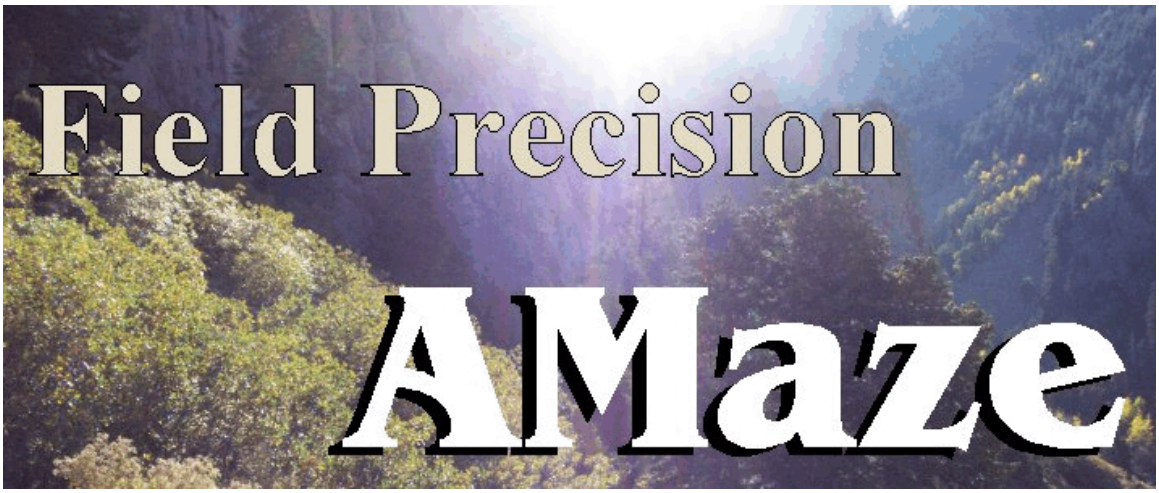

Magnum

Three-dimensional Magnetostatics

Field Precision

Internet: www.fieldp.com
E Mail: techninfo@fieldp.com
PO Box 13595, Albuquerque, New Mexico 87192 U.S.A.
Telephone: 505-220-3975, FAX: 505-294-0222

Magnum
Three-dimensional Magnetostatics

Table of contents

Chapter 1. Introduction to Magnum

1.1 Installation

Magnum applies finite-element techniques to find magnetostatic fields in arbitrary three-dimensional systems. The program can model the effects of applied currents, linear ferromagnetic materials and ideal conducting boundaries for pulsed fields. Utilities are supplied to define drive coils of any geometry. The program can operate in four modes:

- Bounded finite-element solutions, including the effects of ferromagnetic or conducting materials.
- Unbounded free-space fields resulting from a specified distribution of currents.
- Mapping existing 3D or 2D solutions into a single 3D solution.
- Combined bounded and free-space fields.

The fourth option is useful (for example) to determine the total fields created by DC coils and pulsed coils in the presence of conducting boundaries.

Three programs are supplied with the package:

- `CPREP.EXE` – a preprocessor to define applied currents by creating a files of current elements from a script describing coils.
- `MAGNUM.EXE` – the main solution program to determines one or more magnetostatic solutions in a window or under batch file control
- `MAGVIEW.EXE` – a postprocessor to generate plots and to perform analyses of solutions from **Magnum**.

MetaMesh, the **AMaze** conformal mesh generator, is required to create geometry files for solutions.

Magnum features fast and accurate calculations in random-access memory. The programs use dynamic memory allocation; therefore, the size of the solution is limited only by the installed RAM. A computer with

512 MB of memory can handle several million elements. Installation instructions are given in the file `INSTRUCTIONS.TXT` supplied with the program.

The remainder of this chapter describes a step-by-step run that will test your setup and familiarize you with the programs. Chapter 2 reviews the basic physics of **Magnum**. Although a detailed knowledge of numerical theory is not required to apply the program, this brief description of program methods will help you to create effective solutions. Chapter 3 covers the programs used for a complete magnetostatic solution and the organization of input and output files. Defining applied currents is the first step in most **Magnum** simulations. Chapter 4 describes the **CPrep** preprocessor. The program automatically generates large sets of current elements from coil geometry information in a script. Chapter 5 covers the structure of the **Magnum** control script. The script sets parameters for program operation and defines the material properties of regions. Chapter 5 covers program control commands while Chapter 6 describes commands to set material properties. Chapter 7 covers operation of the program interactively in a Windows. Chapter 8 explains how to run complex simulations automatically under batch file control. The chapter also reviews features of the Field Precision utility **GCon** for controlling batch runs. Chapters 9 through13 describe the analysis and plotting capabilities of the **MagView** postprocessor. Finally Chapter 14 summarizes the formats of the current element input file and the **Magnum** output file. This information is useful if you want to write your own analysis programs.

1.2. Test run

A quick way to understand the solution procedure is to step through an example. The calculation describes a finite-length quadrupole magnet. A coarse mesh and symmetry boundaries are used for a fast solution – the process takes less than 1 minute. Figure 1.1 shows a cross-section of the magnet in the *x*-*y* plane. The orientation was chosen so that the planes $x = 0.0$ and $y = 0.0$ are symmetry planes that satisfy the condition $B_{\parallel} = 0.0$. Therefore, it is sufficient to model only one quadrant of the magnet. Similarly, the symmetry condition $B_{\perp} = 0.0$ holds at the axial midplane ($z = 0.0$), so it is sufficient to model only the portion of the magnet in the region $z > 0.0$. The iron pole has an axial length of 12.00 cm. It extends from $z = 0.0$ cm to $z = 6.0$ cm in the half-plane simulation. The outer radius is 5.0 cm. A peripheral region of coarse mesh extending to $x = 8.0$ cm, $y = 8.0$ cm and $z = 16.0$ cm has been included for an accurate representation of the fringing fields. Figure 1.2 shows a three-dimensional view of the pole piece and drive coils.

