TriComp 5.0

Nelson

RF/AC Magnetic Field and Eddy-current analysis

Field Precision
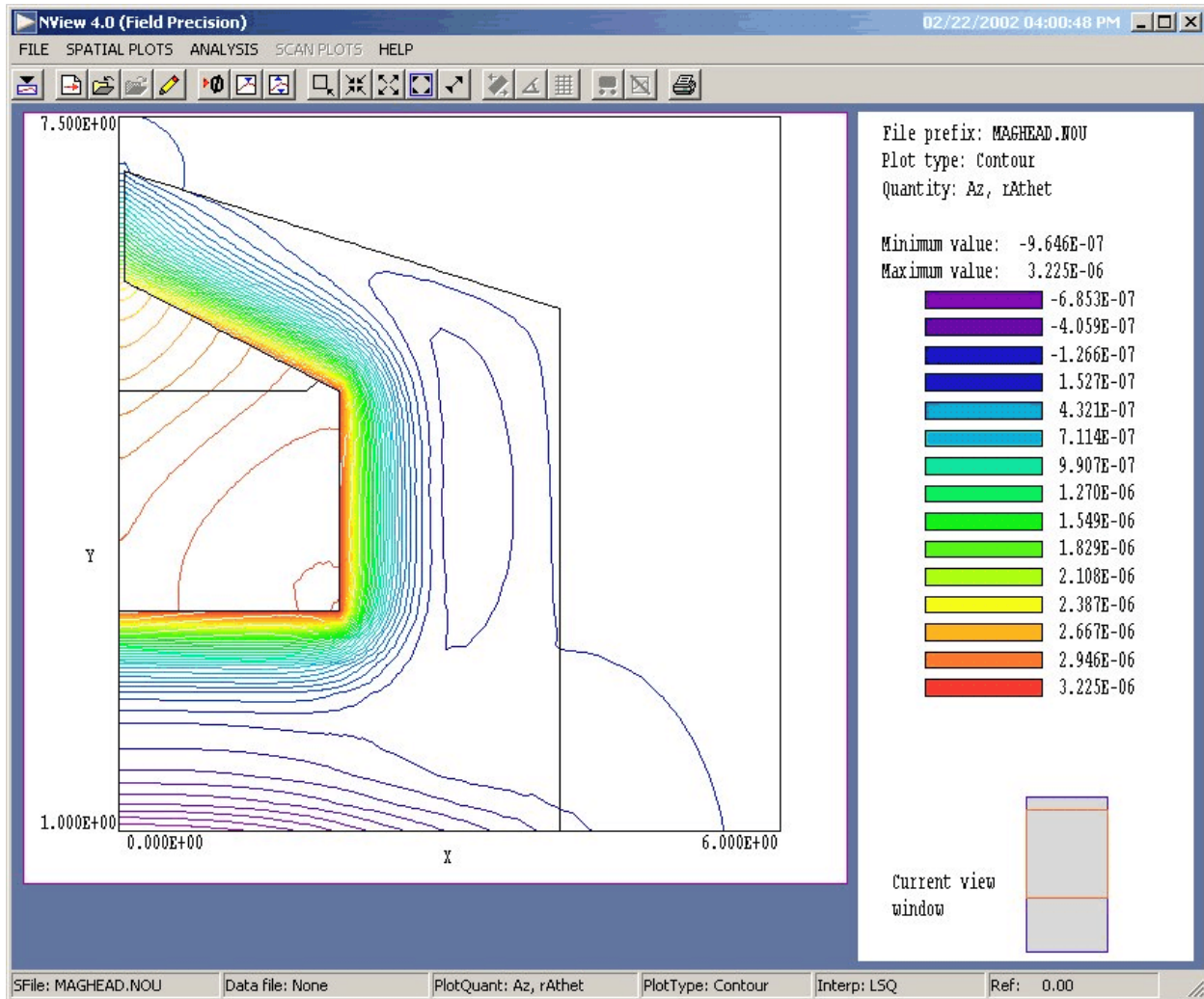Copyright 2004

Nelson 5.0 Manual - Contents

Figure 1. **NView** post-processor work environment

1. Program function

Nelson is a versatile numerical tool to calculate RF and AC magnetic fields in conductive media for complex cylindrical or rectangular geometries. The program applies finite-element methods on a variable-resolution conformal triangular mesh. You can define up to 127 regions to represent coils or different materials. The amplitude and phase of the harmonic coil currents or vector potential on surfaces can be set individually. Conductive materials are characterized by values of the volume resistivity and relative magnetic permeability. **NView**, an interactive graphics analysis program provides full information on the amplitude and phase of the magnetic field and other derived quantities throughout the solution regions. The program makes screen and hardcopy plots of magnetic field lines, electric field amplitude and resistive energy deposition.

3
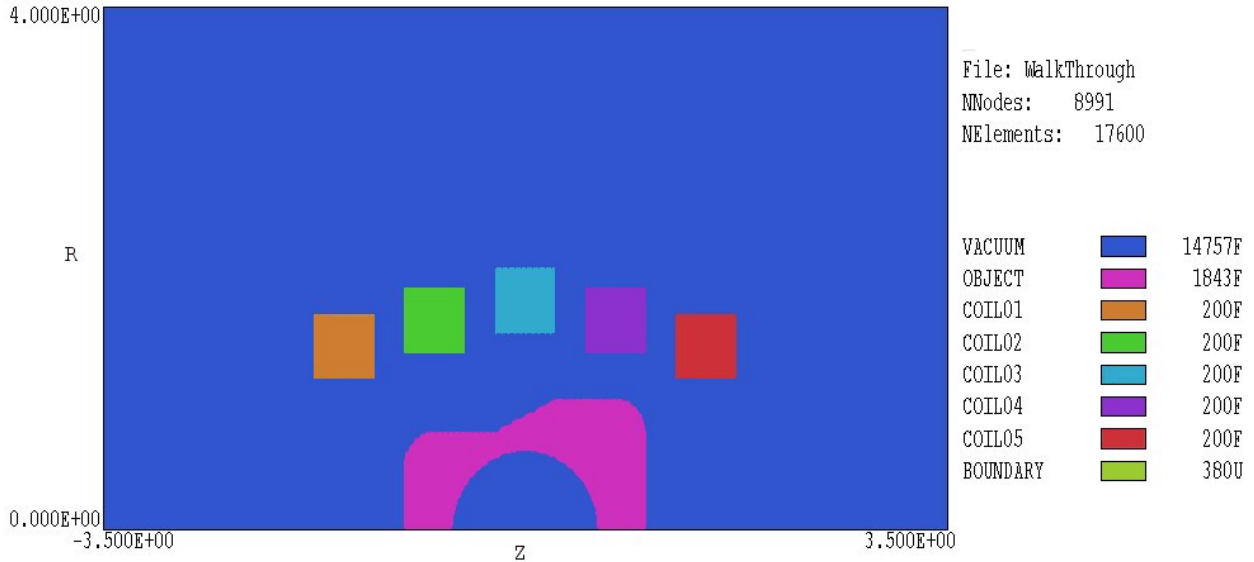
Figure 2. Region divisions for the WALKTHROUGH example.

The frequency-domain solutions of **Nelson** are limited to linear materials and excitation at a single frequency. The Field Precision time-domain **Pulse** code calculates magnetic fields for non-linear magnetic materials with arbitrary drive current waveforms. The theory of finite-element solutions for magnetic fields in the eddy current limit is reviewed in Sect. ##.

2. Walkthough example

We shall consider the *WALKTHOUGH* example included in the **Nelson** library to illustrate a complete solution. The simulation addresses induction heating of a large metal object by a shaped coil. Figure 2 shows the geometry (with dimensions in feet). The system has cylindrical symmetry about the *z* axis (abscissa). The five coils create a solenoid magnetic field at 60 Hz frequency . All coils are in phase and each coil caries a peak current of 500 A. The hollow metal piece has relative magnetic permeability $\mu_r = 3.0$ and volume resistivity $\rho = 120\ \mu\Omega$-cm. In a finite-element solution, a condition must be specified along the boundaries of the solution volume. In this case, we use the condition $rA_\theta = 0.0$, equivalent to perfectly-conducting walls. The extra space around the assembly ensures that the boundary condition has a small effect on fields near the work piece. The goal of the calculation is to determine the distribution of electric and magnetic fields and the total volumetric resistive heating.

