** as the model type on the Executive Commands menu, a special version of the selected field solver is used. It assumes that the 2D geometry being studied sweeps 360° around the z-axis of a cylindrical coordinate system, enabling you to model devices like solenoids and insulators that have an axis of rotational symmetry. All electric or magnetic fields in the problem must also be rotationally symmetric so that the solution in any RZ-plane is the same as any other. Axisymmetric field simulation is available for the following solvers:

- Electrostatic
- Magnetostatic
- Eddy Current
- DC Conduction

Any field solution involving a long plank-shaped object is different from a solution involving an annular object — even though the same equation is solved for both types of geometries. Mathematically, the difference between cartesian and axisymmetric field solutions arises from the difference in how gradients, curls and divergences are expressed in the two coordinate systems.

- In a cartesian (xyz) coordinate system, these operators are defined as follows (where \( F \) is a scalar quantity and \( \mathbf{F} \) is a vector quantity):

\[
\nabla F(x, y, z) = \frac{\partial F}{\partial x} \hat{x} + \frac{\partial F}{\partial y} \hat{y} + \frac{\partial F}{\partial z} \hat{z}
\]

\[
\nabla \cdot \mathbf{F}(x, y, z) = \frac{\partial F_x}{\partial x} + \frac{\partial F_y}{\partial y} + \frac{\partial F_z}{\partial z}
\]

\[
\nabla \times \mathbf{F} = \begin{bmatrix}
\hat{x} & \hat{y} & \hat{z}
\end{bmatrix}
\begin{bmatrix}
\frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z}
\end{bmatrix}
\begin{bmatrix}
F_x \\ F_y \\ F_z
\end{bmatrix}
\]
If \( \mathbf{F} \) has a z-component only, the curl is defined as:

\[
\nabla \times \mathbf{F} = \frac{\partial F_z}{\partial y} \hat{x} - \frac{\partial F_z}{\partial x} \hat{y}
\]

• In a cylindrical \((r, \phi, z)\) coordinate system, these operators are defined as:

\[
\begin{align*}
\nabla F(r, \phi, z) &= \frac{\partial F}{\partial r} \hat{r} + \left( \frac{1}{r} \right) \frac{\partial F}{\partial \phi} \hat{\phi} + \frac{\partial F}{\partial z} \hat{z} \\
\nabla \cdot F(r, \phi, z) &= \left( \frac{1}{r} \right) \frac{\partial (r F_r)}{\partial r} + \left( \frac{1}{r} \right) \frac{\partial F_\phi}{\partial \phi} + \frac{\partial F_z}{\partial z} \\

\nabla \times F &= \begin{bmatrix} \hat{r} & r \hat{\phi} & \hat{z} \\ \frac{\partial}{\partial r} & \frac{\partial}{\partial \phi} & \frac{\partial}{\partial z} \\ F_r & r F_\phi & F_z \end{bmatrix}
\end{align*}
\]

If \( \mathbf{F} \) has a \( \phi \)-component only, the curl is defined as:

\[
\nabla \times \mathbf{F} = \left( \frac{1}{r} \right) \left[ -\frac{\partial (r F_\phi)}{\partial z} \hat{r} + \frac{\partial (r F_\phi)}{\partial r} \hat{z} \right]
\]

In the cylindrical case, an extra \( r \) shows up in places where there is no analogous \( x \) in the cartesian case, resulting in a different field solution. The calculations involving the differences in mathematics between axisymmetric and cartesian models are implicitly handled by the Maxwell 2D and are transparent to you.
**Transient Simulation**

*EMpulse only.*

The transient solver allows you to analyze the magnetic fields, energy, force, power loss, speed, and flux of a model at various time steps of a solution over a specified period of time. This solver allows for non-sinusoidal current or voltage excitation, as well as rotational or translational motion. Transient magnetic field solutions follow the same principles as magnetostatic field solutions.

**Assumptions**

The transient solver assumes the following conditions about the problem:

- If motion occurs in the model, no motion occurs outside the band object.
- Only one type of motion (translational or rotational) occurs in the model.
- More than one object can be assigned identical motion within the band object.
Time-Dependent Magnetic Field Simulation

The time-dependent magnetic equation is expressed as:

\[ \nabla \times \mathbf{u} \nabla \times \mathbf{A} = J_s - \sigma \frac{dA}{dt} - \sigma \nabla v + \nabla \times H_c + \sigma \nabla \times \nabla \times \mathbf{A} \]

where:

- \( H_c \) is the coercivity of the permanent magnet.
- \( v \) is the velocity of the moving parts.
- \( A \) is the magnetic vector potential.
- \( J_s \) is the source current density.

EMpulse applies a reference frame that is fixed with respect to the components in the model by setting the velocity, \( \mathbf{v} \), equal to zero. Because the moving components have now been fixed to their own coordinate system, the partial time derivative becomes the total time derivative of \( \mathbf{A} \). Thus, the motion equation becomes:

\[ \nabla \times \mathbf{u} \nabla \times \mathbf{A} = J_s - \sigma \frac{dA}{dt} - \sigma \nabla v + \nabla \times H_c \]

which is obtainable at each time step at every node in the finite element model.
Stranded Conductors

Stranded conductors lack eddy current behavior and are considered to be filaments too thin to model in a practical finite element grid. Because of this, EMpulse assumes that their contribution to the current density is averaged over the area of problem region.

These filaments can be connected in parallel or series, and carry the same current based on:

\[ \nabla \times \mathbf{u} \nabla \times \mathbf{A} = \mathbf{J}_s \]

where \( \mathbf{J}_s \), the uniformly distributed current density is given by:

\[ \mathbf{J}_s = d_f \frac{N_f i_f}{S_f \cdot a \cdot p} \]

and:

- \( i_f \) is the total terminal current flowing into a filament coil group (or winding).
- \( N_f \) is the total conductor number of the filaments in the winding.
- \( A \) is the magnetic vector potential.
- \( a \) is the number of parallel branches in the winding.
- \( d_f \) is the polarity (+1 or -1) to represent forward or return paths.
- \( S_f \) is the total area of the cross-section of the region occupied by the winding.
- \( p \) is the ratio of the original full model to the field domain to be solved.

Note that the filaments in the winding may reside at several different locations as long as they are connected in series or parallel.

The voltage seen from the terminal of the winding is the sum of the voltage across all the wires as well as any possible external impedance based on:

\[ d_f \frac{N_f i_f}{S_f \cdot a} \int \frac{dA}{dt} \cdot d\Omega + R' \cdot i_f + L' \cdot \frac{di_f}{dt} + u_c = u_s \]

and:

\[ R' = r_{dc} + r_{end} + r_{ext} \]

\[ L' = l_{end} + l_{ext} \]
where:

- $i_f$ is the total terminal current flowing into the winding.
- $N_f$ is the total conductor number of the filaments in the winding.
- $a$ is the number of parallel branches in the winding.
- $d_f$ is the polarity (+1 or -1) to represent forward or return paths.
- $S_f$ is the total area of the cross-section of the region occupied by the winding.
- $p$ is the ratio of the original full model to the field domain to be solved.
- $r_{dc}$ is the total DC resistance.
- $r_{end}$ is the end-turn resistance of the winding
- $r_{ext}$ is the external resistance associated with the wire.
- $u_c$ is the voltage drop across the capacitor.
- $l$ is the thickness of the model.
- $A$ is the magnetic vector potential.

In the global system of equations, there is one equation form for each winding. In order to avoid a second order differential equation due to the inclusion of a capacitor, EMpulse solves for the capacitor voltage, $u_c$, based on:

$$i_f - C \frac{du_c}{dt} = 0$$

where:

- $C$ is the capacitance.
- $i_f$ is the total terminal current flowing into the winding.
Solid Conductors

Solid conductors are large enough to model with finite elements, where skin effects depend not only on the frequency of the system, but on the location of nearby conductors.

Based on Ampere’s Law, the total current density, $J_t$, in the system is given by:

$$ J_t = -\sigma \frac{dA}{dt} - \sigma \nabla V $$

which reduces to:

$$ J_t = -\sigma \frac{dA}{dt} + \frac{1}{l} V_b $$

or:

$$ J_t = J_e + J_s $$

where:

- $V_b$ is the voltage difference across the conductors end points.
- $J_e$ is the eddy current density.
- $J_s$ is the source current density.
Solid Conductors with Current Sources

For solid conductors with a current source, the total current is known, while the source component is unknown. EMpulse computes the current source based on the following circuit equation for the \( n \)th conductor:

\[
\int \int_{\Omega_c} \left( -\frac{\sigma}{dI} + J_s \right) d\Omega = \int \int_{\Omega_c} J_t d\Omega = I_t
\]

where:
- \( \Omega_c \) is the width cross-section of the \( n \)th conductor.
- \( I_t \) is the known total current.
- \( J_s \) is the source component to be solved for.
- \( J_t \) is the total current density.

Solid Conductors with Voltage Sources

For solid conductors with a voltage source, the total voltage is known, while the total current density is unknown. EMpulse computes the unknown quantities based on the following circuit equation which is derived from the solid conductor equations. The resulting equation yields:

\[
\int \int_{\Omega_c} \left( \sigma \frac{dA}{dl} + J_t \right) d\Omega = \int \int_{\Omega_c} \sigma V_b d\Omega
\]

where:
- \( \Omega_c \) is the width cross-section of the \( n \)th conductor.
- \( V_b \) is the known voltage source between the two conductors.
- \( J_t \) is the total current density to be solved for.
- \( \sigma \) is the conductivity.
- \( l \) is the thickness of the model.
- \( A \) is the magnetic vector potential.
Translational Motion

The transient motion simulator generates translational motion solutions based on the following motion equation:

\[ ma + \lambda s = F_{\text{comp}} + F_{\text{load}} \]

where:
- \( m \) is the mass of the object, in kg.
- \( a \) is the acceleration of the object, in m/s\(^2\).
- \( s \) is the speed, in m/s.
- \( F \) is the force, in N.
- \( \lambda \) is the damping, in N·s/m.
Rotational Motion

The transient motion simulator generates rotational motion solutions based on the following motion equation:

\[ J\beta + \lambda \omega = T_{\text{comp}} + T_{\text{load}} \]

where:

- \( J \) is the inertial force, in kg m\(^2\).
- \( T \) is the torque, in N\( \cdot \)m.
- \( \omega \) is the angular speed, in rad/s.
- \( \beta \) is the angular acceleration, in rad/s\(^2\).
- \( \lambda \) is the damping factor, in N\( \cdot \)m\( \cdot \)s.
Phasor Notation

Time varying quantities that have the form:

\[ F(t) = F_m \cos(\omega t + \theta) \]

can be represented as rotating phasors in the complex plane. Using Euler’s formula:

\[ e^{j\alpha} = \cos \alpha + j \sin \alpha \]

If \( \alpha = \omega t + \theta \), \( F(t) \) equals the real portion of \( e^{j(\omega t + \theta)} \):

\[ F(t) = \Re [F_m e^{j(\omega t + \theta)}] = \Re [F_m (\cos(\omega t + \theta) + j \sin(\omega t + \theta))] = F_m \cos(\omega t + \theta) \]

Each time-varying quantity has the form \( F_m e^{j\theta} e^{j\omega t} \). The \( F_m e^{j\theta} \) component is merely a complex constant that can be represented by a stationary phasor in the complex plane. The \( F_m e^{j\omega t} \) component is a complex number that depends on \( t \), and can be represented as a rotating phasor in the complex plane, as shown here.

The phasor’s projection on the real axis oscillates sinusoidally. It reaches a peak when parallel with the real axis, and crosses zero when parallel with the imaginary axis. Thus, a phasor with \( \theta = 90^\circ \) represents a quantity that peaks 90 degrees after a phasor with \( \theta = 0^\circ \).
Real and Imaginary Components

Maxwell 2D expects you to enter magnitudes and phase angles when you specify voltages, currents, and other boundary or source quantities in models where time-varying fields are to be computed (that is, eddy current, AC conduction, and eddy axial models). But when the magnitude and phase angle of a quantity are functions of position, it is easier to specify the functions in terms of real and imaginary components “x+jy” rather than in terms of magnitude and amplitude. Therefore, when you specify currents and boundary conditions as functions, the system expects you to describe the functions in terms of real and imaginary components.