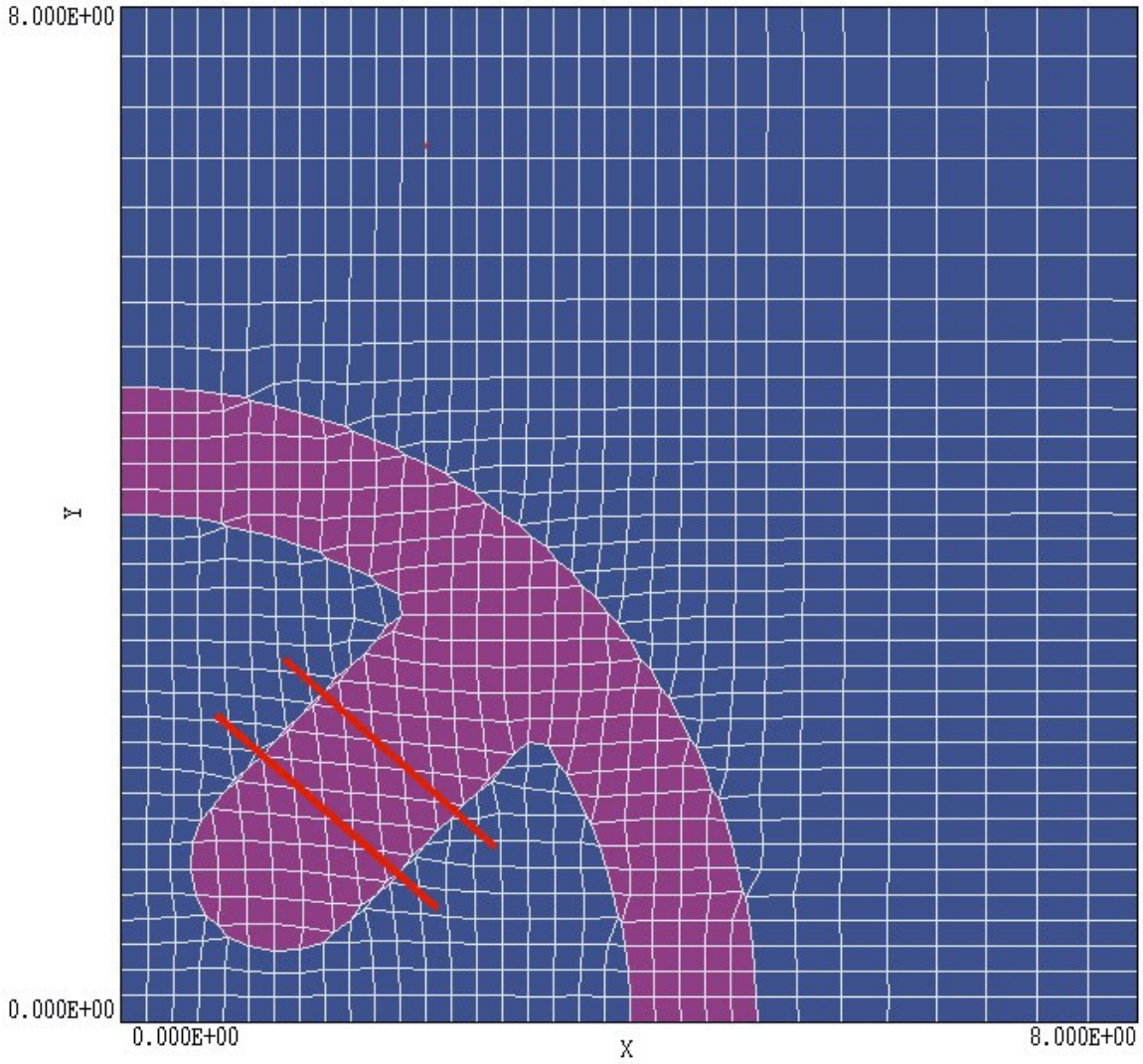

Figure 1.1. Mesh plot of the cross-section of the quadrupole magnet in a plane normal to *z* at *z* = 4.0 cm. A projection of the drive coils is shown in red.

To prepare for the run, move the example input files `SHORTQUAD.MIN`, `SHORTQUAD.CDF`, `SHORTQUAD.GIN` and `SHORTQUAD.SCR` to the working directory (*i.e.*, `\AMAZE\BUFFER`). Run `AMAZE.EXE` (the **AMaze** program launcher) and click on *Set data directory*. Move to `\AMAZE\BUFFER` and click *OK*. Then launch **MetaMesh**. The first step in the solution process is to create a volume mesh. Table 1.1 shows the **MetaMesh** script `SHORTQUAD.MIN`. The mesh serves two purposes. First, the region assignments and shapes of the conformal elements define

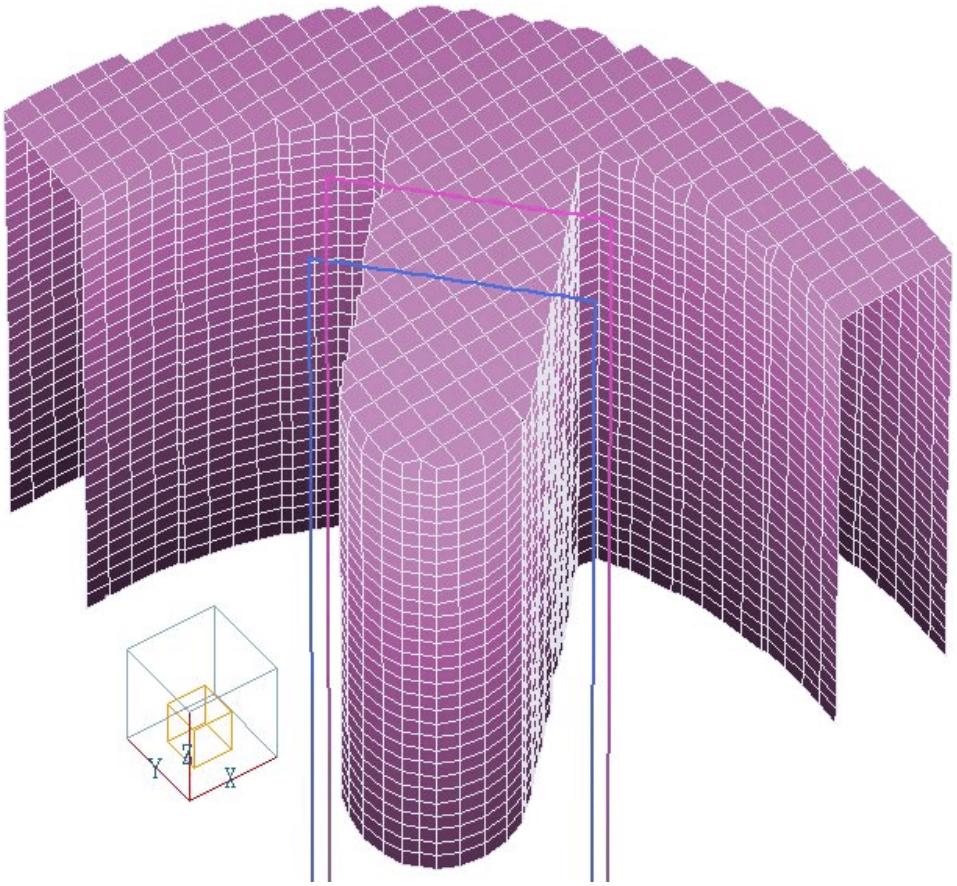

Figure 1.2. Three-dimensional view of the pole piece in the solution volume and a portion of the drive coils.

the material division of the solution volume (*i.e.*, the boundaries of the pole piece in Fig. 1.2). Second, the mesh serves as an armature for projecting the applied fields. The calculated field values at nodes are used to determine source terms for the magnetic field solutions in **Magnum**. Furthermore the **MagView** postprocessor uses the values for interpolations of applied fields at points in the solution volume. Load and process the file `SHORTQUAD.MIN`, and save the mesh to create the file `SHORTQUAD.MDF`.