4
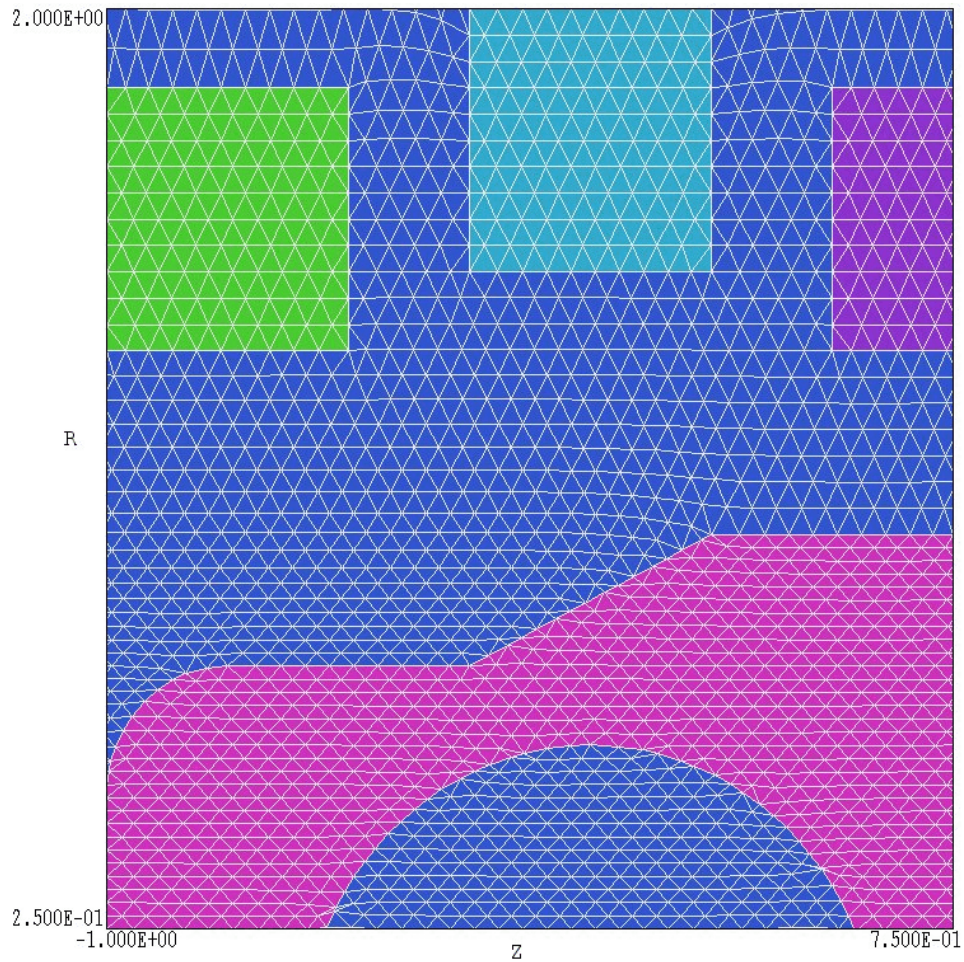
Figure 3. Mesh detail for the WALKTHROUGH example.

Copy the files WALKTHROUGH.MIN, WALKTHROUGH.NIN, and
WALKTHROUGH.SCR to a convenient working directory (*i.e.*,
\TRICOMP\BUFFER). The first step is to generate a file of geometric
information, a required input for **Nelson**. Make sure the data directory in
the **TC** program launcher is set to your working directory, and then run
the **Mesh** program from **TC.** Click on *Load script (MIN)*, pick
WALKTHROUGH.MIN in the dialog. and click *OK*. The **Mesh** manual
describes the format of the file. It contains specifications for the
foundation mesh (*XMesh* and *YMesh* commands) and sets of vectors that
define the shapes in Figure 2. Click on the *Process* command. When mesh
generation is complete, click on *Save mesh (MOU)* to create the file
WALKTHROUGH.MOU. Click on *Plot type* and choose the *Region style* .
Choose *View/Zoom window* and narrow the view to the region near the
work piece (Fig. 3). Note how the size of the elements varies for good
resolution of skin-depth effects on the surface of the work piece.

5

The next step is to run **Nelson** from **TC**. The second required input is the control script `WALKTHROUGH.NIN`. Click on *Edit input files* in the *File* menu and choose `WALKTHROUGH.NIN` to show the following content:

```
* WALKTHROUGH.NIN

Set Freq 60.0
Set ResTarget 2.0E-7
Set Omega 1.25
Set DUnit 3.281
Set MaxCycle 2500
Set Geometry Cylin

Region 1 Material 1.0
Region 2 Material 3.0  120.0E-8
Region 3 Current 500.0
Region 4 Current 500.0
Region 5 Current 500.0
Region 6 Current 500.0
Region 7 Current 500.0
Region 8 Potential 0.0

EndFile
```

The first group of commands beginning with the keyword *SET* controls the solution process. For example, `SET DUNIT 3.281` specifies that dimensions in the **Mesh** file are given in feet. Section ## describes the *SET* commands in detail. The second group of commands beginning with the word *REGION* defines material properties associated with solution regions. The air and coil regions has default fixed values of $\mu_r = 1.0$ and $\rho = \infty$. Each coil region has a total current amplitude 500 A. The current density is distribution uniformly over the cross-section of the region. Region 8 (a line around the outside of the solution volume) has the condition $rA_\theta = 0.0$.

Exit the editor and click the *Start run* command in the *Run* menu. Pick the file `WALKTHROUGH.NIN` and click *OK* to start the solution. **Nelson** employs an iterative relaxation technique to solve the complex-number finite-element equations. The solution takes a few seconds on a high-performance personal computer. When the solution is complete, the program creates the file `WALKTHROUGH.NOU` which contains information on the locations of nodes and the associated values of vector potential or stream function.
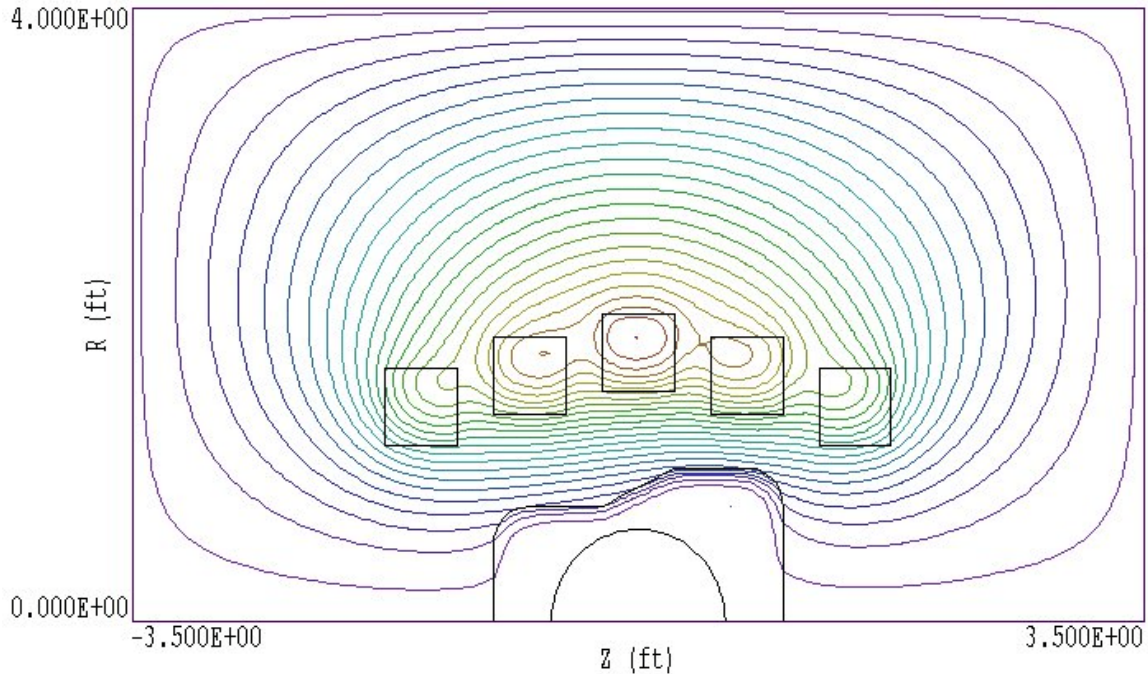
Figure 4. Magnetic field lines in the WALKTHROUGH example.

Run **NView** from **TC** to analyze the solution. Click the *Load solution* command in the *File* menu and pick WALKTHROUGH.NOU. The program creates the default plot of Fig. 4 showing contours of rA_θ. In a two-dimensional solution the contours lie along lines of **B** (magnetic flux density). Because the lines are separated by equal intervals of magnetic flux, the spacing looks larger near the axis of a cylindrical solution. In the *Spatial plot* menu click on *Plot type* and choose *Element*. The program issues a message that plots of $rA\theta$ contours are not supported in the plot type. Click on *Ploted quantity* and choose /*E*/, the amplitude of toroidal electric field. You should see the plot of Fig. 5 which shows electric field penetration into the work piece.

To conclude we shall perform some analysis using the control script feature of **NView**. Click on *File/Run script* and choose WALKTHOUGH.SCR. The file has the content:.

