SimulMag

Micromagnetic Simulation Software

User's Manual Addendum

John O. Oti
Electromagnetic Technology Division
National Institute of Standards and Technology
Boulder, Colorado 80303
July 1998

Introduction

This addendum reflects software updates from version 1.0 through Beta version 2.0g. Enclosed in parenthesis next to each topic below, is the relevant section of the version 1.0 manual to which the update being described applies to.

1. Simulation specifications window (Section. 3.6.3)

Reach this window by pressing the **Modify** button in the **External Field Controls** group of the **System Display** window. This window replaces the **Field Data** dialog of version 1.0. The window contains two groups of controls: The **Field Data** group and the **Parametric Simulations Specifications** group. The latter group is enabled only when the **Generate parametric data** box is checked.

The *Field Data* group

The **Field Data** group allows the specification of the external field to be applied to the system based on the selection from the **Run Method** list. Data entry fields of this group are reconfigured accordingly to accept data that is appropriate for the selected run method. The **Run Method** list contains the following entries: "Single", "Loop", "Rotation" and "Field Pulse". The “Field Pulse”entry is available only for dynamic simulations; for static simulations only the first three entries are available. A field pulse has a base value of zero and is characterized by an amplitude, a rise time, a time duration at amplitude value, a fall time and a field direction (that is defined by entries in the **Azimuth (Deg.)** and **Polar (Deg.)** boxes).

The *Parametric Simulations Specifications* group

This group of controls is used to specify how the value of a design property (the parameter) is to be systematically varied over repeated simulation runs. Graphical output for a parametric simulation is in the form of a three-dimensional surface plot composed of a family of output curves, each corresponding to a specific value of the parameter (see the topic "Parametric Output" below). Parametric calculations are valid only for "Loop" and "Field Pulse" run methods (see above). Two types of parametric outputs are possible: Output that is obtained by using a parameter which is a property of an element in the system, and output that is obtained by using a parameter that is an applied external field property. These plot types are specified by choosing either the **For magnetic system** or the **For external fields** selection in the **Surface Plot type** group. Data entry fields are configured appropriately depending on the type of surface plot selected. The desired parameter to vary can be selected from the **Parameter** list. The **Parameter** list box is filled with appropriate names of possible parameters that can be varied depending on the chosen surface plot type. The **Group** and **Element** list are used to specify the element of what group whose parameter is to be varied. Entries in the **Start value** and **End value** boxes give the range of variation of the chosen parameter. The number of equal steps by which the parameter is varied is entered in the **No. of sweeps** box. Each simulation corresponding to a particular parameter value is referred to as a "scan." At the beginning of each scan the system is

reset to its initial state prior to the start of the parametric simulation.

2. Parametric Output (Section 3.8)

Output data is displayed differently for parametric and non-parametric simulations. Non-parametric output corresponds to the traditional Simulmag output method in which windows displaying single graphs of requested output are launched during the simulation. Each output is calculated for a group and can be specified in the **Device Group Editor** window (see Section 3.8 of version 1.0 manual). For parametric output, a new kind of window is displayed for each of the specified output data. The caption of each such window describes the output it represents. This window consists of three rectangular light-colored boxes for displaying graphical output. The upper-left box labeled "Surface (conventional)", displays the family of generated parametric curves making up the surface plot. The upper-right box labeled "Cut Trace", displays points representing a projection of the locus of the points of the surface plot where it is cut by planes that are oriented parallel to the coordinate planes. The selections Cartesian and Polar below this box are used to specify the type of plot to be displayed. Clicking the right mouse button anywhere within the surface-plot and cut-trace boxes displays a floating menu that can be used to save or print the displayed data. When saving designs displaying parametric outputs using the main program menu, these outputs are saved in the design file together with the rest of the design. Opening such a file re-displays the parametric outputs. Cartesian traces can always be generated for all manners of parametric plots. However polar plots are meaningful only for those simulations involving an angular parameter. The software detects this and enables or disables the **Polar** bullet selection accordingly. The bottom-right box displays the curve of an on-going scan. Surrounding this box are a variety of information describing the nature of the simulation parameter -- its current value, the current scan sequence and the total number of scans. At the completion of each scan, its curve is transferred to the surface plot graph. Pressing the **Next Scan** button at any time terminates the currently running scan and starts the next available one. The entire simulation is terminated by pressing the **Cancel** button. Whenever any point in the cut-trace or scan-curve boxes is clicked on using the left mouse button, the coordinates of that point is displayed in the grooved box region located below the cut-trace box.