Table 1.1. MetaMesh script file SHORTQUAD.MIN

```
* File ShortQuad.MIN
Global
  XMesh
     0.00  5.20  0.20
     5.20  8.00  0.40
  End
  YMesh
     0.00  5.20  0.20
     5.20  8.00  0.40
  End
  ZMesh
     0.00  6.20  0.20
     6.20 16.00  0.40
  End

  RegName 1 Air volume
  RegName 2 Quad pole
  RegName 3 Symmetry boundary
End
* Air volume
Part 1
  Type Box
  Region 1
  Fab 16.00  16.00  32.00
End
* Quad pole
Part 2
  Type Extrusion
     L    4.00E+00  0.00E+00  5.00E+00  0.00E+00
     A    5.00E+00  0.00E+00  6.08398E-08  5.00E+00  0.00E+00  0.00E+00 S
     L    0.00E+00  5.00E+00  0.00000E+00  4.00E+00
     A    4.86719E-08  4.0E+00  2.28388E+00  3.28388E+00  0.0E+00  0.0E+00 S
     L    2.28388E+00  3.28388E+00  7.50000E-01  1.75000E+00 S
     A    7.50E-01  1.75E+00  7.50E-01  7.50E-01  1.25E+00  1.25E+00 S
     A    7.50E-01  7.50E-01  1.75E+00  7.50E-01  1.25E+00  1.25E+00 S
     L    1.75E+00  7.50E-01  3.28388E+00  2.28388E+00 S
     A    3.28388E+00  2.28388E+00  4.0E+00  0.0E+00  0.0E+00  0.0E+00 S
  End
  Region 2
  Fab  12.00
  Surface Region 1
End
* Symmetry boundary in X
Part 3
  Type BoundXDn
  Region 3
End
* Symmetry boundary in Y
Part 4
  Type BoundYDn
  Region 3
End
EndFile
```

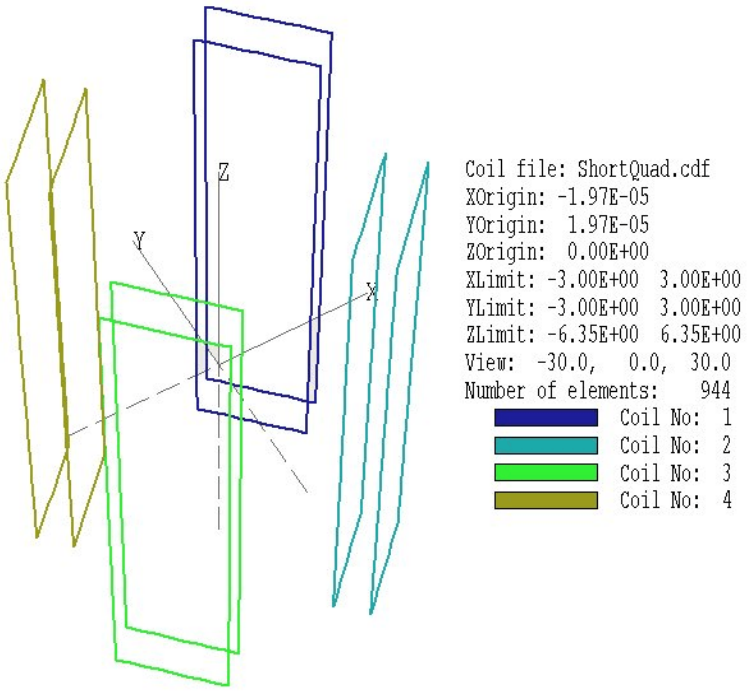

Figure 1.3. Current element set created by **CPrep** for the SHORTQUAD example.

The second required input component is the spatial distribution of applied currents to drive the magnetic fields. The values are contained in a file with a name of the form DataName.FSC, where FSC stands for *free-space coil*. The file contains a set of current element specifications. Each data line gives the start point, end point and current of an element for the applied magnetic intensity $\mathbf{H}_s$ at nodes of the solution volume. The input file SHORTQUAD.FSC contains information on 944 current elements. In most cases, you need not create this file directly. The **Magnum** package contains the utility program **CPrep** which reads coil specifications and automatically divides the coils into small current elements.

Run CPrep.EXE and choose *Open coil file* in the *File* menu. Pick the file SHORTQUAD.CDF (where *CDF* stands for Coil Definition File). **CPrep** immediately processes the file and shows a plot similar to Fig. 1.3. Table 1.2 shows the content of the script. It defines eight box coils that surround the four pole extensions of the quadrupole magnet. You could define a larger number of coils to give a better approximation of the winding density – we used a small number in this example to ensure a fast solution time. Note that we need to define the full complement of four coil sets over the complete axial range even though the solution in **Magnum** is

Table 1.2. CPrep script file SHORTQUAD.CDF

```
* File ShortQuad.CDF
GLOBAL
  DUnit = 100.0
  Ds = 0.250
END
COIL  1  Quadrant 1
  Current = 1000.0
  Rotation 90.0 0.0 -45.0  XYZ
  Shift  1.750 1.750 0.000
  Rectangle  (-1.061, -6.354) (1.061, 6.354)
  Rotation 90.0 0.0 -45.0  XYZ
  Shift  2.250 2.250 0.000
  Rectangle  (-1.061, -6.354) (1.061, 6.354)
END
COIL  2
  Current = -1000.0
  Rotation 90.0 0.0 -135.0  XYZ
  Shift  1.750 -1.750 0.000
  Rectangle  (-1.061, -6.354) (1.061, 6.354)
  Rotation 90.0 0.0 -135.0  XYZ
  Shift  2.250 -2.250 0.000
  Rectangle  (-1.061, -6.354) (1.061, 6.354)
END
COIL  3  Quadrant 3
  Current = 1000.0
  Rotation 90.0 0.0 -225.0  XYZ
  Shift  -1.750 -1.750 0.000
  Rectangle  (-1.061, -6.354) (1.061, 6.354)
  Rotation 90.0 0.0 -225.0  XYZ
  Shift  -2.250 -2.250 0.000
  Rectangle  (-1.061, -6.354) (1.061, 6.354)
END
COIL  4  Quadrant 4
  Current = -1000.0
  Rotation 90.0 0.0 -315.0  XYZ
  Shift  -1.750 1.750 0.000
  Rectangle  (-1.061, -6.354) (1.061, 6.354)
  Rotation 90.0 0.0 -315.0  XYZ
  Shift  -2.250 2.250 0.000
  Rectangle  (-1.061, -6.354) (1.061, 6.354)
END
ENDFILE
```

carried out only in the first quadrant. This is a consequence of the solution technique applied in the program – **Magnum** first calculates applied fields from a Biot-Savart integral and then uses finite-element techniques to compute the field contributions from materials. The **MagView** post-processor displays a superposition of the applied and material field

components. Although symmetry boundaries influence the material fields, we must ensure the symmetry of the applied fields directly by including the full area of all coils. Chapter 2 gives a more detailed description of the solution process.

Next run **Magnum** from **AMaze**. After the program starts, pick *Run* from the main menu or use the run tool. In the dialog, pick the file SHORTQUAD.GIN. Table 1.3 shows the contents of the file. **Magnum** reads the mesh and current elements files, calculates applied fields, determines element matrices and coupling coefficient necessary for the finite-element solution and then proceeds with a dual relaxation solution. The entire process takes less than a minute on a 1.6 GHz computer. The program creates the binary output file SHORTQUAD.GOU and the text listing file SHORTQUAD.GLS. You can use the *Edit listing file* command to inspect SHORTQUAD.GLS.

Table 1.3. Magnum script SHORTQUAD.GIN

```
* File ShortQuad.GIN

SOLTYPE Standard
DUNIT 100.0
SOURCE ShortQuad.FSC
NCHECK = 10
MAXCYCLE = 2000
RESTARGET 5.0E-8

MU(1) = 1.0
MU(2) = 500.0
POTENTIAL(3) = 0.0

ENDFILE
```

Following completion of the solution, start the program **MagView.** Pick *Load solution file* from the *File* menu or click on the tool. Pick the file SHORTQUAD.GOU in the dialog. **MagView** can generate a wide variety of 2D and 3D plots. Figure 1.4 shows an example, a 3D representation of the pole piece with |**B**| plotted in a plane normal to the z axis. Chapters 10, 11and 12 give detailed information on plotting capabilities. Here, we shall create a simple example to test the program. Click on *Plane plots* to bring up the plane plot menu.