7
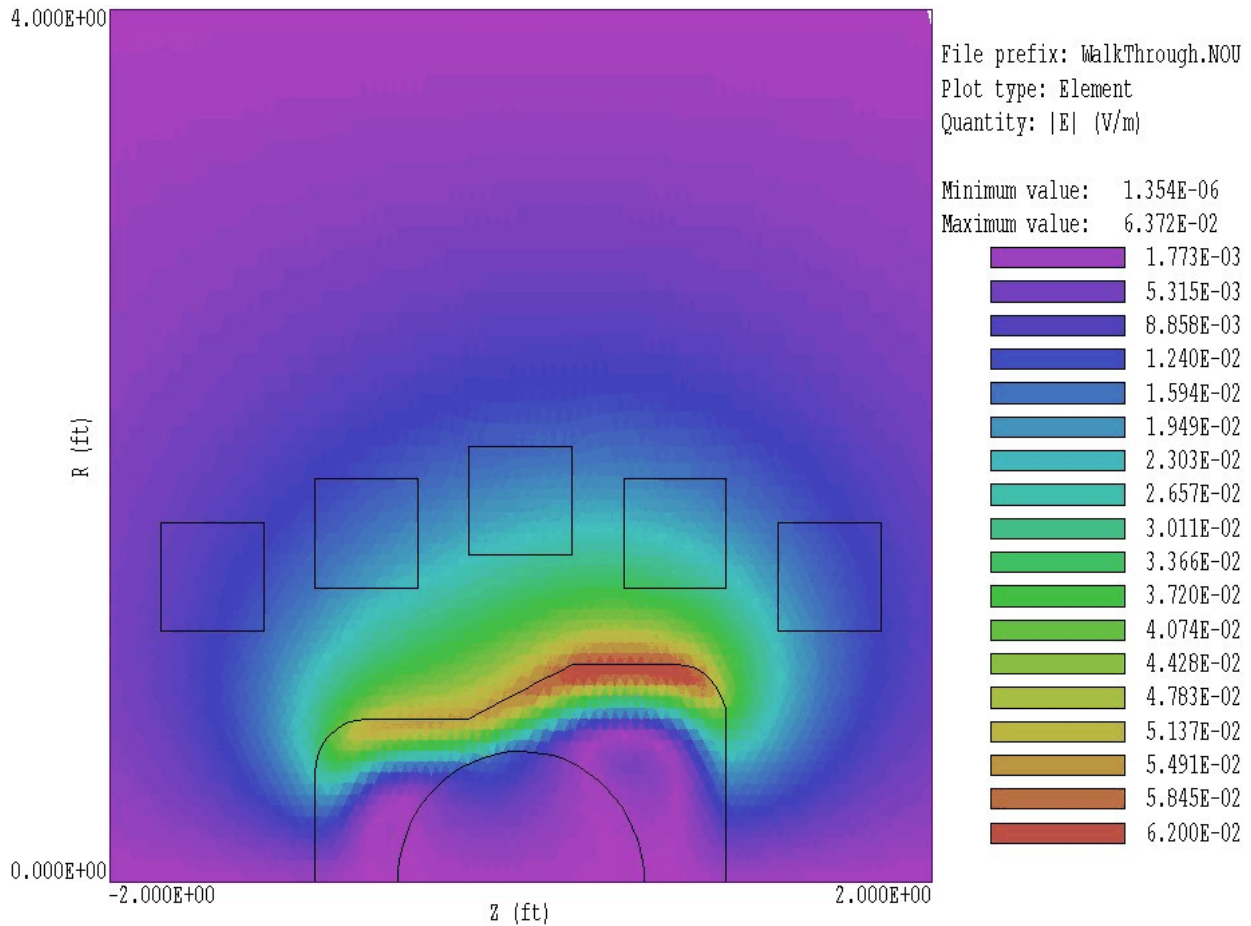
Figure 5. Amplitude of E_θ in the WALKTHROUGH example.

```
* File WALKTHROUGH.SCR
INPUT WalkThrough.NOU
OUTPUT WalkThrough
NSCAN 100
SCAN 0.0 0.0 0.0 1.5
REGION 2
ENDFILE
```

In response, **NView** loads the solution file and opens a text file
WALKTHROUGH.DAT for output. The program sets the number of scan
point to 100 and takes a scan of values at $z = 0.0$ ft from $r = 0.0$ ft to $r =$
1.5 ft. Finally, **NView** does an analysis of region 2 (the work piece) and
closes the output file. It is easy to capture the values in the text file and
transfer them to other software. The plot of $|E_\theta|$ in Fig. 6 was created with
PsiPlot. **NView** also has internal plot capabilities with digital oscilloscope
features. The *Region* command initiates a volume integral and produces
the listing:

8
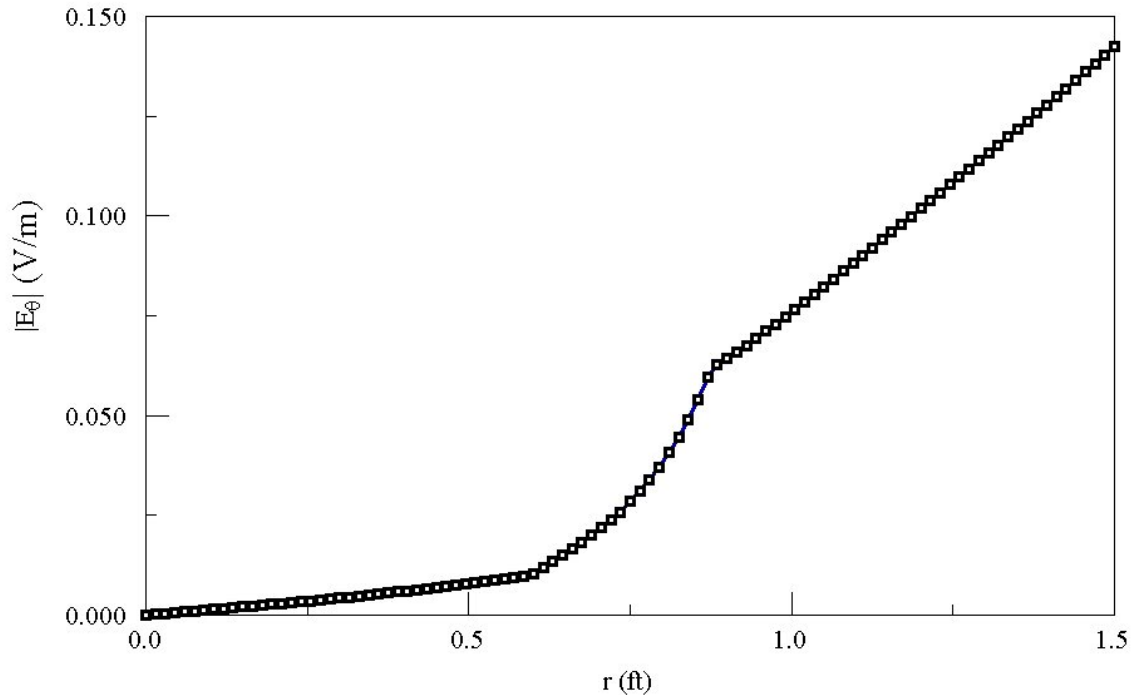
Figure 6. Variation of $|E_\theta|$ with radius at $z = 0.0$. Data created by **NView**, plot created with PsiPlot.

```
   --- Region Information ---
Region No:    2
Volume:   1.090E-01 m3
Energy:   1.563E-12 J
Power:    2.417E+04 W
BPeak:    6.488E-03 tesla
ZPeak:    6.350E-01
RPeak:    9.917E-01
```

The resistive power dissipation is 24.17 kW.

3. Organizing runs

The **Nelson** package consists of a program that computes the physical solution (`Nelson.EXE`) and a dedicated post-processor for analyses (`NView.EXE`). The solution program can run in two modes: interactively in a window or autonomously in the background with the option for batch file control. Similarly, **NView** can run as an interactive application under user control or autonomously under script file control. The autonomous modes allow automatic processing of large or repetitive data sets.

Nelson requires a minimum of two input files.

■ A **Mesh** output file that describes the conformal triangular mesh. The file contains node coordinates and the region numbers of elements and vertices.

■ A command script file that sets control parameters for the solution program and describes the physical properties associated with region numbers.

The mesh output file always has a name of the form FPREFIX.MOU, where FPREFIX is a valid file prefix (1 to 20 characters). The command script must have a name of the form FPREFIX.NIN where FPREFIX is the same prefix as that of the mesh file. **Nelson** issues an error message if both input files are not available in the working directory. To organize data, the resulting output files have names that begin with FPREFIX.

A **Nelson** run consists of several steps that involve three different programs: mesh generation, field solution, and analysis. The programs communicate through data files. Sometimes, you may calculate several solutions from the same mesh by changing the characteristics of materials. Each run includes the following steps.

■ Prepare a **Mesh** input command file with a name of the form FPREFIX.MIN following the instructions in the **Mesh** manual. You can create the file directly with a text editor or graphically using the Drawing Editor of **Mesh**.

■ Run **Mesh** either interactively from the **TC** program launcher or from the Windows Command Prompt to create the file FPREFIX.MOU.

■ Prepare a command script (FPREFIX.NIN) using the *Create command file* menu entry in the solution program or your own text editor. The allowed file commands are described in this manual and the manual for the specific solution program.

■ Run **Nelson** to create the output file FPREFIX.NOU. It is in text format and contains information on the mesh geometry, the physical properties of regions, and values of computed node quantities. In magnetodynamic solutions the region properties are the relative magnetic permeability, conductivity and complex current. The computed quantity is the complex vector potential A_z for planar problems and rA_θ for cylindrical problems.

10

■ Analyze the solution using **NView**. You can also transfer the information in `FPREFIX.NOU` to your own analysis programs.

4. Structure of the command file

The input script for all **TriComp** solution programs is a text file (ASCII format) with data lines containing commands and parameters. The file must end with the `EndFile` command. The entries on a line can be separated by the following delimiters

```
Space, Comma, Tab, Colin, Equal sign,
Left parenthesis, Right parenthsis
```

Any number of delimiters can be used in a line. Blank lines and comment lines are ignored. Comment lines begin with an asterisk (*). Most parameters are real numbers. The following formats are valid.

```
1.000
5.67E6
6.8845E+09
5
```

The bottom number is interpreted as 5.0.

The program will accept commands in any order. The following example illustrates a complete control file for **Nelson**:

```
* MAGHEAD.NIN

Set Freq 2.0E6
Set ResTarget 1.0E-6
Set Omega 1.65
Set DUnit 1000.0
Set MaxCycle 2500
Set Geometry Rect

Region 1 Material 1.0
Region 2 Potential 0.0
Region 3 Current 1.0 0.0
Region 4 Current 1.0 180.0
Region 5 Material 500.0 1.0E-3

EndFile
```

There are two types of commands that begin with different keywords:

```
SET
REGION
```

Commands that begin with the word `SET` control program settings. The keyword that follows determines the function of the command. One or more parameters follow the keyword. For example, the command

```
SET GEOMETRY CYLIN
```

instructs the program to use element weighting for cylindrical coordinates. The available `SET` commands depend on the solution program.