The “x+jy” description of a phasor indicates that the phasor is the sum of two components — a sinusoid that peaks at $\omega t=0^\circ$ and a sinusoid that peaks at $\omega t=90^\circ$.

- The “x” component of the phasor is called the real component and can be represented by a phasor that lies on the real axis of the complex plane.
- The “y” component is called the imaginary component and is represented by a phasor that lies on the j-axis of the complex plane.

The real and imaginary components of a three-phase system are shown below:

The real and imaginary components are related to the magnitude and phase of a sinusoid
in this way:

\[ I_m = \sqrt{x^2 + y^2} \]

\[ \theta = \text{atan} \left( \frac{y}{x} \right) \]
Maxwell 2D — Index

A
AC Conduction
AC conduction solver
  admittance
  boundary conditions for
  field quantities in plane calculator
  material properties in plane calculator
  phase of quantities in line calculator
  phase of quantities in number calculator
  phase of quantities in plane calculator
  purpose of
  selecting for a model
  sources for
  theory
accessing Maxwell 2D
active conductors
active window
Adaptive Analysis
adaptive pass number
adaptive pass, specifying number of
adaptive solution
Add Graphs (parametric Plot command)
Add (Constraint command)
Add (Variables command)
adding text to models
adjacent conductors
admittance
admittance, normalized values
Align (Reshape/Vertex command)
alignment
  of spline control points
  of text
  of vertices
Ampere’s Law
angle

displaying
  line of sight
  measuring
Animate (Variables command)
animating
  parametric setups
  selected setups
  the model
anisotropy
  conductivity (AC Conduction and Eddy Axial)
  imaginary relative permeability (AC Conduction and Eddy Axial)
  imaginary relative permeability (Magnetostatic and Eddy Current)
  permeability (AC Conduction and Eddy Axial)
  permeability (Magnetostatic and Eddy Current)
  permittivity (Electrostatic, AC Conduction and Eddy Axial)
  permittivity (Magnetostatic and Eddy Current)
tensor
Arc (Object command)
  arcs
    changing number of segments in
    defining as line calculator registers
Arrange commands
  Mirror
  Move
  Rotate
Arrow Display (Post/Plane commands)
arow plots
  density of arrows in rectangular region
  displaying in a rectangle
  of common vector field quantities
### Maxwell 2D — Index

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
<th>C</th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>E</td>
<td>F</td>
</tr>
<tr>
<td>G</td>
<td>H</td>
<td>I</td>
</tr>
<tr>
<td>J</td>
<td>K</td>
<td>L</td>
</tr>
<tr>
<td>M</td>
<td>N</td>
<td>O</td>
</tr>
<tr>
<td>P</td>
<td>Q</td>
<td>R</td>
</tr>
<tr>
<td>S</td>
<td>T</td>
<td>U</td>
</tr>
<tr>
<td>V</td>
<td>W</td>
<td>X</td>
</tr>
<tr>
<td>Y</td>
<td>Z</td>
<td>#</td>
</tr>
</tbody>
</table>

#### Thumbtabs:

- **A** B C
- **D** E F
- **G** H I
- **J** K L
- **M** N O
- **P** Q R
- **S** T U
- **V** W X
- **Y** Z #

#### Contents

- of field quantities in plane calculator
- redisplaying
- Arrow Region (Post/Plane commands)
- Arrow (Post/Plane commands)
- Assign commands
  - Boundary
  - Source
- Assign/Boundary commands
  - Balloon
  - Impedance
  - Master
  - Resistance
  - Slave
  - Symmetry
  - Value
- Assign/Source commands
  - Sheet
  - Solid
- Attributes (Edit command)
  - axes
    - alignment of boundary and source quantities with alignment of vector material properties with displaying for BH-curves
  - axis of symmetry
  - axisymmetric models
    - and anisotropic materials
    - axis of symmetry
    - electric flux linkage
  - field simulation for integrating over a volume
  - magnetostatic flux linkage
  - multiplying and dividing by multiplying and dividing by r outside boundaries for scalar components in line calculator
  - scalar components in number calculator
  - scalar components in plane calculator
  - specifying magnetic vector potential for value boundaries in vector components in line calculator
  - vector components in number calculator
  - vector components in plane calculator
  - axisymmetric models, differences from cartesian models
B

B Vector

background
assigning material properties, boundaries and sources to
automatically excluded (DC conduction)
default material for
definition of
effect of ballooning on
excluding from model
balloon boundaries
and default current return path
assigning
charge at infinity
in AC conduction models
in DC conduction models
in eddy axial models
in eddy current models
in electrostatic models
in magnetostatic models
in transient models
used as reference
voltage at infinity

Balloon (Assign/Boundary commands)
batch processing
batch.log file (NT and Win95)
batch.log file (UNIX)
command line
directory paths
error messages (UNIX)
error messages (Windows NT and Windows 95)
for PC (Microsoft Windows)
for UNIX workstations
of nominal problem

of parametric problem
parametric model
project names
script files
setting up models for
batch.log file
for UNIX workstations
for Windows NT and Windows 95

BH Examine (Post commands)
BH Plot (Post commands)
BH-curve
adding points to
axis settings for
changing appearance of
deleting
displaying coordinates of
entering
file formats for
for permanent magnets
from Maxwell 3D
importing from a file
magnetic coercivity of
magnetic retentivity of
modifying B and H values
plotting saturation levels on
saving
units for

Boolean commands
Intersect
Simplify
Subtract
Union

boundaries
deleting
deselecting
selecting
<table>
<thead>
<tr>
<th>Boundary Conditions</th>
<th>Maxwell 2D — Index</th>
</tr>
</thead>
<tbody>
<tr>
<td>and field behavior</td>
<td>boundary conditions, AC conduction balloon default (natural and Neumann) matching (master and slave) symmetry (even and odd) value</td>
</tr>
<tr>
<td>and size of drawing region assigning defining via math functions deleting during matrix solutions effect of grouping on effect on impedance computation effect on inductance computation even symmetry general procedure in axisymmetric models master modifying Neumann odd symmetry picking edges for matching boundaries resetting to default slave summary of use of value</td>
<td></td>
</tr>
<tr>
<td>assigning</td>
<td>boundary conditions, AC conduction balloon default (natural and Neumann) matching (master and slave) symmetry (even and odd) value</td>
</tr>
<tr>
<td>defining via math functions</td>
<td>boundary conditions, DC conduction balloon default (Neumann and natural) matching (master and slave) resistance symmetry (even and odd) value</td>
</tr>
<tr>
<td>deleting</td>
<td>boundary conditions, electrostatic balloon default (natural and Neumann) matching (master and slave) symmetry (even and odd) value</td>
</tr>
<tr>
<td>during matrix solutions</td>
<td>boundary conditions, functional general procedure for defining identifying constant boundary quantities identifying functional boundary quantities orientation of function on boundary</td>
</tr>
<tr>
<td>effect of grouping on</td>
<td>boundary conditions, eddy axial balloon default (natural and Neumann) matching (master and slave) symmetry (even and odd) value</td>
</tr>
<tr>
<td>effect on impedance computation</td>
<td>boundary conditions, eddy current balloon default (natural and Neumann) impedance matching (master and slave) symmetry (even and odd) value</td>
</tr>
<tr>
<td>effect on inductance computation</td>
<td>boundary conditions, eddy current symmetry (even and odd) value</td>
</tr>
<tr>
<td>even symmetry</td>
<td>boundary conditions, functional general procedure for defining identifying constant boundary quantities identifying functional boundary quantities orientation of function on boundary</td>
</tr>
<tr>
<td>general procedure in axisymmetric models</td>
<td>boundary conditions, functional general procedure for defining identifying constant boundary quantities identifying functional boundary quantities orientation of function on boundary</td>
</tr>
<tr>
<td>master</td>
<td>boundary conditions, functional general procedure for defining identifying constant boundary quantities identifying functional boundary quantities orientation of function on boundary</td>
</tr>
<tr>
<td>modifying Neumann</td>
<td>boundary conditions, functional general procedure for defining identifying constant boundary quantities identifying functional boundary quantities orientation of function on boundary</td>
</tr>
<tr>
<td>odd symmetry</td>
<td>boundary conditions, functional general procedure for defining identifying constant boundary quantities identifying functional boundary quantities orientation of function on boundary</td>
</tr>
<tr>
<td>picking edges for matching boundaries resetting to default slave summary of use of value</td>
<td>boundary conditions, functional general procedure for defining identifying constant boundary quantities identifying functional boundary quantities orientation of function on boundary</td>
</tr>
<tr>
<td>resetting to default slave</td>
<td>boundary conditions, functional general procedure for defining identifying constant boundary quantities identifying functional boundary quantities orientation of function on boundary</td>
</tr>
<tr>
<td>symmetry (even and odd) value</td>
<td>boundary conditions, functional general procedure for defining identifying constant boundary quantities identifying functional boundary quantities orientation of function on boundary</td>
</tr>
</tbody>
</table>

Maxwell Online Help System Copyright © 1988-2001 Ansoft Corporation
Calc commands
- Line
- Number
- Plane
calculators
- line
- number
- plane
Calc/Plane/Decompose
Calc/Plane/Export
capacitance
- and electric field
- calculating
- in terms of charges and voltages
- in terms of currents and time-varying voltages
- normalized values
- short circuit
cartesian models
- differences from axisymmetric models
cells
- copying
- resetting
centroid refinement
Change View (Window command)
changing field of view
charge
- and balloon boundaries
- and reference voltage
- assigning to conductors
- assigning to non-conductors
- assigning to surfaces of dielectrics
- on floating conductors
- on non-conducting objects
seeded mesh for
- solid
charge density
- in non-conductors
- on charge sheets
charge sheet
- charge density
- distribution of charge on
- total charge
charge sources
- functional
- solid
circles
- approximated by polygons
- changing number of segments
circumcircle refinement
Clear
Clear (Boundary Manager)
Clear (Edit command)
Close (File command)
closed objects
Closed Objects (Edit/Select command)
coarse mesh
coenergy
- and virtual force
- and virtual torque
coefficency
electric
magnetic
Color (Model/Defaults command)
colors
- changing
- of field plots
- of geometric objects
- of text (PC)
- of text (workstation)
<table>
<thead>
<tr>
<th>Thumbtabs:</th>
<th>Maxwell 2D — Index</th>
</tr>
</thead>
<tbody>
<tr>
<td>A B C D E F G H I J K L M N O P Q R S T U V W X Y Z #</td>
<td>in parametric sweeps</td>
</tr>
<tr>
<td></td>
<td>modifying</td>
</tr>
<tr>
<td>Constraint/Add commands</td>
<td>flux linkage</td>
</tr>
<tr>
<td>Arc Radius</td>
<td>number of triangles</td>
</tr>
<tr>
<td>Line-To-Line Angle</td>
<td>power loss</td>
</tr>
<tr>
<td>Lock X Coordinate</td>
<td>remaining adaptive passes</td>
</tr>
<tr>
<td>Lock Y Coordinate</td>
<td>target energy error</td>
</tr>
<tr>
<td>Point-To-Point Distance</td>
<td>torque</td>
</tr>
<tr>
<td>Rotation</td>
<td>total energy</td>
</tr>
<tr>
<td>summary</td>
<td>viewing</td>
</tr>
<tr>
<td>Constraint/Delete commands</td>
<td>Convergence Display</td>
</tr>
<tr>
<td>All</td>
<td>Plot Current</td>
</tr>
<tr>
<td>By Clicking</td>
<td>Plot Flux</td>
</tr>
<tr>
<td>By Name</td>
<td>Plot Force</td>
</tr>
<tr>
<td>summary</td>
<td>Plot Percent Error Energy</td>
</tr>
<tr>
<td>Constraint/Modify commands</td>
<td>Plot Torque</td>
</tr>
<tr>
<td>By Clicking</td>
<td>Plot Total Energy</td>
</tr>
<tr>
<td>Edit Variables</td>
<td>Plot Total Power Loss</td>
</tr>
<tr>
<td>summary</td>
<td>Plot Triangles</td>
</tr>
<tr>
<td>Contour Display</td>
<td>Table</td>
</tr>
<tr>
<td>contour plots</td>
<td>coordinate system</td>
</tr>
<tr>
<td>of common field quantities</td>
<td>axisymmetric vs. cartesian</td>
</tr>
<tr>
<td>of quantities in the plane calculator</td>
<td>effect on field simulation</td>
</tr>
<tr>
<td>Convergence</td>
<td>Coordinate System (Window command)</td>
</tr>
<tr>
<td>convergence</td>
<td>coordinate system, global</td>
</tr>
<tr>
<td>of field solution</td>
<td>measurements</td>
</tr>
<tr>
<td>viewing during parametric solution</td>
<td>resetting local coordinate system to</td>
</tr>
<tr>
<td>convergence data</td>
<td>coordinate system, local</td>
</tr>
<tr>
<td>completed adaptive passes</td>
<td>for boundaries and sources</td>
</tr>
<tr>
<td>convergence criteria</td>
<td>for functional and vector material properties</td>
</tr>
<tr>
<td>current flow</td>
<td>moving objects in</td>
</tr>
<tr>
<td>delta energy</td>
<td>resetting</td>
</tr>
<tr>
<td>displaying as a table</td>
<td>rotating</td>
</tr>
<tr>
<td>displaying plots of</td>
<td>shifting</td>
</tr>
<tr>
<td>energy error</td>
<td>uv</td>
</tr>
<tr>
<td>error energy</td>
<td>coordinates</td>
</tr>
<tr>
<td></td>
<td>displaying x- and y-</td>
</tr>
<tr>
<td></td>
<td>displaying x- and y-offsets</td>
</tr>
</tbody>
</table>
## Maxwell 2D — Index