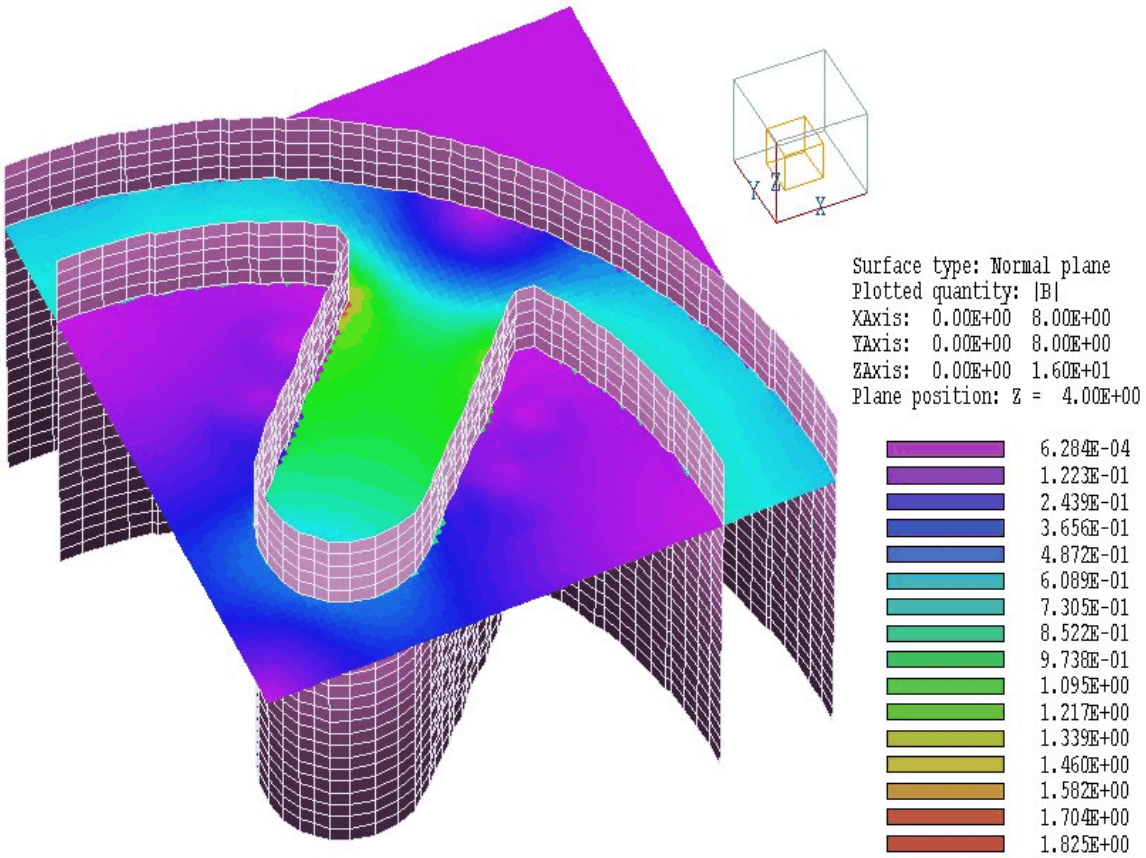

Figure 1.4. Three-dimensional view of physical surfaces with a plot of |**B**| in a plane normal to *z*.

The default plot shows the variation of |**B**| in a plane normal to *z* at the command. In the dialog move the slider to *z* = 4.0 cm, within the region of the pole. Change plot limits to x*max* = 4.0 cm and y*max* = 4.0 cm. Finally, in the *Plot control* menu click on *Plot style*. Increase the plot resolution to 75 in the horizontal direction. You should see a plot similar to Fig. 1.5. If you try some of the other plot styles, note that plane plots may give ragged edges on sloped or curved surfaces. This is because the plot is based on a simple division of the plane into a rectangular array with no special provisions for material boundaries. Slice plots give more accurate representations, although the available styles are more limited.

Although plots are interesting, the primary function of **Magnum** is to generate numbers. Return to the main menu and click on *Run script* in the *File* menu. In the dialog, pick the file SHORTQUAD.SCR. There is a delay while **Magnum** performs an analysis. To begin, we shall inspect the

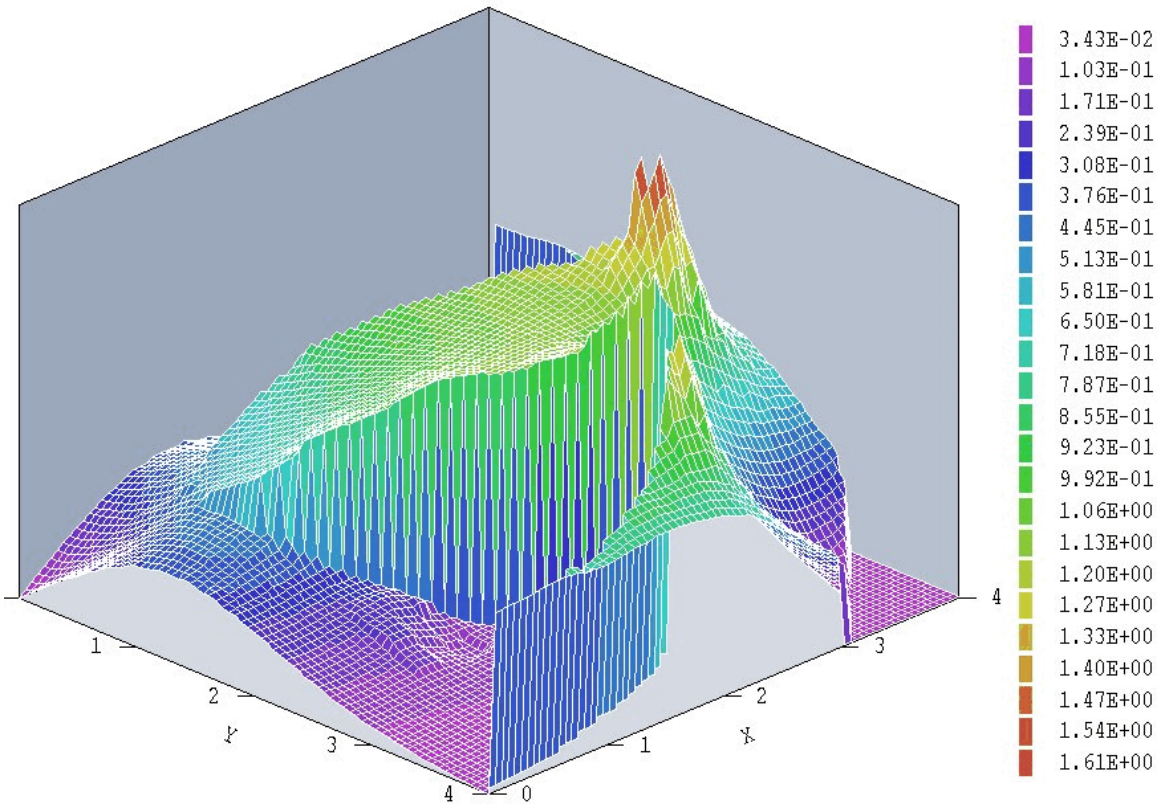

Figure 1.5. Plot of |B| in a plane normal to z at z = 4.0 cm

instructions in the data script. In the *File* menu click on *Edit script* and choose SHORTQUAD.SCR. The internal program editor loads and shows the following content:

```
* File SHORTQUAD.SCR
* Analyze linearity of field variations at
* three axial positions
INPUT ShortQuad.GOU
OUTPUT ShortQuad
NSCAN 50
LINE 0.0 0.0 1.0  0.7  0.7  1.0
LINE 0.0 0.0 5.0  0.7  0.7  5.0
LINE 0.0 0.0 6.5  0.7  0.7  6.5
ENDFILE
```