`REGION` commands set physical properties associated with elements and vertices. They all start with the word `REGION` and have same format.

```
Region   RegNo     Keyword     Value
```

Here, `RegNo` (an integer) is the region number defined in the **Mesh** input file. The string `Keyword` specifies the physical property. The keyword is followed by one or more numbers. As an example, the command

```
REGION 2 VecPot 0.0
```

in a magnetostatic solution sets nodes with region number 2 as fixed vector-potential points with the value $A_z = 0.0$ or $rA_\theta = 0.0$. In some programs, several `REGION` commands may refer to the same mesh volume and set different properties.

5. Commands for general program control

Nelson commands divide into two groups: those that control the program operation and those that set the physical properties of regions. We begin with the program commands. These commands begin with the keyword **SET**. Each command is written as it might appear in the script file (`FName.NIN`).

SET GEOMETRY CYLIN

Nelson finds solutions for planar systems (variations in x and y with infinite extent in z) or cylindrical geometries (variations in r and z with azimuthal symmetry). The command options are *Rect* and *Cylin*. For cylindrical problems, the program takes the z-axis along the x direction of the mesh and the r-axis along y. In this case, the program issues an error message if y is less than 0.0. The **NView** analysis program automatically adjusts labels to represent the two geometry options.

SET FREQ 3.4E5

Sets the RF or AC frequency of the magnetic field and currents in the solution. Enter the value in Hz. The default is 60.0.

SET DUNIT 1.0E4

You can use any convenient distance units for the **Mesh** input file. The quantity *DUnit* tells how to convert coordinates from **Mesh** to the standard distance units of meters in **Nelson**. The quantity *DUnit* is the number of **Mesh** units per m. For example, if the **Mesh** dimensions are entered in microns, set $DUnit = 1.0 \times 10^6$. The default value is 1.00.

SET OMEGA 1.85

Controls the *over-relaxation factor* for the solution, a number in the range 1.0 to 2.0. The default value is 1.75. Generally, high values give faster convergence. Reduce the over-relaxation factor if the solution does not converge. In comparison to **BStat**, the the complex-number solutions of **Nelson** may require lower values.

SET MAXCYCLE 200

The maximum number of relaxation cycles. The default value is 2500.

SET RESTARG 5.0E-6

Nelson keeps track of the relative error in the magnitude of the complex vector potential during the relaxation solution. The program stops when the error falls below *ResTarg*. Values of *ResTarg* less than about 5.0×10^{-6} give good accuracy. The default value is 5.0×10^{-7}.

6. Commands to set material properties

The **Nelson** command set for defining the physical properties of regions is simple. There are four commands that divide into two classes.

- Boundaries or surfaces with fixed values of vector potential.

- Volume regions with values of magnetic permeability, electrical conductivity and (optionally) applied current density.

The definition of a conductive region requires two parameters

- Relative magnetic permeability μ_r (dimensionless)

- Volume resistivity, ρ (ohm-m)

Initially, all conductive regions are set to $\mu_r = 1$ and $\rho = \infty$.

Commands to set the physical properties of materials all start with the word **Region** and have same format.

    Region   RegNo   Keyword   Value

Here, *RegNo* (an integer) is the Region Number defined in the Mesh input file. *Keyword* is one of the words listed below and *Value* is a real number. The first two commands apply to fixed potential regions.

REGION 4 POTENTIAL 3500.0 45.0

The keyword *Potential* designates that the complex vector potential has a fixed value at all nodes of the region. The first parameter is the amplitude of the harmonically-varying vector potential A_z (in tesla-m) for planar problems and the amplitude of the stream function rA_θ (in tesla-m^2) for cylindrical problems. The second parameter is the phase in degrees. If the phase is not specified the default is 0.0. Differences in vector potential at two points are related to magnetic flux between the points. Setting the vector potential equal to (0.0,0.0) on the full boundary of the solution region is equivalent to the specification that the magnetic flux is contained inside the volume (perfectly-conducting box). Magnetic field lines are parallel to a surface of constant vector potential. Note that external boundary nodes with an unspecified vector potential automatically assume the Neumann condition where the magnetic field is normal to the surface.

REGION 2 MATERIAL 40.0 5.0E-4

> Sets material properties. The two parameters are the relative magnetic permeability $\mu_r = \mu/\mu_o$ and the material volume resistivity ρ in Ω-m. If the second number is not included, the program automatically takes $\rho = \infty$. (Note that 1.0 $\mu\Omega$-cm $= 10^{-8}$ Ω-m.)

REGION 7 CURRENT 30.0E4 30.0

> Set a total current for the region to represent a drive coil. Current flows in the z direction for planar problems and in the θ-direction for cylindrical systems. Both *Material* and *Current* commands for a region can appear in the file. The first parameter is the amplitude of the harmonic current in A and the second parameter is the phase in degrees. If the phase is not specified the default is 0.0.

7. Vector potential and boundary conditions

A boundary condition is the specification of the calculated field quantity on the outer edge of the solution region. To understand boundaries, it is important to realize that **Nelson** determines values for the magnetic vector potential at the node points. The analysis program **NView** then takes spatial derivatives of these values to determine the magnetic fields. The boundary conditions therefore apply to the vector potential.

In rectangular problems (variation in x and y with no variation in z) **Nelson** finds the vector potential A_z created by currents that move in the z direction. The magnetic fields are given by

$$B_y = -\frac{\partial A_z}{\partial x}. \tag{1}$$

and

$$B_x = \frac{\partial A_z}{\partial y}. \tag{2}$$

For problems in cylindrical coordinates, **Nelson** calculates the quantity rA_θ, where A_θ is the vector potential created by azimuthal currents. In this case the magnetic fields are given by

$$B_r = -\frac{\partial A_\theta}{\partial z}, \tag{3}$$

and

$$B_z = \frac{1}{r}\frac{\partial}{\partial r} rA_\theta . \tag{4}$$

The above equations show that changes in values of the vector potential are related to the flux of magnetic field lines (the surface integral of magnetic field). For example, in a rectangular problem with boundaries at y_1 and y_2, the difference in the boundary values of the vector potential at position x_o is given by

$$A_z(x_o,y_2) - A_z(x_o,y_1) = \int_{y_1}^{y_2} B_x(y',x_o)\, dy'. \tag{5}$$

A similar equation holds for integrals in the x direction. Therefore, if we set A_z equal to a constant value (usually zero) around the complete solution boundary, all magnetic flux must be trapped inside the region because the integral of B_x or B_y over any plane is zero. This is equivalent to generating the magnetic field inside a perfectly conducting metal box. Another way to see this is to note that because $\mathbf{B} = \nabla \times \mathbf{A}$, the magnetic field must be parallel to a surface of constant $\mathbf{A}$.

In cylindrical geometry, the symmetry of magnetic field lines dictates that the axis ($r = 0$) must always be a line with $\psi = rA_\theta = 0.0$. **Nelson** automatically sets the condition for cylindrical problems that include $r = 0$ by adding a constant potential region on the axis. The value of the stream function on the outer boundary at the outer solution radius is related to the flux of axial field by

$$(rA_\theta)]_{r_o} = \left(\frac{1}{2\pi}\right)\int_0^{r_o} 2\pi r'dr' B_z(r,z) = \frac{Flux}{2\pi} . \tag{6}$$

16
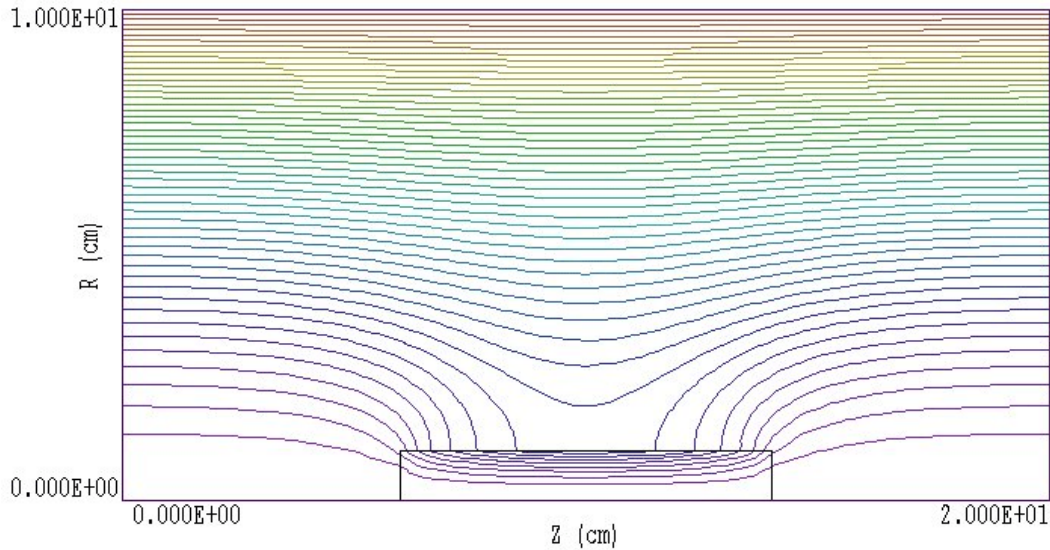
Figure 7. Setting boundary values of the vector potential to create a region of uniform flux. Cylindrical geometry. Vertical: r from 0.0 to 0.1 m. Horizontal: z from 0.0 to 0.2 m. Lower boundary: $rA_\theta = 0.0$ tesla-m^2. Upper boundary: : $rA_\theta = 5.0 \times 10^{-4}$ tesla-m^2.

As an example, the figure above shows the magnetic field lines around an iron cylinder immersed in a uniform solenoidal field of 0.1 tesla. The uniform field was created by setting $rA_\theta = 5.0 \times 10^{-4}$ tesla-m^2 on the outer boundary at $r_o = 0.10$ m. With this background, we can now discuss the implications of the two types of boundary conditions used in **Nelson**.