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
<th>C</th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>E</td>
<td>F</td>
</tr>
<tr>
<td>G</td>
<td>H</td>
<td>I</td>
</tr>
<tr>
<td>J</td>
<td>K</td>
<td>L</td>
</tr>
<tr>
<td>M</td>
<td>N</td>
<td>O</td>
</tr>
<tr>
<td>P</td>
<td>Q</td>
<td>R</td>
</tr>
<tr>
<td>S</td>
<td>T</td>
<td>U</td>
</tr>
<tr>
<td>V</td>
<td>W</td>
<td>X</td>
</tr>
<tr>
<td>Y</td>
<td>Z</td>
<td>#</td>
</tr>
</tbody>
</table>

### Contents

- Copy (Edit command)
- core loss solutions
- coupling coefficient
- CPU time, required for solution tasks
- cross products
  - in the number calculator
  - in the plane calculator
- curl, axisymmetric vs. cartesian
- current
  - and capacitance
  - and inductance
  - and virtual force
- constraints on passive conductors
- instantaneous
- peak
- relationship to current density (eddy current solver)
- return path
- source
- time varying, and inductance
- current density
  - AC
  - DC (magnetostatic)
  - displacement currents
- downloading into plane calculator (AC)
- downloading into plane calculator (Eddy Axial)
- downloading into plane calculator (Eddy Current)
- eddy currents
- functional
  - in Ampere’s Law
  - in eddy current problems
  - in magnetostatic problems
  - in solid DC sources
  - in stranded AC current sources

### Go Back

- on AC current sheet
- on AC current sheet (Assign command)
- on DC current sheet
- on DC current sheet (Assign command)
- plotting
- relationship to electric field (DC conduction)
- source current
- current distribution
  - effect on inductance and resistance
  - in eddy current simulations
  - in magnetostatic simulations
  - seeding mesh for

### Current Flow

- current flow
  - and convergence
  - for DC conduction solver
  - for eddy axial solver
  - for eddy current (AC magnetic) solver
  - in axisymmetric models
  - in cartesian models
  - plotting for each adaptive solution
  - viewing solution
- current mesh
- current sheet, AC
  - and eddy currents
  - current density
  - current distribution
  - magnitude of current
  - phase of current
  - total current
- current sheet, DC
  - current density
  - total current
- current sources, AC
  - assigning to touching conductors
  - constraints on current in solid and parallel
<table>
<thead>
<tr>
<th>D</th>
</tr>
</thead>
<tbody>
<tr>
<td>D Vector</td>
</tr>
<tr>
<td>Data commands (table editor)</td>
</tr>
<tr>
<td>Fill</td>
</tr>
<tr>
<td>Sort</td>
</tr>
<tr>
<td>summary</td>
</tr>
<tr>
<td>Sweep</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>DC Conduction</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC conduction solver</td>
</tr>
<tr>
<td>axisymmetric vs. cartesian models</td>
</tr>
<tr>
<td>boundary conditions for</td>
</tr>
<tr>
<td>conductance</td>
</tr>
<tr>
<td>current flow</td>
</tr>
<tr>
<td>field quantities in plane calculator</td>
</tr>
<tr>
<td>purpose of</td>
</tr>
<tr>
<td>selecting for a model</td>
</tr>
<tr>
<td>sources for</td>
</tr>
<tr>
<td>theory</td>
</tr>
<tr>
<td>when to use</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>DC Magnetic Fields package</th>
</tr>
</thead>
<tbody>
<tr>
<td>Defaults (Global commands)</td>
</tr>
<tr>
<td>Define Model</td>
</tr>
<tr>
<td>Define Model commands</td>
</tr>
<tr>
<td>Draw Model</td>
</tr>
<tr>
<td>Group Objects</td>
</tr>
<tr>
<td>Define Model, for parametric solutions</td>
</tr>
<tr>
<td>Define (Post/Line commands)</td>
</tr>
<tr>
<td>Delete (parametric Plot command)</td>
</tr>
<tr>
<td>Delete (Variables command)</td>
</tr>
<tr>
<td>Deleting Mesh Seeding Operations</td>
</tr>
<tr>
<td>delta energy</td>
</tr>
<tr>
<td>derived materials</td>
</tr>
<tr>
<td>adding</td>
</tr>
<tr>
<td>deleting</td>
</tr>
<tr>
<td>inheritance of properties</td>
</tr>
</tbody>
</table>
Maxwell 2D — Index

- deriving materials
- Deselect All (Boundary manager)
- Deselect All (Edit command)
- dielectrics
  - AC surface voltage on charge on surface of charges on
  - DC surface voltage on (DC Conduction)
  - DC surface voltage on (Electrostatic)
  - polarization of specifying charge in time constant
- dimensions
  - measuring specifying units of
- dipole moment
- displacement current
  - and total source current
- downloading into plane calculator (AC)
  - downloading into plane calculator (Eddy Axial)
    - downloading into plane calculator (Eddy Current)
  - in solid and parallel AC current sources relationship to AC current density relationship to AC electric field
- Display (Global commands)
- Display (Post/Line commands)
- displaying objects
distance
  - between points
  - between points in geometric model
  - measuring
distributed matrices
- divergences
  - axisymmetric vs. cartesian
dot products
  - in the number calculator
  - in the plane calculator
- Drawing Model
- Drawing command
- Drawing Plane
- drawing plane
- Drawing Plane (Model commands)
- drawing region
  - and background objects
  - and boundary conditions specifying
  - use of in solution
- Drawing Size (Model command)
Maxwell 2D — Index

E

E Vector
  AC/DC Conduction
  Eddy Axial
  Electrostatic

Eddy Axial
  eddy axial solver
    boundary conditions for current flow
    derivation of eddy axial field equations
    field quantities in plane calculator
    material properties in plane calculator
    phase of quantities in line calculator
    phase of quantities in number calculator
    phase of quantities in plane calculator
    purpose of selecting for a model
    sources for theory
    virtual force
    virtual torque

Eddy Current
  eddy current
    and skin depth
    and total source current
    downloading into plane calculator
    effect on AC inductance and resistance
    in solid and parallel AC current sources
    relationship to AC current density
    relationship to AC magnetic field
  eddy current field quantities
  eddy current solver
    axisymmetric vs. cartesian models
    boundary conditions for current flow
    derivation of field equations
    field quantities in plane calculator

impedance
  phase of quantities in line calculator
  phase of quantities in number calculator
  phase of quantities in plane calculator
  purpose of selecting for a model
  sources for theory
  virtual force
  virtual torque

edge voltage

Edge (Reshape command)
edges
  adding vertex points to
  assigning sources to
  deleting
  in objects with curved lines
  picking for matching boundaries

Edit commands
  Attributes
  Clear
  Copy
  Cut
  Delete Row
  Duplicate
  Insert Row
  Paste
  Show All
  Undo Clear
  Visibility

Edit commands (Boundary Manager)
  Clear
  Deselect All
  Select
  Undo Clear

Edit commands (table editor)
## Maxwell 2D — Index

<table>
<thead>
<tr>
<th>Clear</th>
<th>Edit/Select/Objects commands (Boundary Manager)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Copy</td>
<td>By Area</td>
</tr>
<tr>
<td>Cut</td>
<td>By Clicking</td>
</tr>
<tr>
<td>Delete Row</td>
<td>By Name</td>
</tr>
<tr>
<td>Deselect All</td>
<td>Edit/Visibility commands</td>
</tr>
<tr>
<td>Insert Row</td>
<td>By Item</td>
</tr>
<tr>
<td>Paste</td>
<td>Hide Selection</td>
</tr>
<tr>
<td>summary</td>
<td></td>
</tr>
<tr>
<td>Edit commands (2D Modeler)</td>
<td>electric coercivity relationship to permittivity, polarization and electric retentivity</td>
</tr>
<tr>
<td>Deselect All</td>
<td></td>
</tr>
<tr>
<td>Select</td>
<td>electric field and capacitance and electric coercivity behavior in floating conductors behavior in perfect conductors behavior on even symmetry boundaries behavior on matching boundaries behavior on odd symmetry boundaries downloading into plane calculator (DC) downloading into plane calculator (Electro-static) in axisymmetric models in cartesian models plotting AC electric field DC electric field required sources of static Electric Fields package electric flux and polarization flux linkage electric flux density and electric coercivity and electric retentivity</td>
</tr>
<tr>
<td>Edit menu</td>
<td></td>
</tr>
<tr>
<td>Edit/Attributes commands</td>
<td></td>
</tr>
<tr>
<td>By Clicking</td>
<td></td>
</tr>
<tr>
<td>Recolor</td>
<td></td>
</tr>
<tr>
<td>Rename</td>
<td></td>
</tr>
<tr>
<td>All Projects</td>
<td></td>
</tr>
<tr>
<td>Current Project</td>
<td></td>
</tr>
<tr>
<td>Edit/Deselect All commands</td>
<td></td>
</tr>
<tr>
<td>All Items</td>
<td></td>
</tr>
<tr>
<td>By Area</td>
<td></td>
</tr>
<tr>
<td>By Name</td>
<td></td>
</tr>
<tr>
<td>Closed Objects</td>
<td></td>
</tr>
<tr>
<td>Model Objects</td>
<td></td>
</tr>
<tr>
<td>NonModel Objects</td>
<td></td>
</tr>
<tr>
<td>Open Objects</td>
<td></td>
</tr>
<tr>
<td>Edit/Duplicate commands</td>
<td></td>
</tr>
<tr>
<td>Along Arc</td>
<td></td>
</tr>
<tr>
<td>Along Line</td>
<td></td>
</tr>
<tr>
<td>Mirror Duplicate</td>
<td></td>
</tr>
<tr>
<td>Edit/Select commands</td>
<td></td>
</tr>
<tr>
<td>By Item</td>
<td></td>
</tr>
<tr>
<td>Hide Selection</td>
<td></td>
</tr>
<tr>
<td>Select</td>
<td></td>
</tr>
<tr>
<td>All Items</td>
<td></td>
</tr>
<tr>
<td>Edit/Select/Objects commands (Boundary Manager)</td>
<td></td>
</tr>
<tr>
<td>Edge</td>
<td></td>
</tr>
<tr>
<td>Object Intersect</td>
<td></td>
</tr>
<tr>
<td>Objects</td>
<td></td>
</tr>
<tr>
<td>Trace</td>
<td></td>
</tr>
</tbody>
</table>
Index