The first statement ensures that the proper solution file is loaded and the second statement opens a data file to write the results. The command *NScan 50* instructs the program to compute 51 values (50 intervals) along

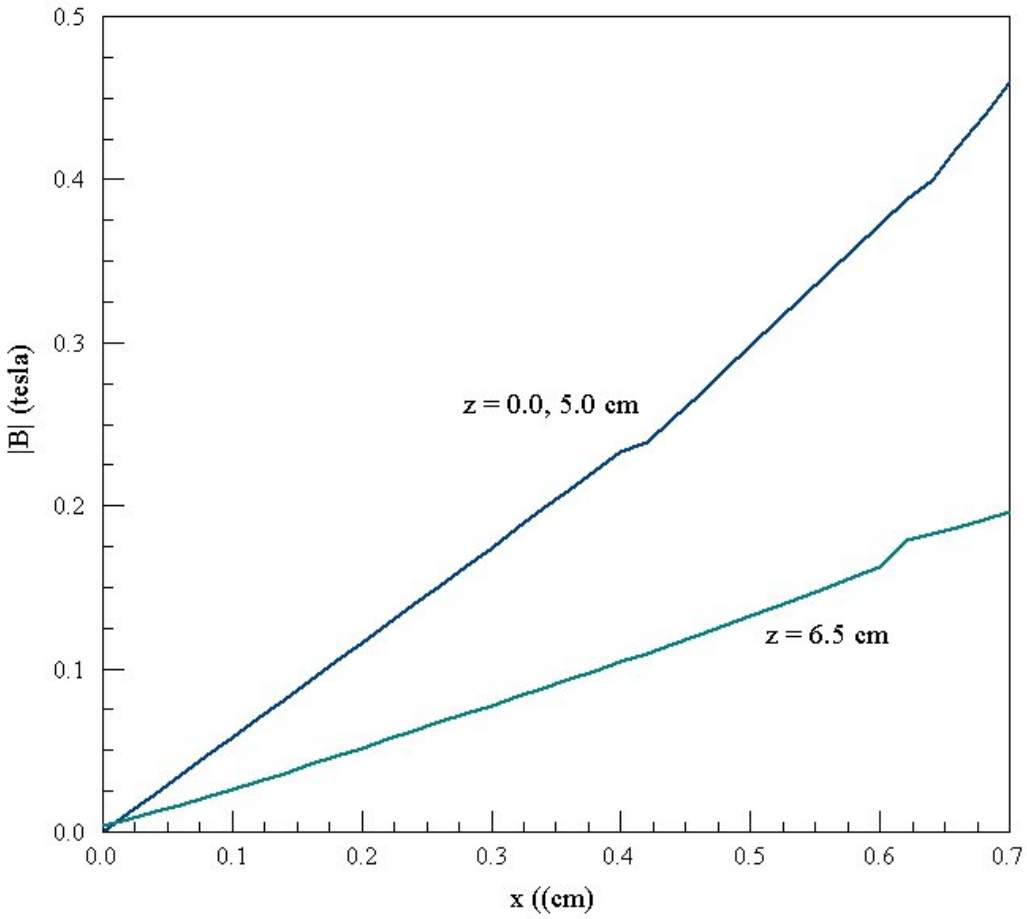

Figure 1.6. Variation of |**B**| as a function of x along a 45° line at three axial positions

a line scan. The next three lines compute scans of values from the axis to in Fig. 1.6. The lines at z = 1.0 and 5.0 cm are almost identical. The field variation near the axis is linear and grows faster than linear near the pole face. The nonlinearity results from the fact that the pole does not have an ideal hyperbolic shape. Note that there are small discontinuities in the plot. We have shown these as a reminder that numerical results are never exact. The interpolation routines have done the best job possible with the coarse mesh (Fig. 1.1). Improved results could be achieved with a finer mesh near the axis.

Chapter 2. Theory of operation

2.1. Reduced potential model

This chapter briefly covers some theoretical background for the numerical methods applied in **Magnum**. A knowledge of the material is essential to create effective solutions. The most familiar method for numerical solution of magnetic fields is to solve the following equation for the vector potential:

$$\nabla \times \left(\frac{1}{\mu} \nabla \times \boldsymbol{A} \right) = \boldsymbol{j}_s \ . \tag{2.1}$$

In Eq. (2.1) the quantity $\mathbf{j}_s$ is the applied current. Currents resulting from the presence of materials are represented by the presence of the factor $1/\mu$. The variation of the quantity μ in space is determined by the geometric distribution of ferromagnetic and conductive materials. Equation (2.1) follows from the definition of vector potential,

$$\boldsymbol{B} = \nabla \times \boldsymbol{A}, \tag{2.2}$$

and Ampere's law,

$$\nabla \times \left(\frac{\boldsymbol{B}}{\mu} \right) = \nabla \times \boldsymbol{H} = \boldsymbol{j}_s \ . \tag{2.3}$$

In Eq. (2.3), the quantity **B** is the *magnetic flux density* and **H** is the *magnetic field intensity*. In this manual, we refer to **B** as the *magnetic field* for brevity.

In two-dimensional solutions, there is only a single component of the vector potential **A** and Eq. (2.1) reduces to the same form as the Poisson equation used in electrostatics. Therefore two-dimensional finite-element magnetostatic solutions are no more difficult than electrostatic solutions. In fact, almost identical routines can be used in programs for the two solution types. Equation (2.1) forms the basis for the Field Precision **TriComp** magnetic codes **BStat**, **PerMag**, **Nelson** and **Pulse**.

The computation is more difficult in three dimensions. The three components of vector potential are coupled through spatial variations in μ. The finite-element equations to represent Eq. (2.1) on a hexahedron mesh

involve 234 coupling coefficients per node as opposed to 26 coefficients in a solution of the Poisson equation. Because of the increased computation times and storage requirements, a direct solution of Eq. (2.1) on a large mesh is impractical on a personal computer. There are two other problems associated with Eq. (2.1):

- It is more difficult to identify boundary conditions on the vector quantity **A**.

- It is challenging to convert current carried by a discrete set of applied current elements to three-dimensional current density $\mathbf{j}_s$ on a conformal mesh of hexahedrons.

Magnum utilizes the indirect *reduced potential* model for three-dimensional magnetostatic solutions. With this method, storage requirements and computation times are only slightly higher than those for an electrostatic solution. The key is to divide the magnetic field intensity into two components:

$$\boldsymbol{H} = \boldsymbol{H}_s + \boldsymbol{H}_m \ . \tag{2.4}$$

The component $\mathbf{H}_s$ arises from the applied currents while $\mathbf{H}_m$ is the component created by currents in materials. By superposition, the components individually satisfy the equations:

$$\nabla \times \boldsymbol{H}_s = j_s \ , \tag{2.5}$$

$$\nabla \times \boldsymbol{H}_m = 0 \ . \tag{2.6}$$

Given the spatial distribution of applied current $\mathbf{j}_s$, the component $\mathbf{H}_s$ can be determined directly from a Biot-Savart integral (Sect. 2.2) at all points in space without recourse to finite-element methods. Equation (2.6) implies that the material component $\mathbf{H}_m$ can be expressed as the gradient of a scalar potential:

$$\boldsymbol{H}_m = -\nabla\phi \ , \tag{2.7}$$

The quantity ϕ is called the *reduced potential*. The equation

$$\nabla \cdot \boldsymbol{B} = 0 \,, \tag{2.8}$$

implies that

$$\nabla \cdot [\, \mu \, (\boldsymbol{H}_s - \nabla\phi) \,] = 0, \tag{2.9}$$

or

$$\nabla \cdot (\mu \, \nabla\phi) = \nabla \cdot (\mu \, \boldsymbol{H}_s \,). \tag{2.10}$$

Equation (2.10) has the form of the Poisson equation with a source term on the right-hand side. The source term can be determined from the known spatial variations of μ and $\mathbf{H}_s$.