Dirichlet

Dirichlet boundary points have a fixed value of vector potential that does not change as the **Nelson** relaxation proceeds. A line region of uniform vector potential represents a perfectly-conducting metal surface with magnetic field lines parallel to the boundary. An internal volume region at fixed vector potential represents a metal body that excludes the magnetic field.

Neumann

A Neumann boundary is one where the normal derivative of the vector potential is specified. The boundaries in **Nelson** are limited to the case where $\partial \mathbf{A}/\partial \mathbf{n} = 0$. The special Neumann condition implies that the magnetic field is normal to the boundary. One of the advantages of the finite-element method is that all boundaries that are not fixed automatically satisfy the special Neumann condition, even if they are slanted or curved. Neumann boundaries are often to reduce computation time by representing only half of a symmetric system.

17
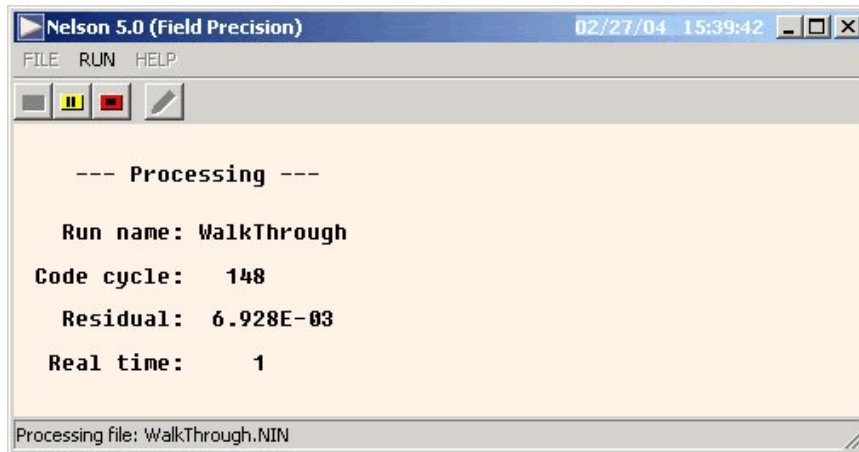
Figure 8. **Nelson** running in a window.

8. Running Nelson interactively

The program Nelson.EXE can run interactively in a Window (Figure 9). In this mode you can carry out several solutions in a session or leave the program while you work in other Windows applications.

The most convenient way to run solution programs and post-processors is from the **TC** program launcher. Operation and setup of the launcher is described in the *TC Manual*. You can also run the executable programs individually from Windows Explorer or add shortcuts to the desktop or the Windows Start Menu.

The program has three popup menus: *File, Run* and *Help*. The following commands appear in the *File* menu.

Create input file

This command calls up a full-featured internal Windows editor to create an input script. Supply a run prefix (1-20 characters). The allowed commands for the specific solution program appear as a set of comment lines. Save or abandon the file to return to the solution program.

Edit input file
Edit listing file
Edit file

The commands call the editor to inspect or to modify ASCII input and output files for **Nelson**. Choosing a file from an alternate

directory does not change the working directory. The *Edit input file* command shows a list of all files with names of the form `FPREFIX.NIN`. The *Edit listing file* command displays files with names `FPREFIX.NLS`.

The *Run* menu has four commands.

Start run

Pick an input file (such as `FPREFIX.NIN`) to start a solution. The working directory is changed if you pick a file from an alternate directory. The run begins if the file `FPREFIX.NOU` and any data files specified in the command script are present. The small text window displays information on the run status.

Pause run

The intensive calculations of the solution program demand the full resources of your computer, causing other tasks to run slowly. If you need to perform critical work, you can pause the solution program and reactivate it later without loss of data.

Stop run

This command terminates the program and saves the current state of the relaxation solution. For example, you may want to stop a relaxation solution at a moderate value of convergence to check whether the problem has been correctly defined.

9. Running Nelson from the command prompt

To run **Nelson** in the background, go to the Command Prompt from Windows and log to the data directory that contains the required `MOU` and `NIN` files.. For example, suppose the input data files `SOLENOID.MOU` and `SOLENOID.NIN` are stored in `\TRICOMP\BUFFER` and that the program `NELSON.EXE` is in the directory `\TRICOMP`. From `\TRICOMP\BUFFER` type

```
..\Nelson SOLENOID <Enter>
```

The program runs silently, writing detailed information in the listing file
SOLENOID.NLS. If the solution is successful, the program writes the
output file SOLENOID.NOU to the data directory. During lengthy runs
you can perform other tasks in Windows. Note that the response of the
computer may be slowed considerably.

The main function of the command mode is autonomous operation under
batch file control. This feature is useful if you want to make an extended
series of solutions overnight or in the background. As an example,
suppose you have prepared the input files
SWT01.MIN,...,SWT08.MIN and SWT01.NIN,...,SWT08.NIN
in the directory \TRICOMP\BUFFER. Furthermore, you have written the
following batch file SWRUN.BAT in the data directory using an ASCII
text editor.

```
@ECHO OFF
ECHO Main switch data run
START ..\MESH.EXE SWT01
START ..\NELSON.EXE SWT01
START ..\MESH.EXE SWT02
START ..\NELSON.EXE SWT02
 ...
START ..\MESH.EXE SWT08
START ..\NELSON.EXE SWT08
```

Type

```
SWRUN <Enter>
```

to generate all solutions without the need for further user input.

Microsoft has released over thirty versions of it's 32-bit operating system
since Windows 95. There is considerable inconsistency in DOS emulation
between versions. To ensure consistent batch file operation we supply the
utility **GCon** with all our software. The program emulates many DOS
commands and has advanced features to organize and to analyze large data
runs. To avoid programs, we advise running batch scripts from **GCon**
rather than from the Command Prompt.

12. NView file menu

NView (for View Nelson) is the interactive graphical analysis program for output files from **Nelson**. The program has special features for harmonic magnetic field analyses. Complex numbers are used to represent harmonic quantities in **Nelson**. The amplitude and phase of a complex number C are given by

$$Amplitude = C\,C^*,$$

$$Phase = \tan^{-1}\,[\mathrm{Im}(C)/\mathrm{Re}(C)],$$

where C^* represents the complex conjugate. The are four ways quantities may be represented in **NView**.

- A snapshot of the time dependent fields at a reference phase ϕ. For example, if the magnetic fields in a region vary as $\cos(\omega t + 45°)$, then a setting of $\phi = 45°$ would show the fields at their peak value. You can set value of ϕ with the *Reference phase* command.

- The amplitude of quantities, such as |**B**| or |**E**|.

- Time-averages of quantities, such as the resistive power density.

- The real and imaginary parts of calculated or derived quantities.

.

Program units
Spatial dimensions: m
Vector potential: tesla-m
Phase: degrees
Electric field: V/m
Current: A
Current density: A/m^2
Volume resistivity: Ω-m
Power dissipation: W/m^3

NView has the following popup menus: *File*, *Spatial plots*, *Analysis*, *Scan plots* and *Help*. When the program starts only the *File* and *Help* menus are active. You must load a data file in order to create plots or to perform analyses. This section reviews options in the *File* menu.

Load solution file

The *Load solution file* command displays a dialog with a list of solution files (suffix `NOU`). Changing the directory in the dialog changes the program working directory. Pick an available file and click *OK*. The program loads the solution and updates the status bar. If data retrieval is successful, the *Spatial plots* and *Analysis* menus become active.

Open data listing file

Several of the analysis commands like *Point calculation* and *Line scan* generate quantitative information. You can automatically record the data generated during an analysis session by opening a data file. Supply a file prefix in the dialog or accept the default. The data file has a name of the form `FPREFIX.DAT` and will be stored in the working directory The file is in text format. You can use a text editor to view the file or to extract information to send to mathematical analysis programs or spreadsheets.

Close data listing file

Use this command if you want to start a new file to record data. You must close the file before attempting to view or edit it using the built-in editor of the post-processor.

Run script

Sometimes you may want to perform complex or repetitive analyses on a set of similar solutions. Script file operation is a powerful feature of the **TriComp** postprocessing programs. This command displays a dialog with a list of analysis files that you have created with the suffix `SCR`. Pick a file and click *OK*. The script file can load data files, open and close history files, and perform any of the quantitative analysis functions described in this manual. The script command language is described in Section 16. Note that the script file should be in the same directory as the data files.

Create script

This command allows you to create script files using the internal editor of the post-processor. A dialog box requests a file prefix. The resulting script file will be saved as `FPREFIX.SCR`. Next, the program opens the file in the editor and writes a list of allowed

commands for reference. This list follows the *EndFile* command and will be ignored. Enter analysis commands above the *EndFile* command. After saving the file, you can run it using the *Run script* command.

Edit script

Use this command to change an existing script file. The dialog lists files in the current directory with the subscript SCR. Changing directories does not change the working directory of the program.

Edit data file

Use this command to view or to modify files with names of the form FPREFIX.DAT.

Edit file

Use this command to view or to modify any text file.

13. NView spatial plot menu

Spatial plots show variations of quantities over the two-dimensional space of the simulation. **NView** makes a default plot when a data file is loaded.

Plot type

NView can create a variety of colored screen and hardcopy plots.

Mesh. Outline of the elements in the computational mesh.

Region. Computational mesh with elements color-coded by region number.

Contour. Lines that follow constant values of a calculated quantity. In two-dimensional magnetostatic solutions constant values of A_z or rA_θ are parallel to lines of **B**. Contours are separated by equal increments of magnetic flux. Therefore, in cylindrical coordinates the line spacing increases near the axis even if magnitude of **B** is uniform. The electric field is equal to $E_z = -\omega A_z$ in planar coordinates and $E_\theta = -\omega A_\theta$ in cylindrical coordinates..