maxwell 2D — Index

downloading into plane calculator (electrostatic)
plotting
AC flux density
DC flux density
electric potential
AC/DC Conduction
downloading into plane calculator
Electrostatic
plotting
AC electric potential
DC electric potential
reference for
setting to a value on a boundary (AC Conduction)
setting to a value on a boundary (DC Conduction)
setting to a value on a boundary (Electrostatic)
tables
theory
electric retentivity
relationship to polarization, permittivity and
electric coercivity
electric susceptibility
Electrostatic
electrostatic solver
axisymmetric vs. cartesian models
boundary conditions for
capacitance
field quantities in plane calculator
flux linkage
purpose of
selecting for a model
sources for
specifying reference for potential
theory
virtual force
when to use
EMPulse
energy
and convergence
and impedance
and virtual force (electrostatic)
delta energy
plotting energy error
plotting total energy
target energy error
energy density, and manual mesh refinement
energy error
and solution accuracy
current
need for manual refinement
plotting for each field solution
stopping criteria
target
Entry (Post/Line commands)
even symmetry boundaries
assigning
behavior of electric field
behavior of magnetic field
behavior of magnetic field in transient models
in AC conduction models
in DC conduction models
in eddy axial models
in eddy current models
in electrostatic models
in magnetostatic models
phase of AC electric field
phase of AC magnetic field
relationship to Neumann boundaries
excluded objects
Maxwell 2D — Index

automatically excluded objects including in model
Executive Commands menu
  Define Model
  Draw Model
  Drawing
  Group Objects
  Model
  Post Processor
  Profile
  Setup Boundaries/Sources
  Setup Executive Parameters
  Setup Materials
  Setup Solution Options
  Solutions
  Solve
  Solver
  Zoom In

executive module
executive parameters commands
external fields
assigning in axisymmetric models
modeling with value boundaries

F
field of view
  adjusting
  changing
  displaying entire drawing space
  displaying small objects
  magnifying
  shifting
  zooming in
field quantities
  displaying instantaneous values of
  loading into registers
  manipulating
  multiplying by material properties
  plotting common quantities
  plotting on a line
  plotting quantities in plane calculator
  values at selected points
field simulation
  axial magnetic fields (AC)
  conduction currents (AC)
  conduction currents (DC)
  eddy currents
  electric potential (DC)
  in cartesian and axisymmetric models
  magnetic fields (AC)
  magnetic fields (DC)
  selecting a field solver
  static electric fields
field solutions, viewing
Fields
File commands
  Close
  Exit
Import
New
Open
Print
Print Setup
Revert
Save
Save As
File commands (Post Processor)
Exit
Open
summary
Trans
File commands (2D Modeler)
file extensions
file formats, Maxwell 2D version 4.33 (.obs)
File menu
files
saving circuit models
saving parametric plots
files, geometry
closing
creating
importing
Maxwell 2D version 4.33 (.obs files)
opening
reverting to previous version
File/Print commands
Active View
Project
Rectangle
File/Trans commands (Post Processor)
Delete
Execute
Start
Stop
Fill (Data command)
fine mesh
finite element mesh
and large dimensional difference
current mesh
displaying
displaying shaded mesh
displaying wireframe mesh
information about
information about (post processor)
initial mesh
intermediary
need for manual refinement
problems in creating
size of
specifying a starting mesh
Fit All (parametric Plot command)
Fit All (Window/Change View command)
Fit Drawing (Window/Change View command)
Fit Selection (Window/Change View command)
floating conductors, defining as charge sources
flux density, electric flux density
flux lines
First Point
Line Color
Line Name
Second Point
Flux Linkage
flux linkage
and convergence
electrostatic
magnetostatic
plotting for each adaptive solution
units of
viewing solution
Force
Maxwell 2D — Index

force
  magnitude
  plotting for each adaptive solution
  viewing solution
virtual (Eddy Current)
virtual (Electrostatic)
virtual (Magnetostatic)
x-component
y-component
Format (parametric Plot command)
frequency
  and impedance
  and skin depth
effects on impedance and admittance computations
specifying for field solutions
specifying for nominal problem
Full Screen
functional materials (for parametric solutions)
functional mechanical values
Functional Properties in RZ Solvers
Functions
  Assign/Boundary and Assign/Source commands
  Setup Materials

G
Gauss’s Law
general operations
  in the line calculator
  in the number calculator
  in the plane calculator
geometric models
  adding text
  as a collection of closed objects
cartesian and axisymmetric
complexity and processing requirements
drawing
general strategy for creating
grouping objects in
in axisymmetric problems
level of precision
problems with large dimensional differences
Global commands
  Defaults
  Display
  Recolor
  Refresh
global parameters
gradients, axisymmetric vs. cartesian
Grid (Window command)
grids
cartesian
displaying
divisions in 2D
mouse
polar
snapping to
spacing between points
3D
grounded systems, modeling with balloon
boundaries

grounds
  effect of grouping on
  in admittance matrix
  in capacitance matrix
  in conductance matrix
  optimizing placement of

Group Objects
  exiting
  general procedure

grouped objects
  and parallel current sources
  assigning materials to
  effect on force and torque solutions
  effect on matrix computations

H

H Vector
hatches
help
  hotkeys
  tool bars

Help commands
  About Help
  Contents
  Index
  On Context
  On Module
  On M2DFS
  Shortcuts

help system
  button commands
  document title
  graphics
  links
  menu bar
  selecting text and graphics
  topics list
  using
  window functions

hiding objects
hotkeys
  Meshmaker
  parametric table
  2D Boundary Manager
  2D Modeler
<table>
<thead>
<tr>
<th>I</th>
</tr>
</thead>
<tbody>
<tr>
<td>icons, tool bar</td>
</tr>
<tr>
<td>imaginary quantities</td>
</tr>
<tr>
<td>representing with phasors</td>
</tr>
<tr>
<td>imaginary quantities, representing with phasors</td>
</tr>
<tr>
<td>imaginary relative permeability</td>
</tr>
<tr>
<td>anisotropic (AC Conduction and Eddy Axial)</td>
</tr>
<tr>
<td>anisotropic (Magnetostatic and Eddy Current)</td>
</tr>
<tr>
<td>impedance</td>
</tr>
<tr>
<td>and AC inductance</td>
</tr>
<tr>
<td>and AC resistance</td>
</tr>
<tr>
<td>and frequency</td>
</tr>
<tr>
<td>and time-varying currents</td>
</tr>
<tr>
<td>and time-varying voltages</td>
</tr>
<tr>
<td>difference in AC and DC computations</td>
</tr>
<tr>
<td>matrix</td>
</tr>
<tr>
<td>normalized values</td>
</tr>
<tr>
<td>impedance boundaries</td>
</tr>
<tr>
<td>and skin depth</td>
</tr>
<tr>
<td>approximation of eddy currents on</td>
</tr>
<tr>
<td>assigning</td>
</tr>
<tr>
<td>behavior of magnetic field on</td>
</tr>
<tr>
<td>relative permeability</td>
</tr>
<tr>
<td>when to use</td>
</tr>
<tr>
<td>Impedance (Assign/Boundary commands)</td>
</tr>
<tr>
<td>Insert (Reshape/Vertex command)</td>
</tr>
<tr>
<td>insulated systems, modeling with balloon boundaries</td>
</tr>
<tr>
<td>insulators, perfect</td>
</tr>
<tr>
<td>integrating</td>
</tr>
<tr>
<td>over a line</td>
</tr>
<tr>
<td>over a volume (axisymmetric models)</td>
</tr>
<tr>
<td>over a 2D region</td>
</tr>
<tr>
<td>Intersect (Boolean command)</td>
</tr>
<tr>
<td>intersecting objects</td>
</tr>
<tr>
<td>intersecting objects, open objects</td>
</tr>
<tr>
<td>isotropic materials</td>
</tr>
</tbody>
</table>

| open circuit |
| Information |
| initial mesh |
| index (2D Modeler) |
| index |
| inductance |
| AC vs. DC |
| and impedance |
| in terms of magnetic flux and currents |
| in terms of voltages and time-varying currents |
| normalized values |
J Vector

keyboard entry keys
  display of displaying (uv) grid (uv)
<table>
<thead>
<tr>
<th>L</th>
<th>lumped matrices</th>
</tr>
</thead>
<tbody>
<tr>
<td>licensing and non-graphical interfaces</td>
<td></td>
</tr>
<tr>
<td>Line</td>
<td></td>
</tr>
<tr>
<td>line calculator</td>
<td></td>
</tr>
<tr>
<td>entering line segments into general operations</td>
<td></td>
</tr>
<tr>
<td>phase</td>
<td></td>
</tr>
<tr>
<td>plotting quantities from register operations</td>
<td></td>
</tr>
<tr>
<td>registers</td>
<td></td>
</tr>
<tr>
<td>scalar operations</td>
<td></td>
</tr>
<tr>
<td>vector operations</td>
<td></td>
</tr>
<tr>
<td>line matching, mesh on master and slave boundaries</td>
<td></td>
</tr>
<tr>
<td>line of sight</td>
<td></td>
</tr>
<tr>
<td>line plots</td>
<td></td>
</tr>
<tr>
<td>changing display of displaying coordinates of of common field quantities of quantities in line calculator saving</td>
<td></td>
</tr>
<tr>
<td>line segments</td>
<td></td>
</tr>
<tr>
<td>arcs</td>
<td></td>
</tr>
<tr>
<td>defining for use in plots and line calculator registers</td>
<td></td>
</tr>
<tr>
<td>deleting</td>
<td></td>
</tr>
<tr>
<td>displaying in post processor entering into line calculator inspecting and modifying object edges straight lines</td>
<td></td>
</tr>
<tr>
<td>linear materials</td>
<td></td>
</tr>
<tr>
<td>logs</td>
<td></td>
</tr>
<tr>
<td>of batch solution tasks of solution tasks</td>
<td></td>
</tr>
</tbody>
</table>
### Maxwell 2D — Index