The **Magnum** solution procedure consistsn of the following steps:

- Read a file (`MDF`) created by **MetaMesh** that gives node coordinates and the region identities of elements.

- In the **Magnum** script (`GIN`), read values of μ to associate with element region numbers.

- Using geometric and material information, calculate finite-element coefficients to represent the expression $\nabla\cdot(\mu\nabla\,\phi)$ as a set coupled linear equations, one for each node.

- Read a file (`FSC`) of applied current elements created by **CPrep** and apply a Biot-Savart integral to determine values of $\mathbf{H}_s$ at nodes.

- Analyze the mesh to calculate source terms $\nabla\cdot(\mu\mathbf{H}_s)$ at nodes.

- Solve the coupled set of node equations by iterative methods to find the reduced potential ϕ.

- Record information on nodes values of applied field $\mathbf{H}_s$ and reduced potential ϕ in the Magnum output file (`GOU`).

In the **MagView** post-processor, the magnetic flux density is determined by interpolations using the equation:

$$\boldsymbol{B} = \mu\,(\boldsymbol{H}_s - \nabla\phi)\ . \qquad (2.11)$$

Although the sequence of operations is somewhat involved, the reduced potential formulation offers flexibility. As a result, **Magnum** supports several computational modes. We call the full calculation with applied fields and reduced potential the *STANDARD* solution mode. If there are no materials that have $\mu_r \neq 1.0$, we can omit the calculation of ϕ and simply record values of $\mathbf{H}_s$ in the output file. We refer to this as the *FREE* solution mode (for *free*-space fields). In this case, interpolations in **MagView** return values of the applied fields over the solution space. The corresponding field values are those that occur in an infinite space, unaffected by the boundaries of the solution volume.

A third solution mode (*ADDFREE*) is useful in systems where there is a superposition of steady-state fields and pulsed fields in the presence of highly-conductive materials. These materials are modeled by setting a value of permeability $\mu_r \ll 1.0$ in the conductive volume (Sect. 2.3). In this case we prepare two current-element files to represent the two classes of coils (i.e., `DCNAME.FSC` and `PULSENAME.FSC`). **Magnum** first reads `PULSENAME.FSC` and uses the applied fields to generate source terms for the reduced potential calculation. This calculation determines the contributions of the conducting boundaries. Next the program reads `DCNAME.FSC` and superimposes the steady-state components of $\mathbf{H}_s$. **MagView** can display the magnetic flux density arising from applied currents ($\mathbf{B}_s = \mu_o\mathbf{H}_s$) and as well as the total value **B**.

2.2. Applied field calculation

Magnum uses a simple yet versatile method to define three-dimensional coils of any degree of complexity. The program expects to receive a predigested list of short current elements in the `FSC` input file. Chapter 14 describes the file format. Figure 2.1 shows the geometry of a filamentary element of length

$$L = \sqrt{(x_e - x_s)^2 + (y_e - y_s)^2 + (z_e - z_s)^2}\ , \qquad (2.12)$$

at average position

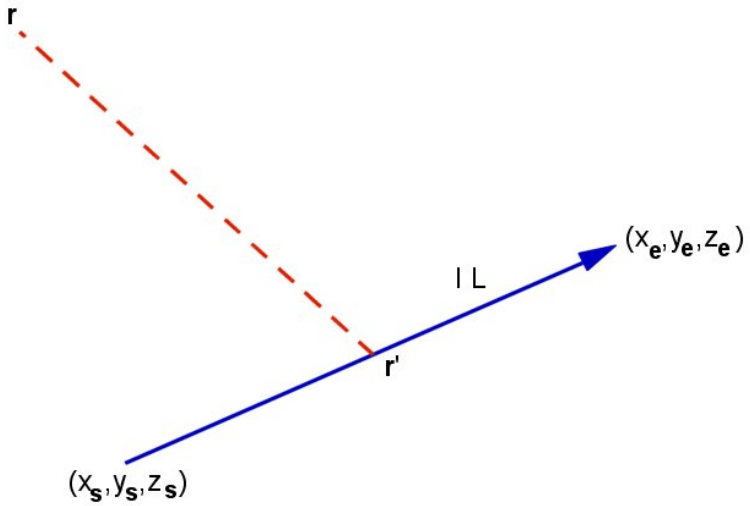

Figure 2.1. Current element geometry.

$$\boldsymbol{r}' = \left[\frac{x_s+x_e}{2} , \frac{y_s+y_e}{2} , \frac{z_s+z_e}{2} \right] \tag{2.13}$$

carrying current *I*. The unit vector

$$\boldsymbol{u} = \left[\frac{x_e-x_s}{L} , \frac{y_e-y_s}{L} , \frac{z_e-z_s}{L} \right] \tag{2.14}$$

lies along the filament direction. The contribution to the magnetic field intensity at position *r* from the element at position *r*' is given by

$$d\boldsymbol{H}_s = \frac{I\,L}{4\pi} \left[\frac{\boldsymbol{u} \times (\boldsymbol{r} - \boldsymbol{r}')}{|\boldsymbol{r} - \boldsymbol{r}'|^3} \right] . \tag{2.15}$$

Magnum computes the total applied magnetic field intensity $\mathbf{H}_s$ at each node position by taking a sum over current elements in the *FSC* file. advantage of the approach is that numerically-derived model particle orbits in the **OmniTrak** code can be treated as a sequence of current

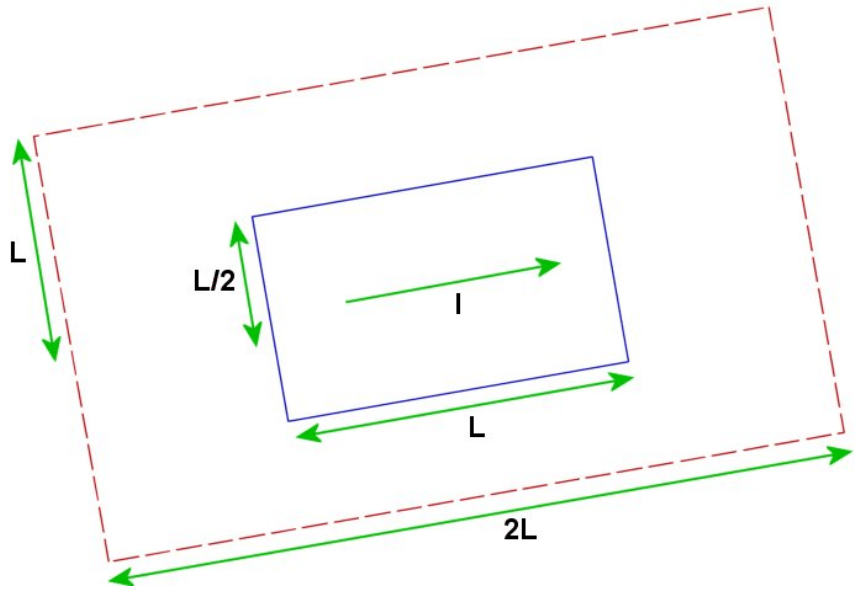

Figure 2.2. Representation of a current element as a cylinder of radius *L*/2. Dashed line indicates the transition boundary to the Biot-Savart calculation.

elements. Hence it is straightforward to include the contribution of beam currents to magnetic fields. The drawback of the method is that it takes time to calculate fields created by thousands of current elements at millions of node points.