Element. Elements of the solution space color-coded according to a computed quantity (such as |**B**|). You can add or remove element boundaries with the *Toggle element outline* command.

Vector. An element plot with orientation lines included in each element to show the local direction of a **B**.

Surface. A three-dimensional plot where a computed quantity is displayed along *z* over a region in the *x-y* or *z-r* plane. The spatial limits of the plot correspond to the current view window for *Mesh, Region, Contour, Element* or *Vector* plots.

A plot type may not support some plotted quantities. For example, a vector plot of magnetic permeability is undefined. If you receive a message when you switch plot types that the current quantity is not allowed, use the *Plotted quantity* command to pick a valid option.

Plotted quantity

A dialog shows a list of available quantities consistent with the current plot type. Note that the list will be empty for *Mesh* and *Region* plots.

NView supports the following plot quantities.

Contour plot quantities
A_z or rA_θ at the reference phase
Magnetic field magnitude, |**B**|
Electric field magnitude, |**E**|

Element plot quantities
Magnetic field magnitude, |**B**|
Electric field magnitude, |**E**|
Induced current-density magnitude, |**j**|
Time-average resistor power dissipation, P

Vector plot quantities
Magnetic field magnitude, |**B**|

Surface plot quantities

A_z or rA_θ at the reference phase
Magnetic field magnitude, $|\mathbf{B}|$
Electric field magnitude, $|\mathbf{E}|$
Induced current-density magnitude, $|\mathbf{j}|$
Time-average resistor power dissipation, P
B_x or B_z at the reference phase
B_y or B_r at the reference phase
E_z or E_θ at the reference phase
Real part of B_x or B_z
Real part of B_y or B_r
Real part of E_z or E_θ
Imaginary part of B_x or B_z
Imaginary part of B_y or B_r
Imaginary of E_z or E_θ

Line scan quantities

A_z or rA_θ at the reference phase
Magnetic field magnitude, $|\mathbf{B}|$
Electric field magnitude, $|\mathbf{E}|$
Induced current-density magnitude, $|\mathbf{j}|$
Time-average resistor power dissipation, P
B_x or B_z at the reference phase
B_y or B_r at the reference phase
E_z or E_θ at the reference phase
Real part of B_x or B_z
Real part of B_y or B_r
Real part of E_z or E_θ
Imaginary part of B_x or B_z
Imaginary part of B_y or B_r
Imaginary of E_z or E_θ

Plot limits

In the default *autoscale* mode the program picks limits in *Contour, Element, Vector* and *Surface* plots that span the full range of the current quantity. With this command you can set manual limits. In the dialog uncheck the *Autoscale* box and fill in minimum and maximum values. Note that the program does not check that the values you supply are physically reasonable. This operation will not affect scaling of other plot quantities. Check the box to return to *autoscale* mode.

Toggle grid

A set of dashed grid lines in *x-y* or *z-r* can be superimposed on *Mesh, Region, Contour, Element* and *Vector* plots. The program automatically chooses intervals and positions so that lines occur at convenient values of *x* or *y* (for example, 0.01 rather than 0.01153). Grids corresponding to the axes ($x = 0.0$ or $y = 0.0$) are plotted as solid lines.

Set contour plot style

This command is active only when the current plot type is *Contour*. There are four choices: monochrome, monochrome with labels, colored and colored with labels. In the colored mode, the lines are color-coded according to the value of the plotted quantity. A legend is included in the information window to the right of the plot. In the labeled modes, contour lines are numbered according to their values. Overlapping labels on closely-spaced lines may look better in a zoomed view.

Toggle element outline

This command determines whether the element boundaries are included in *Element* and *Vector* plots. It may be necessary to deactivate outlines for large meshes.

Toggle mouse/keyboard

By default the program uses interactive mouse entry of coordinates for commands like *Line scan* and *Zoom*. This command switches between mouse and keyboard input. Enter keyboard coordinates in the distance units used in **Mesh**. In other words, if the solution program has $DUnit = 1.0 \times 10^6$, then enter dimensions in microns.

Toggle snap mode

The mouse snap mode is a convenient feature of **NView**. When snap mode is active, the mouse returns the coordinate values closest to an integer multiple of the quantity *DSnap*. In other words, if *DSnap* = 0.5 and the mouse position is [5.4331,-2.6253], the returned coordinates are [5.5,-2.5]. By default, snap mode is *ON*. Snap mode is automatically turned off for coordinate input to the commands *Point calculation* and *Element properties*.

26

Otherwise, the program would pick a location closest to the snap point rather than the point at the tip of the mouse arrow, giving misleading information.

Set DSnap

Set the distance scale for the mouse snap mode.

The following commands change the view limits in *Mesh*, *Region*, *Contour*, *Element* and *Vector* plots. The current view limits of the two-dimensional plots are used when creating three-dimensional *Surface* plots.

Zoom window

You can zoom in on any area of the plot by specifying the two corners of a view box with the mouse. The coordinates are displayed in the window below the plot. The returned coordinate values depend on whether snap mode is active. You can also enter view coordinates from the keyboard by issuing the *Toggle mouse/keyboard* command.

Zoom in

Magnifies the view about the center of the current plot.

Expand view

Expands the view area about the center of the current plot.

Global view

Returns the plot boundaries to the full solution area.

Pan

Moves the center of the plot. Enter two points to define a displacement vector.

The following commands control the appearance of *Surface* plots. The commands are active only when a *Surface* plot is displayed. You may notice that there is a delay if you choose the *Surface* option in the *Plot type* command or if you change the plotted quantity when a *Surface* plot is

active. To create the plot, **NView** must map the current quantity to a rectangular grid, a process that requires a large number of interpolations.

Rotate 3D image

Rotate the *Surface* plot 90° in the spatial plane.

View angle 3D

Change the angle of view for the *Surface* plot.

Set grid 3D

To create the plot a quantity is mapped to a rectangular grid with dimensions $NX \times NY$. These numbers also determine the total number of grid lines in the *Surface* plot. The default values are $NX = NY = 40$. You can change values with this command. Although higher numbers give plots with more detail, the regeneration time is longer and the screen display may be unattractive. The command causes a program delay because values must be recalculated.

The following commands control plot output to printers and plot files.

Default printer

Spatial and scan plots can be ported to any installed Windows printer (including network printers, postscript drivers,...). You can generate colored plots if you have a color printer. When you issue the *Default printer* command, the current screen plot is sent to the default Windows printer. If necessary, change the default using the *Settings* command of Windows.

Plot file (PostScript)

Use this command to make a plot file of the current screen plot in Encapsulated PostScript format. Supply a file prefix in the dialog box. The plot file will be created in the current directory with the name `FPREFIX.EPS`.

Plot file (BMP)

Use this command to make a plot file of the current screen plot in Windows Bitmap format. Supply a file prefix in the dialog box.

The plot file will be created in the current directory with the name `FPREFIX.BMP`.

Plot file (PNG)

Use this command to make a plot file of the current screen plot in Portable Network Graphics format (GIF equivalent). The plot file will be created in the current directory with the name `FPREFIX.BMP`.

Copy to clipboard

Copy the current plot to the clipboard in Windows MetaFile format. The plot can then be pasted into a compatible graphics program.

11. NView analysis menu

The commands in the Analysis menu generate numerical data. Most of the functions require coordinate input from the user, usually through the mouse. Therefore, the *Analysis menu* is active only when a *Mesh*, *Region*, *Contour*, *Element* or *Vector* plots is displayed.

Set reference phase

Set the reference phase for associated plotted and calculated quantities. Enter the value in degrees. The default reference phase is 0.0°.

Point calculation

NView employs a sophisticated interpolation technique that preserves discontinuities at material boundaries. The program gives correct values for magnetic flux density on both sides of a material boundary. Click on the command and then point to any position in a *Mesh, Region, Contour, Element* or *Vector* plot. Note that snap mode is turned *OFF* for coordinate input. The program writes a subset of interpolated quantities to the window below the plot and also records complete information if a data file is open. To enter point coordinates by keyboard, use the *Toggle mouse/keyboard* command.

Toggle interpolation

The default interpolation method for the *Point calculation* and *Line scan* commands is a second-order least-squares fit with intelligent collection of data points. For example, only points on the side of a dielectric boundary that contains the target point are included to give the correct field discontinuity at the boundary. The least-squares fit may fail in very small regions or enclosed areas if the program cannot identify enough data points. In this case, toggle to the linear mode. Here, field values are determined by a first order fit in the element that contains the target point.

Line scan

The line scan is one of the most useful functions of **NView**. After clicking on the command, supply two points with the mouse in a view of a *Mesh, Region, Contour, Element* or *Vector* plot to define a scan line. The snap mode is useful in this application (for example, you may want a scan to extend from 0.000 to 5.000 rather than 0.067 to 4.985.) The program computes a series of values of field quantities at equal intervals along the line. The information is recorded if a history file is open. The program also makes a screen plot of the currently selected quantity versus distance along the scan and activates the commands in the *Scan plot* menu (see Section 12). The program adds fiducial lines to the plot using intelligent grid selection. This means that the plot is adjusted to fill the screen and grids are drawn at useful intervals (like 0.05 or 2.00).

Scan plot quantity

With this command you can choose the quantity to display in screen and hardcopy plots of line scans. Pick the quantity from the list box and click *OK*. This setting has no effect on the history file listing which includes all field quantities. The available line scan quantities are listed in the previous section.

Set number of scan points

This command sets the number of line scan points used for the screen plot and recorded in the history file. The default value is 50 and the maximum number is 500.

Element properties

After you pick a triangular element with the mouse (or keyboard), **NView** writes material and field properties of the element to the screen. The information is also recorded if a history file is open. It is usually best to start from a *Mesh, Region, Element* or *Vector* plot to identify the element.