<table>
<thead>
<tr>
<th>M</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>M</strong></td>
</tr>
<tr>
<td>macros</td>
</tr>
<tr>
<td>adding</td>
</tr>
<tr>
<td>defining</td>
</tr>
<tr>
<td>executing</td>
</tr>
<tr>
<td>in parametric solutions</td>
</tr>
<tr>
<td>removing</td>
</tr>
<tr>
<td>Mag B</td>
</tr>
<tr>
<td>Mag D</td>
</tr>
<tr>
<td>Mag E</td>
</tr>
<tr>
<td>AC/DC Conduction</td>
</tr>
<tr>
<td>Eddy Axial</td>
</tr>
<tr>
<td>Electrostatic</td>
</tr>
<tr>
<td>Mag H</td>
</tr>
<tr>
<td>Mag J</td>
</tr>
<tr>
<td>magnetic coercivity</td>
</tr>
<tr>
<td>in non-linear materials</td>
</tr>
<tr>
<td>in permanent magnets</td>
</tr>
<tr>
<td>relationship to permeability, magnetization</td>
</tr>
<tr>
<td>and magnetic retentivity</td>
</tr>
<tr>
<td>magnetic field</td>
</tr>
<tr>
<td>and magnetic coercivity</td>
</tr>
<tr>
<td>and saturation</td>
</tr>
<tr>
<td>behavior inside perfect conductors</td>
</tr>
<tr>
<td>behavior on balloon boundaries</td>
</tr>
<tr>
<td>behavior on even symmetry boundaries</td>
</tr>
<tr>
<td>behavior on odd symmetry boundaries</td>
</tr>
<tr>
<td>displaying values at selected points</td>
</tr>
<tr>
<td>downloading into plane calculator (Eddy Current)</td>
</tr>
<tr>
<td>downloading into plane calculator (Magneto-static)</td>
</tr>
<tr>
<td>in Ampere’s Law</td>
</tr>
<tr>
<td>in axisymmetric models (AC)</td>
</tr>
<tr>
<td>in axisymmetric models (DC)</td>
</tr>
<tr>
<td>in cartesian models (AC)</td>
</tr>
<tr>
<td>in cartesian models (DC)</td>
</tr>
<tr>
<td>phase on even symmetry boundaries</td>
</tr>
<tr>
<td>phase on odd symmetry boundaries</td>
</tr>
<tr>
<td>plotting DC magnetic fields</td>
</tr>
<tr>
<td>relationship to electric field (eddy current solver)</td>
</tr>
<tr>
<td>relationship to magnetization</td>
</tr>
<tr>
<td>required sources of</td>
</tr>
<tr>
<td>setting to a value on a boundary</td>
</tr>
<tr>
<td>setting to a value on a boundary (Assign command)</td>
</tr>
<tr>
<td>static</td>
</tr>
<tr>
<td>units</td>
</tr>
<tr>
<td>magnetic field simulation, time-dependent</td>
</tr>
<tr>
<td>magnetic flux</td>
</tr>
<tr>
<td>magnetic flux density</td>
</tr>
<tr>
<td>and magnetic retentivity</td>
</tr>
<tr>
<td>and saturation</td>
</tr>
<tr>
<td>displaying values at selected points</td>
</tr>
<tr>
<td>downloading into plane calculator</td>
</tr>
<tr>
<td>in non-linear materials</td>
</tr>
<tr>
<td>plotting</td>
</tr>
<tr>
<td>relationship to magnetization</td>
</tr>
<tr>
<td>units</td>
</tr>
<tr>
<td>magnetic flux, and inductance</td>
</tr>
<tr>
<td>magnetic losses</td>
</tr>
<tr>
<td>magnetic moment</td>
</tr>
<tr>
<td>magnetic retentivity</td>
</tr>
<tr>
<td>in non-linear materials</td>
</tr>
<tr>
<td>in permanent magnets</td>
</tr>
<tr>
<td>magnetic susceptibility</td>
</tr>
<tr>
<td>magnetic vector potential</td>
</tr>
<tr>
<td>downloading into plane calculator (Eddy Current)</td>
</tr>
<tr>
<td>downloading into plane calculator (Magneto-static)</td>
</tr>
</tbody>
</table>

Maxwell Online Help System

Copyright © 1988-2001 Ansoft Corporation
<table>
<thead>
<tr>
<th>Thumbtabs:</th>
<th>Maxwell 2D — Index</th>
</tr>
</thead>
<tbody>
<tr>
<td>A B C D E F G H I J K L M N O P Q R S T U V W X Y Z #</td>
<td>static) in axisymmetric models on balloon boundaries reference for required sources of setting to a value on a boundary (Eddy Current) setting to a value on a boundary (Magnetostatic) magnetization as a vector function as source of magnetic field direction of relationship to permeability, magnetic coercivity and retentivity Magnetostatic magnetostatic field quantities magnetostatic solver axisymmetric vs. cartesian models boundary conditions for field quantities in plane calculator flux linkage inductance material properties in plane calculator purpose of selecting for a model sources for specifying reference for potential theory virtual force virtual torque Magnify magnifying view windows Manual Mesh manual mesh refinement master boundaries matching mesh on slave boundaries phase of AC magnetic field Master (Assign/Boundary commands) matching boundaries and excluded objects assigning master boundaries assigning slave boundaries in AC conduction models in DC conduction models in eddy axial models in eddy current models in electrostatic models in magnetostatic models in transient models matching mesh points on phase of AC electric field on master and slave boundaries phase of AC magnetic field on master and slave boundaries restrictions on master boundaries restrictions on slave boundaries selecting edges between vertex points material characteristics anisotropic defaults dependencies between electrostatic material characteristics dependencies between magnetostatic material characteristics derived (inheritance) editable functional isotropic linear multiplying or dividing field quantities by need for</td>
</tr>
<tr>
<td>A</td>
<td>B</td>
</tr>
<tr>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>D</td>
<td>E</td>
</tr>
<tr>
<td>G</td>
<td>H</td>
</tr>
<tr>
<td>J</td>
<td>K</td>
</tr>
<tr>
<td>M</td>
<td>N</td>
</tr>
<tr>
<td>P</td>
<td>Q</td>
</tr>
<tr>
<td>S</td>
<td>T</td>
</tr>
<tr>
<td>V</td>
<td>W</td>
</tr>
<tr>
<td>Y</td>
<td>Z</td>
</tr>
</tbody>
</table>

### Perfect Conductors
- **Vector**

### Material Characteristics, AC Conduction
- **Anisotropic Conductivity**
- **Anisotropic Permittivity**
- **Conductivity**
- **Relative Permittivity**

### Material Characteristics, DC Conduction
- **Conductivity**

### Material Characteristics, Eddy Axial
- **Anisotropic Conductivity**
- **Anisotropic Imaginary Permeability**
- **Anisotropic Permeability**
- **Anisotropic Permittivity**
- **Conductivity**
- **Imaginary Permeability**
- **Relative Permittivity**

### Material Characteristics, Eddy Current
- **Anisotropic Conductivity**
- **Anisotropic Permeability**
- **Anisotropic Permittivity**
- **Anisotropic Relative Permittivity**
- **Conductivity**
- **Imaginary Permeability**
- **Relative Permeability**
- **Relative Permittivity**

### Material Characteristics, Electrostatic
- **Anisotropic Permittivity**
- **Conductivity**
- **Dependent and Independent Material Characteristics**
- **Electric Coercivity**
- **Electric Retentivity**
- **Polarization**
- **Relative Permittivity**

### Material Characteristics, Functional
- **Alignment with Local Coordinate System**
- **Assigning**
- **Dependent and Independent Material Properties**
- **Identifying Functional and Constant Properties**
- **Specifying Direction of**
- **Specifying Origin of**
- **Vector Functions**

### Material Characteristics, Magnetostatic
- **Anisotropic Permeability**
- **Dependent and Independent Material Characteristics**
- **Magnetic Coercivity**
- **Magnetic Retentivity**
- **Magnetization**
- **Non-Linear Permeability**
- **Relative Permeability**

### Material Database, Global
- **Adding Materials to**
- **Changing Characteristics of Materials**
- **Deleting Derived Materials**
- **Deleting Materials from**

### Material Database, Local
- **Adding Materials To**
- **Changing Characteristics of Materials**
- **Deleting Derived Materials**
- **Deleting Materials From**

### Material Manager
- **Materials**
- **Adding to Local Material Database**
- **Changing Properties of**
- **Deleting from Local Database**
- **Derived**
- **Effect of Grouping On**
- **Linear**
- **Local vs. Global**
- **Lossy**
- **Non-Linear**
<table>
<thead>
<tr>
<th>Thumbtabs:</th>
<th>Maxwell 2D — Index</th>
</tr>
</thead>
<tbody>
<tr>
<td>A B C D E F G H I J K L M N O P Q R S T U V W X Y Z #</td>
<td>non-linear (Solver Residual)</td>
</tr>
<tr>
<td></td>
<td>permanently polarized</td>
</tr>
<tr>
<td></td>
<td>underiving</td>
</tr>
<tr>
<td></td>
<td>materials setup (for parametric solutions)</td>
</tr>
<tr>
<td></td>
<td>math functions</td>
</tr>
<tr>
<td></td>
<td>orientation on boundary</td>
</tr>
<tr>
<td></td>
<td>origin on boundary</td>
</tr>
<tr>
<td></td>
<td>used to define boundaries</td>
</tr>
<tr>
<td></td>
<td>used to define material properties</td>
</tr>
<tr>
<td></td>
<td>used to define sources</td>
</tr>
<tr>
<td></td>
<td>math functions (for boundaries and sources)</td>
</tr>
<tr>
<td></td>
<td>alignment with x- and y-axes</td>
</tr>
<tr>
<td></td>
<td>defining</td>
</tr>
<tr>
<td></td>
<td>deleting</td>
</tr>
<tr>
<td></td>
<td>modifying</td>
</tr>
<tr>
<td></td>
<td>math functions (for materials)</td>
</tr>
<tr>
<td></td>
<td>defining</td>
</tr>
<tr>
<td></td>
<td>deleting</td>
</tr>
<tr>
<td></td>
<td>modifying</td>
</tr>
<tr>
<td></td>
<td>matrices</td>
</tr>
<tr>
<td></td>
<td>coupling coefficient</td>
</tr>
<tr>
<td></td>
<td>distributed</td>
</tr>
<tr>
<td></td>
<td>lumped</td>
</tr>
<tr>
<td></td>
<td>matrix norm</td>
</tr>
<tr>
<td></td>
<td>selecting</td>
</tr>
<tr>
<td></td>
<td>selecting conductors to include in viewing</td>
</tr>
<tr>
<td></td>
<td>matrices, admittance</td>
</tr>
<tr>
<td></td>
<td>coupling coefficient</td>
</tr>
<tr>
<td></td>
<td>distributed</td>
</tr>
<tr>
<td></td>
<td>effect of frequency on</td>
</tr>
<tr>
<td></td>
<td>grounds</td>
</tr>
<tr>
<td></td>
<td>lumped</td>
</tr>
<tr>
<td></td>
<td>signal lines</td>
</tr>
<tr>
<td></td>
<td>units of</td>
</tr>
<tr>
<td></td>
<td>viewing</td>
</tr>
<tr>
<td></td>
<td>matrices, capacitance</td>
</tr>
<tr>
<td></td>
<td>coupling coefficient</td>
</tr>
<tr>
<td></td>
<td>diagonal terms in</td>
</tr>
<tr>
<td></td>
<td>distributed</td>
</tr>
<tr>
<td></td>
<td>grounds</td>
</tr>
<tr>
<td></td>
<td>lumped</td>
</tr>
<tr>
<td></td>
<td>matrix norm</td>
</tr>
<tr>
<td></td>
<td>mutual capacitance</td>
</tr>
<tr>
<td></td>
<td>off-diagonal terms in</td>
</tr>
<tr>
<td></td>
<td>self-capacitance</td>
</tr>
<tr>
<td></td>
<td>sequence of solutions</td>
</tr>
<tr>
<td></td>
<td>setup</td>
</tr>
<tr>
<td></td>
<td>signal lines</td>
</tr>
<tr>
<td></td>
<td>units of</td>
</tr>
<tr>
<td></td>
<td>viewing</td>
</tr>
<tr>
<td></td>
<td>matrices, conductance</td>
</tr>
<tr>
<td></td>
<td>and current</td>
</tr>
<tr>
<td></td>
<td>and voltage drops</td>
</tr>
<tr>
<td></td>
<td>coupling coefficient</td>
</tr>
<tr>
<td></td>
<td>distributed</td>
</tr>
<tr>
<td></td>
<td>grounds</td>
</tr>
<tr>
<td></td>
<td>lumped</td>
</tr>
<tr>
<td></td>
<td>matrix norm</td>
</tr>
<tr>
<td></td>
<td>setup</td>
</tr>
<tr>
<td></td>
<td>signal lines</td>
</tr>
<tr>
<td></td>
<td>units of</td>
</tr>
<tr>
<td></td>
<td>viewing</td>
</tr>
<tr>
<td></td>
<td>matrices, impedance</td>
</tr>
<tr>
<td></td>
<td>coupling coefficient</td>
</tr>
<tr>
<td></td>
<td>current return path</td>
</tr>
<tr>
<td></td>
<td>diagonal terms in</td>
</tr>
<tr>
<td></td>
<td>distributed</td>
</tr>
<tr>
<td></td>
<td>effect of frequency on</td>
</tr>
<tr>
<td></td>
<td>lumped</td>
</tr>
<tr>
<td></td>
<td>matrix norm</td>
</tr>
<tr>
<td></td>
<td>mutual impedance</td>
</tr>
</tbody>
</table>
of diagonal terms in self-impedance
setup
solution process
theory
units of
viewing
matrices, inductance
coupling coefficient
current return path
diagonal terms in
distributed
lumped
matrix norm
mutual inductance
off-diagonal terms in
self-inductance
sequence of solutions
setup
units of
viewing
Matrix
matrix norm
matrix norm, versus coupling coefficient
matrix solvers
Maxwell Control Panel
and Ansoft software packages
and PlotData
and 2D Modeler
relationship to Maxwell 2D
Maxwell 2D
relationship to Maxwell Control Panel
relationship to Maxwell 3D
Maxwell 2D Modeler
Maxwell 3D
available field quantities
importing BH-curves from
problem types
relationship to Maxwell 2D
translating 2D geometries to 3D
Maxwell 3D Field Simulator
types of problems
Maxwell’s equations
Measure (Model command)
Measure (Window commands)
memory, and solutions
menu bar for 2D Modeler
mesh
adding points to
area refinement
aspect ratios of triangles
deleting
displaying shaded triangles
displaying wireframe outlines of
general procedure for refining
general procedure for seeding
generating
information on
refining inside objects
seeding object surfaces
viewing information about
viewing triangles, seed points and objects
Mesh commands
Delete
Display
Information
Line Match
Make
Seed
Mesh Seeding for Parametric Sweeps
meshmaker
Meshmaker hotkeys
<table>
<thead>
<tr>
<th>Thumbtabs:</th>
<th>Maxwell 2D — Index</th>
</tr>
</thead>
<tbody>
<tr>
<td>A B C D E F G H I J K L M N O P Q R S T U V W X Y Z #</td>
<td>Mesh/Seed commands</td>
</tr>
<tr>
<td></td>
<td>Delete</td>
</tr>
<tr>
<td></td>
<td>Object</td>
</tr>
<tr>
<td></td>
<td>QuadTree</td>
</tr>
<tr>
<td></td>
<td>Skin</td>
</tr>
<tr>
<td></td>
<td>Surface</td>
</tr>
<tr>
<td>Mesh/Seed/SaveSeed</td>
<td>message bar (in 2D Modeler)</td>
</tr>
<tr>
<td>Mirror Duplicate (Edit/Duplicate command)</td>
<td>Mirror (Arrange command)</td>
</tr>
<tr>
<td>mirroring</td>
<td>and copying</td>
</tr>
<tr>
<td></td>
<td>objects</td>
</tr>
<tr>
<td></td>
<td>text</td>
</tr>
<tr>
<td>Model</td>
<td>model</td>
</tr>
<tr>
<td></td>
<td>defining (for parametric solutions)</td>
</tr>
<tr>
<td></td>
<td>viewing during parametric solution</td>
</tr>
<tr>
<td>Model commands</td>
<td>Default Color</td>
</tr>
<tr>
<td></td>
<td>Defaults</td>
</tr>
<tr>
<td></td>
<td>Drawing Plane</td>
</tr>
<tr>
<td></td>
<td>Drawing Size</td>
</tr>
<tr>
<td></td>
<td>Measure</td>
</tr>
<tr>
<td></td>
<td>Object Attributes</td>
</tr>
<tr>
<td></td>
<td>SnapToMode</td>
</tr>
<tr>
<td>Model Objects (Edit/Select command)</td>
<td>models, modifying the geometry</td>
</tr>
<tr>
<td>Model/Defaults commands</td>
<td>Color</td>
</tr>
<tr>
<td>modifying models</td>
<td>motion</td>
</tr>
<tr>
<td></td>
<td>mechanical setup</td>
</tr>
<tr>
<td></td>
<td>modifying the transient motion setup</td>
</tr>
<tr>
<td></td>
<td>rotational</td>
</tr>
<tr>
<td></td>
<td>translational</td>
</tr>
<tr>
<td>mouse</td>
<td>displaying coordinates</td>
</tr>
<tr>
<td></td>
<td>functions of buttons (2D Modeler)</td>
</tr>
<tr>
<td></td>
<td>grid snap</td>
</tr>
<tr>
<td></td>
<td>keyboard entry mode</td>
</tr>
<tr>
<td></td>
<td>object snap</td>
</tr>
<tr>
<td></td>
<td>snap-to behavior of</td>
</tr>
<tr>
<td></td>
<td>snap-to behavior of (2D Modeler)</td>
</tr>
<tr>
<td>mouse grid</td>
<td>Move (Arrange command)</td>
</tr>
<tr>
<td>Move (Reshape/Vertex command)</td>
<td>mutual capacitance</td>
</tr>
<tr>
<td></td>
<td>mutual impedance</td>
</tr>
<tr>
<td></td>
<td>mutual inductance</td>
</tr>
</tbody>
</table>
### Maxwell 2D — Index