The field contribution of a filamentary current-element may be quite large if it lies close to a node. To avoid diverging field values **Magnum** treats an element of length *L* as a uniform-current-density cylinder with diameter *L* (Figure 2.2). The code uses a look-up table derived from a numerical solution within the region shown as a dashed line of Fig. 2.2. To reduce calculation time, the code applies Eq. 2.15 (using the average position of the element) outside the region. The effects of the finite-width of applied currents will be apparent in **MagView** plots. You can adjust the width through the choice of element length in **CPrep** (Chapter 4).

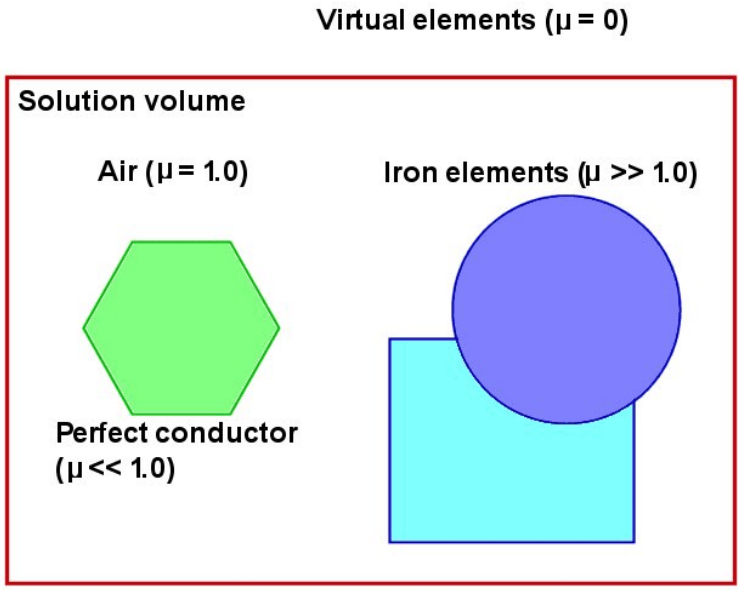

Figure 2.3. Schematic view – division of the solution volume for a dual potential solution.

2.3. Dual potential model

The reduced potential model provides good results in regions of the simulation where the relative magnetic permeability μ_r is not much larger than unity. In iron regions where $\mu_r \gg 1.0$, the two contributions in Eq. (2..11) are almost equal. The small difference is multiplied by μ_r to determine the magnetic flux density **B**. Small interpolation errors in $\mathbf{H}_m$ and $\mathbf{H}_s$ give large errors in **B**. For this reason, **Magnum** generates an alternate finite-element solution for iron regions. This approach is called the *dual potential* model.

Figure 2.3 shows the division of a solution volume into regions representing air ($\mu_r = 1.0$), perfect conductors ($\mu_r \ll 1.0$) and ferromagnetic materials ($\mu_r \gg 1.0$). In a physically-correct solution, applied currents may wrap around iron objects but they do pass directly through them. With the condition $\mathbf{j}_s = 0.0$, the following condition holds on the total magnetic field intensity inside the iron:

$$\nabla \times \boldsymbol{H} = \mathbf{0}. \tag{2.16}$$

Therefore, we can set the total magnetic field intensity **H** equal to the gradient of a scalar potential:

$$\boldsymbol{H} = -\nabla\Psi \tag{2.17}$$

In this case, Eq. (2.8) implies that the potential ψsatisfies the Laplace equation:

$$\nabla \cdot (\mu\nabla\Psi) = 0 \tag{2.18}$$

The approach in the finite-element method is to generate discrete node equations by setting the local integrals of the left-hand side of Eq. (2.18) around nodes equal to zero. This procedure leads to the following equation for ψ at node *i*:

$$\iiint_{\Omega_i} \mu \nabla N_i \cdot \nabla\Psi \, dV = \iint_{S_i} N_i \mu \frac{\partial\Psi}{\partial n} \, dS_i \, . \tag{2.19}$$

In Eq. (2.19) the quantity N_i is a weighting function that we can identify with the shape functions of the hexahedron elements surrounding the node. The volume integral on the left-hand side is identical in form to the integral for the reduced potential ϕ. Therefore, we can use the coupling coefficients that have already been determined for the reduced potential calculation. The new term is surface integral on the right-hand side. The normal derivative of ψ at the surface is equivalent to

$$-\mu \frac{\partial\Psi}{\partial n} = B_n. \tag{2.20}$$

In other words, the node equations include source terms given by surface integrals of the normal component of the magnetic flux density. The form of the node weighting functions N_i and the continuity of magnetic flux guarantees that contributions from all surfaces inside contiguous iron regions equal zero, even if μ varies through the material or if two different regions intersect. Non-zero contributions occur only on the interface between iron and air or conductor regions.

With this background, we can understand the additional steps in the **Magnum** solution procedure when ferromagnetic materials are present.

- After completing the reduced potential calculation, **Magnum** organizes the mesh and marks nodes that are connected to iron

elements. Boundary nodes connected to both iron and air elements are also marked. The goal is to determine values of ψ on this node set. The set may contain multiple connected or disconnected iron regions.

■ Coupling coefficients for the nodes are taken from those created for the reduced-potential calculation. Coupling coefficients to nodes outside the iron region or its boundaries are set to zero.

■ Source terms for the boundary nodes are calculated from values of the normal component of **B** derived from the reduced potential model according to Eqs. (2.19) and (2.20).

■ **Magnum** performs a second iterative solution to find ψ at the mesh nodes and records values in the output `GOU` file.

Field calculations performed in **MagView** depend on the setting of the interpolation mode. In the default *STANDARD* mode, the program applies the equation

$$\boldsymbol{B} = \mu\,(\boldsymbol{H}_s - \nabla\phi)\ . \tag{2.21}$$

at points inside elements with $\mu_r \le 1.0$ and uses

$$\boldsymbol{B} = -\mu\,\nabla\psi, \tag{2.22}$$

in elements with $\mu r > 1.0$.

The other interpolation method uses only the reduced potential [Eq. (2.21)] and makes no distinction between elements according to magnetic permeability:

It is important to note that non-physical results may occur if drive currents pass through iron regions. The codes incorporate two features to avoid this problem.

■ **Magnum** checks the validity of the geometry after opening the mesh (`MDF`) and current element (`FSC`) files. The program checks all current elements and reports an error if the average position of an element lies within an element with $\mu_r > 1.0$.

■ After loading solution (`GOU`) and current element (`FSC`) files, you can create three-dimensional plots in **MagView** to check the relative orientation of coils and iron regions.

Note that the tests do not account for the width of current elements (Section 2.2). You must ensure that no portion of a finite-width element extends into an iron region.

2.4. Material properties

Version 1.0 of **Magnum** is limited to linear, isotropic magnetic materials.(*i.e.*, soft iron and ferrites below saturation). Linear materials are characterized by a constant value of μ that holds at all positions in space. In this case, the magnetic field intensity and flux density vectors are co-linear and have magnitudes related by

$$\boldsymbol{B} = \mu \boldsymbol{H} = \mu_o \mu_r \boldsymbol{H}. \tag{2.23}$$

Magnum 1.0 does not handle self-consistent saturation effects or non-linear permanent magnets.