Region properties

To see the physical properties associated with a region of the solution space, click the mouse close to any arc or line vector of the region. Partial results are shown on the screen and a complete analysis is included in the data file. **NView** calculates volume integrals of field energy density and power density over the region.

Volume integrals

No input is needed for this command. **NView** calculates the total electromagnetic energy and power dissipation in all regions of the solution volume.

Matrix file

NView can make matrix files of field values to help you create your own analysis routines. Although information is also available in the output file of the solution program, it is difficult to deal with the conformal triangular mesh. The *Matrix file* command uses the interpolation capabilities of the program to create an ASCII data file of field quantities on a rectangular grid in *x-y* or *z-r*. The command displays a dialog box where you set the matrix file prefix, the dimensions of the box and the number of intervals along *x* and *y* (or *z* and *r*). The program creates the file FPREFIX.MTX in the current directory.

12. NView scan plot menu

The commands of the *Scan plot* menu become active when a plot is created following the line scan command. The commands to export plots were described in Section 11. This section describes unique commands of the Scan plot menu.

Oscilloscope mode

In the oscilloscope mode, a scan plot assumes many of the characteristics of a digital oscilloscope. The program superimposes a cross-hair pattern on the graph. Plot values at the intersection are displayed in the information window. Move the marker along the plot by moving the mouse. If you click the left mouse button at a point, the program displays the plot values along with the numerical derivative and integral of the curve. The definite integral is taken from the left-hand side of the plot to the current point. Values are displayed on the screen and written to the data file if open. Press the right mouse button to exit the oscilloscope mode.

Toggle scan symbols

The setting determines whether plot symbols are added to the scan plot showing calculated points.

Toggle grid

The setting determines whether grid lines are added to the screen and hardcopy scan plots.

Close scan plot

The scan plot must be closed before you can use the *File* and *Analysis* functions of **NView**. This command closes the scan plot and returns the program to the previous spatial plot.

13. NView script file commands

Script files to control **NView** have a name of the form FPREFIX.SCR. They should be in the same directory as the data files. Scripts are text files that follow the **TriComp** syntax conventions. The program ignores blank lines and indentations. Data lines use the standard delimiters and comment lines begin with '* ' (asterisk). Processing ends when the *EndFile* command is encountered.

To run a script, click on the *Run script* command in the *File* menu. The program shows a list of available script files. Pick a file and click *OK*. The script operates on the presently loaded data file or you can load other files from within the script. You can also sequentially open one or more data files.

NView can run autonomously under script file control from the Windows Command Prompt. Suppose you have a file GTest.SCR in the directory \TRICOMP\BUFFER and that NVIEW.EXE is in the directory \TRICOMP. From \TRICOMP\BUFFER, type

```
..\NView GTEST <Enter>
```

The main application of the Command Prompt mode is to generate data files and to perform extensive analyses under batch file control.

The following commands can appear in a script:

Input SOLENOID.NOU

Close the current data file and load a file for analysis. The parameter is the full name of the data file. For the command illustrated, the post-processor would load the file SOLENOID.NOU. You can load several files for sequential analysis.

Output SW02

Close the current history file and open an output file SW02.DAT.

Point 5.65 10.68

Perform interpolations at the specified point and write the results to the data file. The two real number parameters are the coordinates of the point (*x-y* or *z-r*) in **Mesh** units.

Scan 0.00 0.00 10.00 0.00

Write the results of a line scan between the specified points to the data file. The four real number parameters are the starting and end coordinates in **Mesh** units: $[x_{start}\ y_{start}\ x_{end}\ y_{end}]$ or $[z_{start}\ r_{start}\ z_{end}\ r_{end}]$.

Interpolation LSQ

Set the interpolation method for subsequent *Point*, *Line scan* and *Matrix* commands. The options are *LSQ* (least-squares fit) and *Linear*.

Element 5.65 10.68
> Write the properties of the element at the specified point to the
> data file. The two real number parameters are the coordinates of
> the point (*x-y* or *z-r*) in **Mesh** units.

NScan   150
> Set the number of intervals in a line scan. The default is 50 and the
> maximum number is 500.

Region 5
> Write volume and surface integrals for a region to the data file.
> The integer parameter is the region number.

VolumeInt
> Write volume integrals for the full solution and regions to the data
> file.

Matrix Switch1 10 20   0.00 0.00 5.00 10.00
> Create a matrix file and record values. This command must have
> the following seven parameters.
> Param 1: The prefix of the matrix file `FPREFIX.MTX` (string).
> Param 2: Number of intervals along the *x* (or *z*) direction (integer).
> Param 3: Number of intervals along the *y* (or *r*) direction (integer).
> Param 4-7: The coordinates of the corners of a box in the solution
> volume, $[x_1,y_1,x_2,y_2]$ or $[z_1,r_1,z_2,r_2]$ (real).

Endfile
> Terminate execution of the script file. You can add descriptive text
> in any format after this command.

The following is an example of a script file to compare field values along
the axes of four different solutions and to write the results in a file
`COMP.DAT`.

```
OPEN OUTPUT COMP
OPEN INPUT SWITCH01.NOU
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH0N.BOU
SCAN 0.00 -50.00 0.00 50.00
```

34

```
OPEN INPUT SWITCH03.NOU
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH04.NOU
SCAN 0.00 -50.00 0.00 50.00
ENDFILE
```

14. Format of the Nelson output file

The **Nelson** output file `FPrefix.NOU` is in text format. It can be inspected with any text editor. The file has three sections:

- Header with general information on the run

- Node and element information

- Region informaton

The header section consists of a title line and 9 data lines:

```
--- Run parameters ---
XMin: -1.270003E-01
XMax:  1.270003E-01
KMax:  101
YMin:  0.000000E+00
YMax:  2.540005E-01
LMax:  101
DUnit:  3.937000E+01
NReg:     5
ICylin:  1
Frequency: 6.000000E+01
```

Lines 2 and 3 list x_{min} and x_{max}, the limits along the horizontal axis (*x* or *z*) of the solution volume. Dimensions are given in meters. The quantity K_{max} in Line 4 is the number of nodes along the horizontal direction. Lines 5-7 describe the vertical axis (*y* or *r*). Line 8 contains the quantity *DUnit*, the conversion factor from dimensions used in **Mesh** to meters. Line 9 gives the number of regions in the solution, while Line 10 specifies the symmetry (0: planar, 1: cylindrical). The final line gives the frequency in Hz.

The node section consists of 4 title lines and $K_{max} \times L_{max}$ data lines, one for each node of the solution space.

```
--- Vertices ---
  k   l  RgNo RgUp RgDn      x             y           A(Real)          A(Imag)
================================================================================
   1   1    8    1     0 -1.066748E+00  0.000000E+00  0.000000E+00  0.000000E+00
   2   1    8    1     0 -1.035344E+00  0.000000E+00  0.000000E+00  0.000000E+00
   3   1    8    1     0 -1.004005E+00  0.000000E+00  0.000000E+00  0.000000E+00
```

```
   4    1    8    1     0 -9.727584E-01   0.000000E+00   0.000000E+00   0.000000E+00
   5    1    8    1     0 -9.416392E-01   0.000000E+00   0.000000E+00   0.000000E+00
```

Each data line contains the following quantities:

- The indices of the node (K, L)

- The region number of the node ($RgNo$) and region numbers for two associated elements ($RgUp$ and $RgDn$). The upper element lies about the line between nodes (K, L) and $(K+1, L)$ and the lower element lies below the line.

- The coordinates of the element in meters (x, y)

- The real and imaginary parts of the vector potential A_z at the node in tesla-m or the stream function rA_θ in tesla-m^2.

The region section consists of four title lines following by *NReg* data lines, one for each region. For a dielectric solution, the region section has the following appearance:

```
--- Regions ---
RegNo Fix                           VecPotAmp      VecPotPhase         Mu

   1    0    0    0    0     0  0.000000E+00   0.000000E+00   1.256637E-06
   2    0    0    0    0     0  0.000000E+00   0.000000E+00   3.769911E-06
   3    0    0    0    0     0  0.000000E+00   0.000000E+00   1.256637E-06
   4    0    0    0    0     0  0.000000E+00   0.000000E+00   1.256637E-06
   5    0    0    0    0     0  0.000000E+00   0.000000E+00   1.256637E-06
   6    0    0    0    0     0  0.000000E+00   0.000000E+00   1.256637E-06
   7    0    0    0    0     0  0.000000E+00   0.000000E+00   1.256637E-06
   8    1    0    0    0     0  0.000000E+00   0.000000E+00   1.256637E-06


              Sigma        CurrentAmp    CurrentPhase

         0.000000E+00   0.000000E+00   0.000000E+00
         8.333333E+05   0.000000E+00   0.000000E+00
         0.000000E+00   5.000000E+02   0.000000E+00
         0.000000E+00   5.000000E+02   0.000000E+00
         0.000000E+00   5.000000E+02   0.000000E+00
         0.000000E+00   5.000000E+02   0.000000E+00
         0.000000E+00   5.000000E+02   0.000000E+00
         0.000000E+00   0.000000E+00   0.000000E+00
```