<table>
<thead>
<tr>
<th>N</th>
<th>names</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>allowable characters in</td>
</tr>
<tr>
<td></td>
<td>assigning to 2D objects</td>
</tr>
<tr>
<td></td>
<td>changing</td>
</tr>
<tr>
<td></td>
<td>changing by clicking</td>
</tr>
<tr>
<td></td>
<td>reserved (Background)</td>
</tr>
<tr>
<td>natural boundaries</td>
<td></td>
</tr>
<tr>
<td>behavior of conduction current on</td>
<td></td>
</tr>
<tr>
<td>behavior of electric field on</td>
<td></td>
</tr>
<tr>
<td>behavior of magnetic field on</td>
<td></td>
</tr>
<tr>
<td>in AC conduction models</td>
<td></td>
</tr>
<tr>
<td>in DC conduction models</td>
<td></td>
</tr>
<tr>
<td>in eddy axial models</td>
<td></td>
</tr>
<tr>
<td>in eddy current models</td>
<td></td>
</tr>
<tr>
<td>in electrostatic models</td>
<td></td>
</tr>
<tr>
<td>in magnetostatic models</td>
<td></td>
</tr>
<tr>
<td>Neumann boundaries</td>
<td></td>
</tr>
<tr>
<td>behavior of conduction current on</td>
<td></td>
</tr>
<tr>
<td>behavior of electric field on</td>
<td></td>
</tr>
<tr>
<td>behavior of magnetic field on</td>
<td></td>
</tr>
<tr>
<td>in AC conduction models</td>
<td></td>
</tr>
<tr>
<td>in DC conduction models</td>
<td></td>
</tr>
<tr>
<td>in eddy axial models</td>
<td></td>
</tr>
<tr>
<td>in eddy current models</td>
<td></td>
</tr>
<tr>
<td>in electrostatic models</td>
<td></td>
</tr>
<tr>
<td>in magnetostatic models</td>
<td></td>
</tr>
<tr>
<td>in transient models</td>
<td></td>
</tr>
<tr>
<td>relationship to even symmetry boundaries</td>
<td></td>
</tr>
</tbody>
</table>

New (File command)
New (parametrics Plot command)
New (Window command)
nominal model
  definition of
  general procedure for creating

relationship to parametric model
non-graphical interfaces
non-linear materials
  adding to database
displaying B and H values
entering BH-curves for
identifying saturated materials
plotting saturation levels in
relationship of B and H in
nonlinear permanent magnets
NonModel Objects (Edit/Select command)
number calculator
  entering field values into
general operations
phase
register commands
scalar operations
vector operations
number of triangles
number registers
Maxwell 2D — Index

O
Object commands
  Arc
  Circle
  Polyline
  Rectangle
  Spiral
  Spline
  Text
object edges, defining as line calculator registers
object information
Object Orientation Display
objects
  adding points and edges
  adding vertex points to
  allowable characters in names
  assigning materials to
  attributes of
  automatically excluded
  background
  changing colors of
  changing scale of
  changing the scale
  changing the view
  circles
  closed
  colors of
  complex shapes
  contained within other objects
  copying
    between projects
  copying along arc
  copying along line
  deleting
deselecting
deselecting (for grouping)
display options
displaying
displaying as wireframe or shaded
displaying in Meshmaker
displaying small ones
displaying with hatches
effects of deleting edges
excluding from model
filling view
grouping
hiding
importing complex objects
including excluded objects
intersecting
lists of
making larger or smaller
mirroring
modeling thin objects
moving
names of
naming
non-model
not appearing
number of triangles in
open
overlapping
pasting
rectangles
refining interior mesh
refreshing
repositioning
reshaping
rotating
saving
seeding interior of
seeding surface of
selecting
selecting with the mouse
selecting (for grouping)
self-intersecting
setting default color
simplifying after boolean operation
specifying the color
spirals
subtracting
temporary
ungrouping
unioning
viewing dimensions of in Meshmaker
visibility
zero-area
Object/Arc commands
  Clockwise
  Counter-Clockwise
Object/Circle commands
  2 Point
  3 Point
Object/Spiral commands
  Circular
  Rectangular
odd symmetry boundaries
  and default current return path
  and voltage
assigning
behavior of electric field
behavior of magnetic field
behavior of magnetic field in transient models
in AC conduction models
in DC conduction models
P

padding percent
panning with scrollbars
parallel current sources
  AC source current in
  and grouped objects
  displacement current in
  eddy current in

Parameters
parameters
  viewing current flow
  viewing flux linkage
  viewing force
  viewing matrices
  viewing torque

parametric analysis
  frequency
  general procedure for performing
  nominal vs. parametric models
  of boundary and source quantities
  of geometric dimensions
  of material properties
  overview

Parametric Analysis module
  accessing
  batch processing
  relationship to Maxwell 2D

parametric model
  batch processing (UNIX)
  batch processing (Windows NT)
  definition of
  general procedure for creating
  relationship to nominal model
  setting up solution for
  parametric plot windows