The generation of cut traces is controlled by the controls in the group labeled "Cutting Plane." Checking the **Show Cut** box displays a cutting plane on the surface-plot box. Cut points are displayed where the surface and cutting plane meet, and a projection of the cut points is displayed in the **Cut Trace** box. The location of cut points are obtained by a linear interpolation solution of the problem of a surface intersected by a plane. The orientation and position of the cutting plane are specified by the **X-Y**, **Y-Z** and **X-Z** option selections and the slider control. The **Value** box displays the current position of the cutting plane. The display of coordinate axes and the grids on the surface can be accomplished by checking the **Show axes** and **Show grid** boxes.

3. Group motion description for dynamic simulations (Section 3.7.)

In version 1.0 one type of window was used to specify group motion for both dynamic and static simulations. This window is launched by pressing **Group Motion Specs** button in the **Device Group Editor** window. In the new update, a window similar to the old window is still used when specifying motion in static simulations. For dynamic simulations a slightly different window with caption "Dynamic:Device motion description..." is launched. In this window the magnitude of velocity is entered in m/s. Much of the remaining controls of the window such as **Azimuthal**, **Polar**, **Add New Leg** etc. function exactly the same as their counterparts in version 1.0. The time entered in the **Time duration of motion steps (ns):** box determines, together with the velocity, the distance the group will cover during each step of the motion leg. During a calculation, the group will be moved whenever the time step during a current leg expires. Defining a motion leg includes the option of specifying a range of variation of the common current that flows through the group for the duration of the leg. (See Section 4.1. of the version 1.0. manual for a discussion of electrical circuits). This is accomplished by checking the **Enable variation of group common current** box, wherein the **Common current range (mA)** group of controls is enabled. The current will change uniformly over the range entered in the **Start value** and **End value** boxes during the motion leg. Current ranges are defined individually for each leg of motion.

4. Modeling soft magnets.

Beta version 2.0g, includes the modeling of soft-magnetic elements. A soft element is characterized by a relative permeability μ. The component of magnetization along the *i*th system axis is given in S.I. units by

$$M_i = \frac{(\mu\&1)H_i}{1\&(\mu\&1)N_i}$$

where H_i and N_i are respectively the applied field component and the element demagnetization factor along the *i*th axis. This procedure is combined with the usual static and dynamic relaxation methods for calculating the rotation of magnetization vectors. The net result of this is that the magnetization vectors of soft-magnetic elements are able to change in both magnitude and direction. The **Media Characteristics** window (see Section 3.5 of version 1.0 manual) has been updated accordingly to reflect this new feature. The **Magnet Type** list now includes the additional entry "Soft Magnet" which when selected enables the element to be treated as a soft magnet. The **Magnetic and Electrical Properties** group of controls now contains the sub group **Relative permeability** for specifying the permeability of an element. If the **Normal** option is selected, then the permeability entered in the **Value** box is used in above equation in determining the response of the element. Selecting the options **+ infinity** and **- infinity** results in a behavior of the element in which it is always saturated parallel and anti-parallel to the applied field direction.

5. New zoom feature (Section 3.6.2)

Two additional buttons which provide zooming capabilities have been included among the other buttons in the **System-Display Window** next to the status bar. These are the **Zoom** and **Zoom out** buttons. (Floating tool tips identifying these buttons are displayed when the mouse pointer is placed over them). To zoom into a particular region of a displayed design first click on the **Zoom** button, then hold the left mouse button down near the region of interest and drag the mouse over the design while holding the mouse button down. A zoom box is generated, whose contents expands to occupy the complete view space when the mouse button is released. Clicking on the **Zoom out** button forces the whole design to be displayed within the view space.

5. Miscellaneous

A new menu item **Special|Calculation Preferences** is included in the main menu of the program (ref. Section 3.1 of version 1.0 manual). Selecting this item launches a window that (I) allows the user to control the numerical iterative algorithm for calculating the system response and (ii) gives the user the ability to turn off individual field components for groups.

Three new additions have been made to the group data output repertoire (ref. Section 3.4 of version 1.0 manual). These are the three components of the average applied field acting on elements making up a group.