The table has been folded to fit on the page. A one in the second column designates a fixed-potential region such as a Dirichlet boundary.
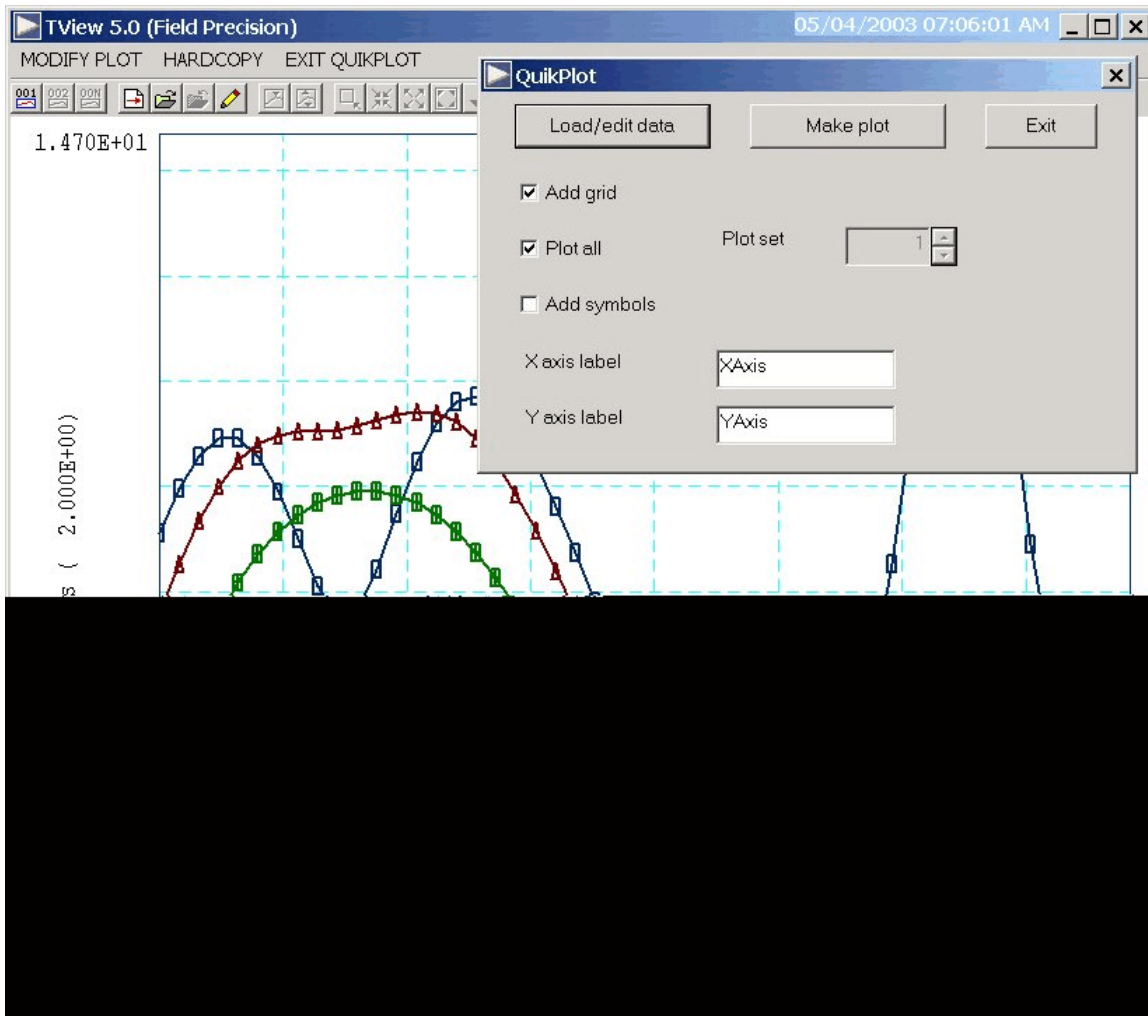
Figure 9. Control dialog superimposed on **QuikPlot** screen.

18. QuikPlot utility

Postprocessors for **TriComp** and **Amaze** programs contain a utility for making fast, simple plots of numerical data. The feature may be useful, for example, to check a table that defines non-linear magnetic permeability. Click on *QUIKPLOT* in the *TOOLS* menu to bring up a dialog (Fig. 10). Initially, all commands are deactivated except *LOAD/EDIT DATA*. Clicking this command brings up an editor window. You can paste numerical data that you have copied from text files or spreadsheets into the window. You can also use the editor function to make changes or to save the data to a file.

The data should consist of columns of numbers separated by any of the standard delimiters (space, tab, comma, equal sign, parentheses). The first

column contains *x*-values. Additional columns contain from 1 to 5 sets of *y*-values. All columns must have the same number of rows. Although there is no limit to the number of rows, the program will pick a maximum of 250 evenly-spaced values to make the plots. Click *SAVE DATA* when you are through.

When you return to the main dialog all commands should be active. Click *MAKE PLOT* to see a plot of the data. The program adjusts the ranges and grid intervals automatically. When the plot is active you can export it in various formats or copy it to the clipboard. Click *MODIFY PLOT* to return to the main dialog to make changes. Here you can change the display of grids and plot symbols or add text labels to the *x* and *y* axes. Note that the *x-y* labels also show the grid intervals. By default, **QuikPlot** shows all data sets (columns of *y*-values) with different line colors and plot symbols.

You can plot a particular set by unclicking the *PLOT ALL* box and specifying the set number (1-5). The plot range is automatically adjusted to the set. In the main dialog, click on *EXIT QUIKPLOT* to return to the normal functions of **NView**.

16 . Finite-element solutions for magnetic fields with eddy current

Static magnetic field solutions are created by applied currents in coils and atomic currents that result from domain alignment. Time-varying magnetic fields can generate a third type of current. The electric field associated with changing magnetic flux drives currents through conductive materials. Eddy currents are of particular concern in magnetic materials with high conductivity like iron. They can significantly reduce the effectiveness of transformers and other devices.

To derive approximate equations we assume that real currents in the solution volume are much larger than displacement currents. The condition is equivalent to

$$f \ll \frac{1}{2\pi\epsilon\rho} \ . \tag{7}$$

In ferrites with high volume resistivity the limit of Eq. (7) holds for frequencies below about 1 MHZ. The approximation holds for most frequencies of interest in iron. Dropping the displacement current, we can write Ampere's law as:

$$\nabla \times \frac{\boldsymbol{B}}{\mu} = \boldsymbol{J}_o + \boldsymbol{J}_c . \tag{8}$$

In Eq. (8) the quantity $\mathbf{J}_o$ is the applied current density in coils and $\mathbf{J}_c$ is conductive current in metals and semiconductors. Expressing the magnetic field in terms of the vector potential, we can rewrite Faraday's law as

$$\nabla \times \boldsymbol{E} = - \frac{\partial}{\partial t} (\nabla \times \boldsymbol{A}) . \tag{9}$$

If there are no free charges to create electrostatic-type fields, then Eq. (9) implies that the total electric field is related to the vector potential by

$$\boldsymbol{E} = - \frac{\partial \boldsymbol{A}}{\partial t} \tag{10}$$

The conductive current is therefore given by

$$\boldsymbol{J}_c = - \sigma \frac{\partial \boldsymbol{A}}{\partial t} \tag{11}$$

Combining Eqs. (8) and (11) gives a differential equation for the vector potential,

$$\nabla \cdot (\frac{1}{\mu} \nabla \boldsymbol{A}) = -\boldsymbol{J}_o - \sigma \frac{\partial \boldsymbol{A}}{\partial t} = -\boldsymbol{J}_o - j\sigma\omega\boldsymbol{A}. \tag{12}$$

The final form applies to a system with steady-state harmonic drive currents at angular frequency ω.

Equation (12) is the same as that for a magnetic field with the exception of the term $j\sigma\omega\mathbf{A}$. We can convert the equation to the finite-element form on a conformal mesh, treating the eddy current density the same way as the source current. For example, consider a planar solution determined by the vector potential A_z. The eddy current contributes a term

$$j \sum_i \frac{\sigma_i \, \omega \, a_i \, A_{zi}}{3} . \tag{13}$$

We must decide how to take the sum in Eq. (13) over the vector potential in surrounding elements. If the local variation of A_z is approximately linear with x and y, contributions to the average from neighboring vertices cancel and it is sufficient to approximate the term as

$$\frac{j \, \omega \, A_{zo}}{3} \sum_i \sigma_i \, a_i . \tag{14}$$

In Eq. (14) the quantity A_{zo} is the value at the test vertex. The finite-element difference equation at a node is

$$\sum_i W_i \, A_{zi} - A_{zo} \sum_i W_i = - \sum_i \frac{J_{zoi} \, a_i}{3} - \frac{j\omega \, A_{zo}}{3} \sum_i \sigma_i \, a_i . \tag{15}$$

Solving for A_{zo} gives a form suitable for a relaxation solution,

$$A_{zo} = \frac{\displaystyle\sum_i W_i \, A_{zi} + \sum_i \frac{J_{zoi} \, a_i}{3}}{\displaystyle\sum_i W_i - j\omega \sum_i \frac{a_i \, \sigma_i}{3}} . \tag{16}$$

The coupling constants W_i are real numbers. The quantities that must be stored as a complex numbers are the vector potential (or stream function) and the sum of applied current densities around a point,

$$\sum_i \frac{J_{zoi} \, a_i}{3} . \tag{17}$$

The complex values of J_{zoi} represent the amplitude and phase of element drive currents. The mechanics of the solution is similar to that for static field solutions. For example, in a relaxation solution we calculate a complex residual

$$R = \frac{\sum_i W_i A_{zi} + \sum_i \dfrac{J_{zoi} \, a_i}{3}}{\sum_i W_i - j\omega \sum_i \dfrac{a_i \, \sigma_i}{3}} - A_{zo} \ . \qquad\qquad (18)$$

for each node and correct values according to

$$A_{zo}' = A_{zo} + \omega R \ . \qquad\qquad (19)$$

The quantity ω in Eq. (19) is the over-relaxation factor. We can find the amplitude and phase of magnetic fields by taking derivative of the complex vector potential. The resistive power dissipation per volume is a useful quantity for applications. The value in an element with nodes n_1, n_2 and n_3 is approximately