parametric setup, definition of
parametric sweep, setting up
parametric table hotkeys
passive conductors
  displacement currents in
  eddy currents in
Paste (Edit command)
perfect conductors
  and field solutions
  assigning AC currents to
  assigning AC currents to (Assign command)
  assigning DC currents to
  assigning DC currents to (Assign command)
  assigning voltages to
  behavior of magnetic field
  defining
perfect insulators, excluding
permanent magnets
  linear
  magnetic coercivity
  magnetic retentivity
  magnetization
  non-linear
  used as source of magnetic field
permanent magnets, nonlinear
permeability
  anisotropic (Eddy Axial)
  anisotropic (Magnetostatic and Eddy Current)
  imaginary
  in plane calculator (Eddy Axial)
  in plane calculator (Magnetostatic)
  non-linear
  of free space (Eddy Current)
  of free space (Magnetostatic)
  relationship to skin depth
relative (Eddy Current)
relative (Magnetostatic)
permittivity
anisotropic (Eddy Current)
anisotropic (Electrostatic, AC Conduction and Eddy Axial)
in plane calculator (AC)
in plane calculator (Eddy Axial)
of free space
relative
phase
in multi-phase systems
in the line calculator
in the number calculator
in the plane calculator
instantaneous values of field quantities
of AC magnetic field on even symmetry boundaries
of AC magnetic field on master and slave boundaries
of AC magnetic field on odd symmetry boundaries
of plotted quantities
phasors
notation for
real and imaginary components
used to express current
used to express voltage
phi, angle in line of sight
plane calculator
downloading field quantities into
general operations
phase
plotting quantities from
register operations
scalar operations
vector operations
planes, entering into plane calculator
Plot Options (BH plots)
Plot (parametric) commands
Add Graphs
Create Composite Plot
Delete
Fit All
Format
New
Open
Save As
Show Coordinates
summary
Zoom In
Zoom Out
plotting
convergence data
saturation levels
Plotting Complex Data
Plotting Vector Data
plotting (data files)
plotting a family of curves
specifying interpolation order
plotting (parametric equations)
specifying degree of interpolation
plotting (signals)
adding
cartesian plots
complex numbers
creating new
deleting
formatting
opening existing
polar plots
saving
tool bar
zooming in
zooming out
plotting, fields
aborting
arrow plots
common quantities
contour plots
D Vector
E Vector
entering field values into number calculator
field quantities
from calculator registers
from line calculator
from plane calculator
graphs
line plots
Mag D
Mag E
phase
plot location
selecting window for
shaded plots
spectrum on black and white printers
Plot/Format (parametric) commands
Axes
Graphs
points
and mouse selection
displaying
entering from the keyboard
measuring distance between
selecting from several subwindows
points, spline control
aligning
moving
used in reshaping objects
points, vertex
adding
aligning
moving
snapping to
used in reshaping objects
polarization
as a vector function
direction of
relationship to permittivity, electric retentivity
and coercivity
used as source of electric field
Polyline (Object command)
position
of subwindows
Post commands
BH Examine
BH Plot
Line
Plane
Plot
Post Process (Executive commands)
Nominal Problem
Variables
post processing
nominal problem
parametric problem
transient motion data
Post Processor
exiting
File commands
post processor
post processor macros
adding to the menu
executing by name
in parametric solutions
removing from the menu
summary
viewing results of
PostScript
and plot quality
Post/Line commands
Define
Display
Entry
Plot
Value
Post/Plane commands
Arrow
Arrow Display
Arrow Region
Contour
Contour Display
Shade
Value
potential
power loss
and convergence
plotting for each field solution
precision
printing
active view
and black and white printers
effect of hidden windows
projects
setup
spectrum plots
Profile
project windows
cascading
creating
organizing display of
tiling
versus subwindows
projects
creating
printing
<table>
<thead>
<tr>
<th>Q</th>
<th>R</th>
</tr>
</thead>
<tbody>
<tr>
<td>Quad All</td>
<td>Recolor</td>
</tr>
<tr>
<td>Quad Screen</td>
<td>Recolor (Edit/Attributes command)</td>
</tr>
<tr>
<td></td>
<td>Rectangle (Object command)</td>
</tr>
<tr>
<td></td>
<td>reduced matrices</td>
</tr>
<tr>
<td></td>
<td>and matrix norm</td>
</tr>
<tr>
<td></td>
<td>reference for voltage and magnetic vector potential</td>
</tr>
<tr>
<td></td>
<td>refine area</td>
</tr>
<tr>
<td></td>
<td>Refine commands</td>
</tr>
<tr>
<td></td>
<td>Area</td>
</tr>
<tr>
<td></td>
<td>Object</td>
</tr>
<tr>
<td></td>
<td>Point</td>
</tr>
<tr>
<td></td>
<td>refine number</td>
</tr>
<tr>
<td></td>
<td>refinement</td>
</tr>
<tr>
<td></td>
<td>aborting (for areas)</td>
</tr>
<tr>
<td></td>
<td>aborting (for objects)</td>
</tr>
<tr>
<td></td>
<td>adaptive</td>
</tr>
<tr>
<td></td>
<td>general procedure for adaptive refinement</td>
</tr>
<tr>
<td></td>
<td>general procedure for manual refinement</td>
</tr>
<tr>
<td></td>
<td>goals</td>
</tr>
<tr>
<td></td>
<td>inside objects</td>
</tr>
<tr>
<td></td>
<td>inspecting the mesh</td>
</tr>
<tr>
<td></td>
<td>manual</td>
</tr>
<tr>
<td></td>
<td>overview</td>
</tr>
<tr>
<td></td>
<td>specifying percentage of undoing</td>
</tr>
<tr>
<td></td>
<td>Refresh (current window)</td>
</tr>
<tr>
<td></td>
<td>Refresh (Global)</td>
</tr>
<tr>
<td></td>
<td>register commands</td>
</tr>
<tr>
<td></td>
<td>in the line calculator</td>
</tr>
<tr>
<td></td>
<td>in the number calculator</td>
</tr>
<tr>
<td></td>
<td>in the plane calculator</td>
</tr>
<tr>
<td></td>
<td>registers</td>
</tr>
<tr>
<td></td>
<td>complex</td>
</tr>
</tbody>
</table>
**Maxwell 2D — Index**

<table>
<thead>
<tr>
<th>Thumbtabs:</th>
<th>Maxwell 2D — Index</th>
</tr>
</thead>
<tbody>
<tr>
<td>A B C</td>
<td>in the line calculator</td>
</tr>
<tr>
<td>D E F</td>
<td>in the plane calculator</td>
</tr>
<tr>
<td>G H I</td>
<td>number</td>
</tr>
<tr>
<td>J K L</td>
<td>reading and writing</td>
</tr>
<tr>
<td>M N O</td>
<td>real</td>
</tr>
<tr>
<td>P Q R</td>
<td>scalar</td>
</tr>
<tr>
<td>S T U</td>
<td>vector</td>
</tr>
<tr>
<td>V W X</td>
<td>relative permeability</td>
</tr>
<tr>
<td>Y Z #</td>
<td>and skin depth</td>
</tr>
</tbody>
</table>

|  | anisotropic (Eddy Axial)  |
|  | anisotropic (Magnetostatic and Eddy Current)  |
|  | entering  |
|  | imaginary  |
|  | imaginary (anisotropic, Eddy Axial)  |
|  | imaginary (anisotropic, Eddy Current)  |
|  | non-linear  |
|  | relationship to magnetization, magnetic retentivity and magnetic coercivity  |
|  | relative permeability (Magnetostatic)  |
|  | relative permittivity  |
|  | anisotropic (Eddy Current)  |
|  | anisotropic (Electrostatic, AC Conduction and Eddy Axial)  |
|  | entering  |
|  | relationship to polarization, electric coercivity and electric retentivity  |
|  | Rename (Edit/Attributes command)  |
|  | reserved names (Background)  |
|  | Reset (Window/Coordinate System/Reset command)  |
|  | Reshape commands  |
|  | Edge  |
|  | Scale Selection  |
|  | Vertex  |
|  | Reshape/Edge commands  |

|  | Delete  |
|  | Number of Segments  |
|  | Reshape/Vertex commands  |
|  | Align  |
|  | Insert  |
|  | Move  |
|  | resistance  |
|  | AC vs. DC  |
|  | and conductance  |
|  | and impedance  |
|  | resistance boundaries  |
|  | and excluded objects  |
|  | assigning  |
|  | conductivity  |
|  | location of  |
|  | thickness off  |
|  | voltage on  |
|  | Resistance (Assign/Boundary commands)  |
|  | resistive layers  |
|  | modeling with resistance boundaries  |
|  | modeling with resistance boundaries (Assign command)  |
|  | restoring deleted boundaries and sources  |
|  | restoring deleted items  |
|  | retentivity  |
|  | electric  |
|  | magnetic  |
|  | return path  |
|  | and boundary conditions  |
|  | and impedance computation  |
|  | and inductance computation  |
|  | default  |
|  | identifying an object as  |
|  | Revert  |
|  | root  |
|  | running Maxwell 2D as  |
Maxwell 2D — Index

Rotate (Arrange command)
Rotate (Window/Coordinate System command)
rotational motion
rows
  sorting
RZ Plane

S
saturation
definition of
  plotting in non-linear materials
  selecting points in non-linear materials
Save As (parametric Plot command)
Save As (2D Modeler)
Save (2D Modeler)
saving
circuit models
  2D LINE definitions
scalar fields
scalar operations
  in the line calculator
  in the number calculator
  in the plane calculator
Scale Selection (Reshape command)
Scale to Window or Scale to Problem
scaling
drawing region
  object dimensions
screen
  refreshing (all windows)
  refreshing (current window)
screen layout
scroll bars
scrolling
seed points
  adding to inside of objects
  adding to object surface
  adding within skin depth
  as vertices of triangles
  deleting all seed points
  deleting mesh
  displaying in meshmaker
### Maxwell 2D — Index

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
<th>C</th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>E</td>
<td>F</td>
</tr>
<tr>
<td>G</td>
<td>H</td>
<td>I</td>
</tr>
<tr>
<td>J</td>
<td>K</td>
<td>L</td>
</tr>
<tr>
<td>M</td>
<td>N</td>
<td>O</td>
</tr>
<tr>
<td>P</td>
<td>Q</td>
<td>R</td>
</tr>
<tr>
<td>S</td>
<td>T</td>
<td>U</td>
</tr>
<tr>
<td>V</td>
<td>W</td>
<td>X</td>
</tr>
<tr>
<td>Y</td>
<td>Z</td>
<td>#</td>
</tr>
</tbody>
</table>

**Select commands**
- All Objects
- By Area
- By Name

**Select Matrix Entries**
- parametric solutions

**Self-capacitance**
- self-impedance
- self-inductance

**Setting**
- printer options

**Setup**

**Setup Boundaries/Sources**
- assigning boundaries and sources
- exiting
  - general procedure for defining boundaries and sources
  - modifying existing boundaries and sources
  - using default boundaries and sources

**Setup Boundaries/Sources commands**
- Assign
- File
- Model
- Window

**Setup Boundaries/Sources (Executive commands)**
- for parametric solutions

**Setup Executive Parameters**
- Current Flow
- Flux Linkage
- Force

**Setup Materials (Executive commands)**
- for parametric solutions

**Setup Solution (Executive commands)**
- Options (parametric)
- Shade
- Shaded Mesh
- shaded plots
  - of common field quantities
  - of quantities in the plane calculator

**Show commands**
- Mesh Information
- Shaded Mesh
- Wireframe Mesh

**Show Coordinates (parametric Plot command)**
- signal lines
  - in admittance matrix
  - in capacitance matrix
  - in conductance matrix

**significant digits**

**Simplify (Boolean command)**
<table>
<thead>
<tr>
<th>Thumbtabs:</th>
<th>Maxwell 2D — Index</th>
</tr>
</thead>
</table>
| A B C D E F G H I J K L M N O P Q R S T U V W X Y Z # | sinusoidal values  
size of subwindows  
skin depth  
and conductivity  
and eddy currents  
and frequency  
and magnetic field  
and permeability  
and skin effect  
effect on AC resistance  
seeding mesh in  
slave boundaries  
matching mesh on master boundaries  
phase of AC electric field  
phase of AC magnetic field  
Slave (Assign/Boundary commands)  
smoothing curved lines  
SnapToMode (Model command)  
solenoid  
alternating fields in  
solid current sources  
AC current  
AC source current, displacement current and eddy current  
DC current  
Solid Modeler  
format, saving 2D Modeler files in  
solid modeling  
displaying objects  
solid sources  
Solid (Assign/Source commands)  
solution  
and drawing region  
solution files  
reading into post processor  
solution options  
for nominal solution  
for parametric solutions  
solution passes  
parametric solution  
solution setup  
parametric  
Solutions  
solutions  
aborting  
accuracy  
and boundary conditions  
and computing resources  
and Maxwell’s Equations  
convergence criteria  
deleting mesh  
frequency of  
general procedure for generating  
generating with non-graphical interfaces  
manipulating  
monitoring  
nominal  
parametric  
plotting  
plotting convergence data  
reading from disk  
residual (error)  
setting up parametric sweep  
specifying criteria for  
superimposing  
viewing convergence data  
writing to disk  
Solutions commands  
Current Flow  
Flux Lines  
Flux Linkage  
Force/Torque |
Maxwell 2D — Index

Matrix
Number Registers
solutions, parametric
  aborting
displaying
errors
Solver
  changing field solvers
  selecting a field solver
solver packages
  AC Magnetic Fields
  complete
  Electric Fields
  Parametric Analysis
solver residual
solver type
  AC Conduction
  DC Conduction
  Eddy Axial
  Eddy Current
  Electrostatic
  Magnetostatic
Solve/Nominal Problem (Executive commands)
Solve/Variables (Executive commands)
sort keys
  changing priority
deleting
selecting
Sort (Data command)
source current
downloading into plane calculator
in active conductors
in axisymmetric models
in cartesian models
in passive conductors
relationship to AC current density
source setup
  for parametric solutions
Source (Assign commands)
sources
  and field behavior
  charge
  charge sheet
  current
  current sheet
defining via math functions
deleting
deselecting
during matrix solutions
edge voltage
effect of grouping on
functional
general procedure for assigning
in axisymmetric models
magnetized materials
modifying
polarized materials
required
selecting
sheet
solid
summary of
use of
voltage
sources, AC conduction
  edge voltage
  required
  solid voltage
sources, DC conduction
  edge voltage
  edge voltage (Assign command)
  required
solid voltage
solid voltage (Assign command)
sources, eddy axial
defining via boundary conditions
external magnetic fields
required
sources, eddy current
active conductors
current sheet
current sheet (Assign command)
parallel current
parallel current (Assign command)
passive
perfect current
perfect current (Assign command)
required
solid current
solid current (Assign command)
stranded current
stranded current (Assign command)
sources, electrostatic
charge
charge sheet
charge sheet (Assign command)
edge voltage
edge voltage (Assign command)
polarized materials
required
solid charge
solid charge (Assign command)
solid voltage
solid voltage (Assign command)
sources, functional
general procedure for defining
identifying constant source quantities
identifying functional source quantities
orientation of function in source
sources, magnetostatic
current density
current density (Assign command)
current sheets
current sheets (Assign command)
perfect current
perfect current (Assign command)
permanent magnets
required
total current
total current (Assign command)
sources, transient
required
specifying printer settings
Spiral (Object command)
spirals
circular
rectangular
Spline (Object command)
status information
cursor position
entering data with keyboard
snap-to behavior of mouse
unit of length
steady-state conditions
stopping criteria
adaptive passes
energy error
stranded conductors
stranded current sources
AC source current in
Subtract (Boolean command)
subtracting objects
open objects
subwindows
active
active (post processor)
adding new windows
cascading
changing the field of view
changing view of
closing
creating
display of octant only
displaying different views of drawing region
displaying entire model in
displaying selected items in
expanding to fill the screen
field of view
local coordinate system
magnifying
manipulating local coordinate system
organizing display of
positioning
post processor
reconfiguring
refreshing (all)
refreshing (current window)
returning to original size
selecting for plots
selecting points from several
shifting
sizing
specifying as 2D or 3D
tiling
turning off
unzooming
versus project windows
zooming

Suggested Values
superposition

surface charge
susceptance
Sweep (Data command)
symmetry
even
modeling via boundary conditions
odd
symmetry boundaries
behavior of electric field on
behavior of magnetic field in transient models
behavior of magnetic field on
in AC conduction models
in axisymmetric models
in DC conduction models
in eddy axial models
in eddy current models
in electrostatic models
in magnetostatic models
phase of AC magnetic field
Symmetry (Assign/Boundary commands)
| T | AC magnetic field (eddy current) simulation |
|   | DC conduction field simulation          |
|   | DC magnetic field (magnetostatic) simula-|
|   | tion                                   |
|   | eddy axial field simulation             |
|   | electrostatic field simulation          |
|   | thermal coupling                        |
|   | thermal field solver                    |
|   | theta, angle in line of sight           |
|   | time                                    |
|   | time constant                           |
|   | for electrostatic and DC conduction simu-|
|   | lation                                  |
|   | time-stepping analysis                  |
|   | time-varying quantities                 |
|   | real and imaginary components           |
|   | time-varying quantities, representing with pha-|
|   | sors                                   |
|   | tool bar                                |
|   | in boundary manager                     |
|   | in Meshmaker                            |

| Text (Object command) | AC conduction field simulation |
| theory               | AC conduction field simulation |

| Trans commands (Post Processor) | Text (Object command) |
| total energy and solution accuracy | AC conduction field simulation |
| plotting for each field solution   | AC conduction field simulation |

| Table cells | AC conduction field simulation |
| editing     | AC conduction field simulation |
| selecting   | AC conduction field simulation |
| Table columns | AC conduction field simulation |
| selecting   | AC conduction field simulation |
| Table editor | AC conduction field simulation |
| Tool bar    | AC conduction field simulation |
| Table of contents | AC conduction field simulation |
| Selecting   | AC conduction field simulation |
| Table rows  | AC conduction field simulation |
| Selecting   | AC conduction field simulation |
| Target error| AC conduction field simulation |
| Plotting    | AC conduction field simulation |
| Text        | AC conduction field simulation |
| Adding      | AC conduction field simulation |
| Alignment   | AC conduction field simulation |
| Attributes of| AC conduction field simulation |
| Changing    | AC conduction field simulation |
| Color of    | AC conduction field simulation |
| Copying     | AC conduction field simulation |
| Deleting    | AC conduction field simulation |
| For annotating screen captures | AC conduction field simulation |
| Italicizing | AC conduction field simulation |
| Mirroring   | AC conduction field simulation |
| Moving      | AC conduction field simulation |
| Pasting     | AC conduction field simulation |
| Rotating    | AC conduction field simulation |
| Selecting   | AC conduction field simulation |
| Setting default color | AC conduction field simulation |
| Size of     | AC conduction field simulation |
| Unselecting | AC conduction field simulation |
| Visibility  | AC conduction field simulation |

**Index**

**Contents**

**Go Back**
Delete
Execute
Start
Stop
transient boundary conditions
transient field quantities
transient functions
transient motion
  restrictions
transient motion module
transient motion, see motion
transient operations
transient simulation
transient solid conductors
  current sources
  voltage sources
transient solver
  field quantities in plane calculator
transient sources
translational motion
translational motion equations
Translators
triangles
  aspect ratios
  displaying shaded view of
  plotting for each field solution
  point refinement of
  refining in rectangular area
  size of largest and smallest in mesh
  total number of
  total number of vertex points
  viewing
  wireframe display of
  trigonometric functions

U
underiving materials
Undo Clear
Undo Clear (Boundary Manager)
Undo Clear (Edit command)
Union (Boolean command)
unioning objects
  open objects
units
  anisotropic materials
  BH-curves
  by material attributes
  charge sheets
  current sheets
  electromagnetic quantities in plots
  in the 2D Post Processor
  MKS
  nonlinear materials
  of BH-curves
  of length
  perfect conductors
  plotting in the Post Processor
  Post Processing field quantities
  rescaling geometric models
  specifying
  specifying for geometric model
  types
  used in the geometric model
  2D modeler
UnZoom
value boundaries
   and default current return path
   in AC conduction models
   in axisymmetric models
   in DC conduction models
   in eddy axial models
   in eddy current models
   in electrostatic models
   in magnetostatic models
   in transient models
   phase of AC magnetic field
   relationship to odd symmetry boundaries
   used as reference
   used to model external fields
Value (Assign/Boundary commands)
Value (Post/Line commands)
Value (Post/Plane commands)
variables
   adding to the table
   boundary conditions
   defining for nominal and parametric solutions
   deleting from the table
   frequency
   geometric dimensions
   material properties
   selecting for parametric sweep
   setting to different values
   sources
   viewing
   viewing during solution
Variables commands
   Rearrange Columns
Variables commands (table editor)
Add
Animate
Delete
summary
View
Variables (Executive commands)
   after solution sweep
Vector Data Type for Force
vector fields
vector functions
   defining
   specifying vector components
vector material properties
   aligning with a given direction
   alignment with local coordinate system
   assigning
   vector functions
vector operations
   in the line calculator
   in the number calculator
   in the plane calculator
vector quantities
   direction on boundary
   in axisymmetric models
   plotting common field quantities
   plotting on a cross-section
   plotting on a line
   plotting within a rectangular area
Vector Variable Type for Force
vertex points
   aligning
   inserting
   moving
Vertex (Reshape command)
viewing
   field solutions
viewing angle
viewing parameters
viewing solutions
  field solutions
  for current flow
  for flux linkage
  for force
  for matrix
  for torque
virtual force
  and current
  and energy
  and magnetic coenergy
  and virtual work (electrostatic)
  and virtual work (magnetostatic)
effect of grouping on
  instantaneous
  theory (eddy current)
  theory (electrostatic)
  theory (magnetostatic)
time-averaged value of
virtual torque
  and magnetic coenergy
  and virtual force (magnetostatic)
effect of grouping on
  instantaneous
  theory (eddy current)
  theory (magnetostatic)
time-averaged value of
virtual work
  and force (electrostatic)
  and force (magnetostatic)
  and torque (magnetostatic)
voltage
  and balloon boundaries
  and capacitance
and inductance
assigning to edges
downloading into plane calculator
downloading into plane calculator (AC)
downloading into plane calculator (Electrostatic)
functional
magnitude of
on conductors
on perfect conductors
on resistance boundaries
on surface of dielectric
on surface of dielectrics
phase of
reference
required sources or
setting on boundary
time-varying
total
voltage sources, AC
defining
edge voltages
solid
used as reference
voltage sources, DC
defining
edge voltages (DC Conduction)
edge voltages (Electrostatic)
solid (DC Conduction)
solid (Electrostatic)
used as reference
### Maxwell 2D — Index

<table>
<thead>
<tr>
<th>W</th>
</tr>
</thead>
<tbody>
<tr>
<td>Window commands</td>
</tr>
<tr>
<td>Magnify</td>
</tr>
<tr>
<td>Measure</td>
</tr>
<tr>
<td>Refresh</td>
</tr>
<tr>
<td>Setup</td>
</tr>
<tr>
<td>Shift</td>
</tr>
<tr>
<td>UnZoom</td>
</tr>
<tr>
<td>Zoom</td>
</tr>
<tr>
<td>Window commands (2D Modeler)</td>
</tr>
<tr>
<td>Cascade</td>
</tr>
<tr>
<td>Change View</td>
</tr>
<tr>
<td>Close</td>
</tr>
<tr>
<td>Coordinate System</td>
</tr>
<tr>
<td>Fill Solids</td>
</tr>
<tr>
<td>Grid</td>
</tr>
<tr>
<td>New</td>
</tr>
<tr>
<td>Tile</td>
</tr>
<tr>
<td>Toolbar</td>
</tr>
<tr>
<td>Wire Frame</td>
</tr>
</tbody>
</table>

### Window/Cascade commands
- All

### Window/Change View commands
- Fit All
- Fit Drawing
- Fit Selection
- Zoom In
- Zoom Out

### Window/Coordinate System commands
- Reset
- Rotate
- Shift

### Window/Setup commands
- Full Screen
- On-Off
- Quad All
- Quad Screen
- Settings

### Window/Tile commands
- All

### Windows
- active
- adjusting the view
- printing active
- project
- specifying default settings
- subwindow

### Windows, project
- cascading
- tiling

### Windows, subwindows
- cascading
- tiling

### Window/Cascade commands
- All

### Windows, project
- cascading
- tiling

### Windows, subwindows
- cascading
- tiling

### Window/Cascade commands
- All

---

**Projects**

**Subwindows**

---

**Window/Change View commands**

**Fit All**

**Fit Drawing**

**Fit Selection**

**Zoom In**

**Zoom Out**

---

**Window/Coordinate System commands**

**Reset**

**Rotate**

**Shift**

---

**Window/Setup commands**

**Full Screen**

**On-Off**

**Quad All**

**Quad Screen**

**Settings**

---

**Window/Tile commands**

**All**

---

**Windows**

**active**

---

**Windows, project**

**cascading**

---

**Windows, subwindows**

**cascading**

---

**Window/Cascade commands**

**All**

---

**Register to disk**
X
x- and y- offsets, displaying
XY Plane

Z
Zoom
Zoom In (Window/Change View command)
Zoom Out (Window/Change View command)
### Numerics

- 2 Point (Object/Circle command)
- 2D Boundary Manager
  - hotkeys
  - menu commands
  - tool bar
- 2D Meshmaker
- 2D Meshmaker commands
  - File
  - Mesh
  - Model
  - Refine
  - Windows
- 2D Modeler
  - accessing from Maxwell 2D
  - accessing from Utilities panel
  - active windows
  - commands
  - drawing region
  - general procedure
  - hotkeys
  - menu bar
  - message bar
  - printing parameters
  - project windows
  - screen layout
  - status bar
  - subwindows
  - tool bar
- 2D modeler
- 3 Point (Object/Circle command)