The physical properties of high-μ materials are familiar from introductory courses on electromagnetism. In the remainder of this section, we shall discuss the properties of low-μ materials ($\mu_r \ll 1.0$) and their application in **Magnum**. The magnetic flux density inside a material with $\mu_r \simeq 0.0$ is **B** = 0.0. Equation (2.8) implies that the normal component B_n across the boundary is continuous. Therefore, the magnetic flux density just outside the material must be parallel to the surface. In other words, a material with $\mu_r \ll 1.0$ acts like a perfect conductor. Figure 2.4*a* shows the effect of such a material on the fields produced by an external pulsed current. You can find the distribution of surface current on the conductor by calculating the magnetic field in the adjacent air region and taking,

$$J_s = \frac{\Delta B_{\parallel}}{\mu_0}, \tag{2.24}$$

where the surface current J_s has units of A/m.

A more interesting situation results when a drive current is inside a region with $\mu_r \ll 1.0$. In this case, the reduced potential solution creates a distribution of material currents that ensure that **B** = 0.0 at all points inside the material. As an example, Figure 2.4*b* shows a cross section of a coaxial transmission line – the inner and outer conductors are represented as materials with $\mu_r \simeq 0.0$. An offset drive current flows in the *z* direction.

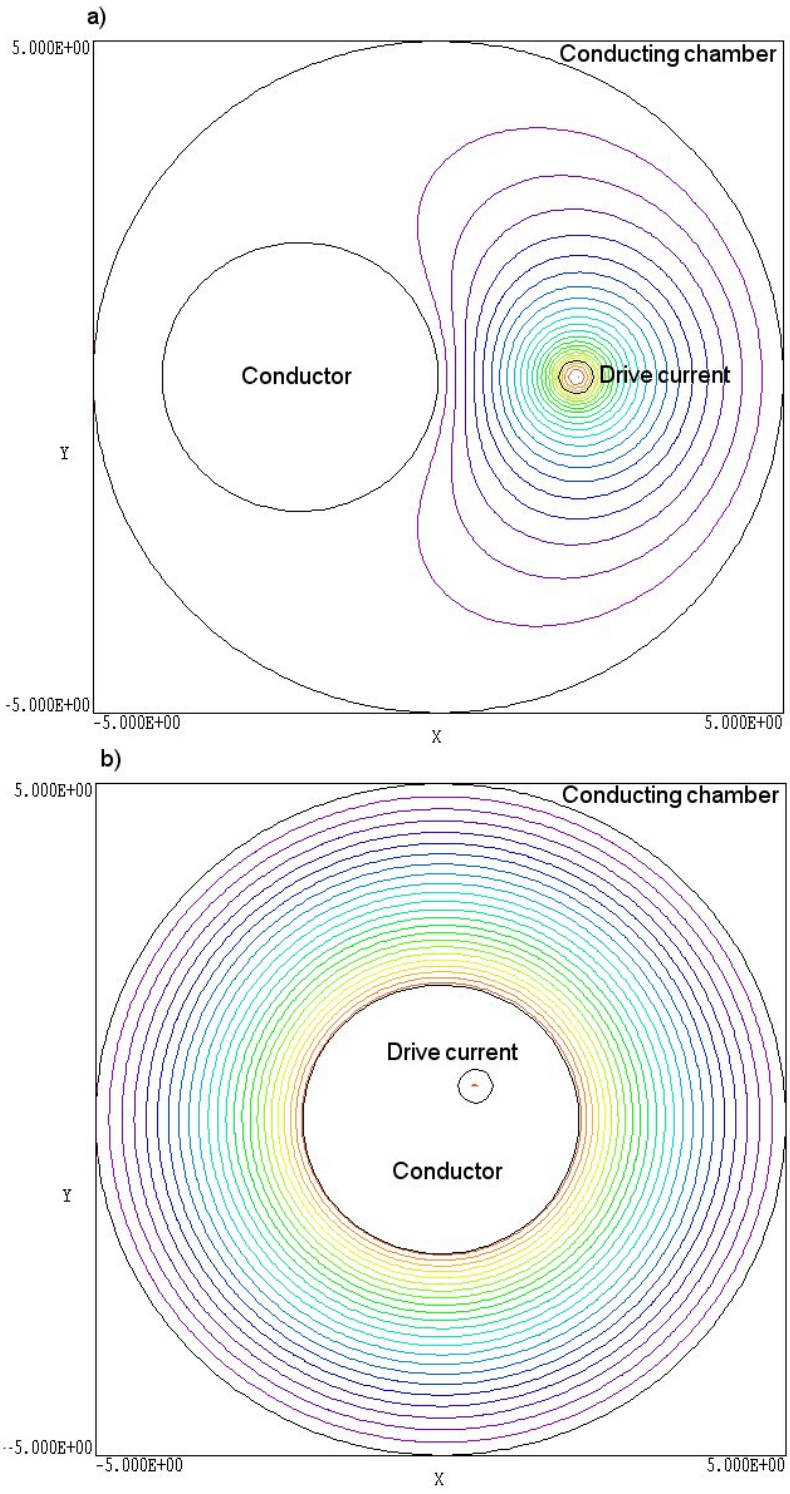

Figure 2.4. Effect of materials with $\mu_r \ll 1.0$ on lines of magnetic flux density **B**. *a*) Drive current external to the region. *b*) Drive current internal to the region.

Even though the drive current is displaced from the axis, the numerically-derived fields are azimuthally symmetric and closely follow the $1/r$ variation of magnitude predicted for the geometry.

An example will clarify methods for modeling perfectly-conducting materials. Figure 2.5 shows the geometry. The input files `SURFACECURRENT.MIN`, `SURFACECURRENT.CDF` and `SURFACECURRENT.GIN` are included in the example library. A toroidal conductor inside a toroidal chamber carries a pulsed azimuthal current of 2000 A – we want to find fields in the intervening space. The problem as shown is two-dimensional to enable comparison of results with **BStat**. The finite-element simulation treats one-quarter of the geometry. The default boundary condition of $B_{\perp} = 0.0$ (Section 2.5) is appropriate for the open ends. It is necessary to use a full circular drive current loop to ensure that the applied fields have correct symmetry at the boundaries. The condition $\mu_r = 10^{-4}$ is assigned to elements inside the brown facets of Fig. 2.5 (inner conductor) and elements outside the blue facets (outer conductor). Elements in the intervening space have $\mu_r = 1.0$. We find that the results are independent of the radius or position of the drive current loop as long as it is inside the inner conductor. Figure 2.6 shows calculated values of $|\mathbf{B}|$ in a normal plane, the summation of applied and material contributions. The field is zero inside the conductors and is azimuthally symmetric in the air gap. Results are in quantitative agreement with two-dimensional calculations performed with **BStat**.

2.5. Boundary and symmetry conditions

The application of boundary conditions in **Magnum** can be challenging because calculations may involve three sources of information: applied fields and material fields (from the gradient of the reduced potential ϕ) in non-iron regions and total fields from the gradient of the dual potential ψ in iron regions.

The field solution for ϕ and ψ with attendant boundary conditions has no effect on the applied field $\mathbf{H}_s$. Therefore, we must ensure the collection of current elements in the `FSC` file generates a physically-correct applied field solution, regardless of the solution-volume limits and symmetries of the finite-element calculations for ϕ and ψ. For example, we must include the full current loop for the calculation of Fig. 2.5. Using only one-quarter

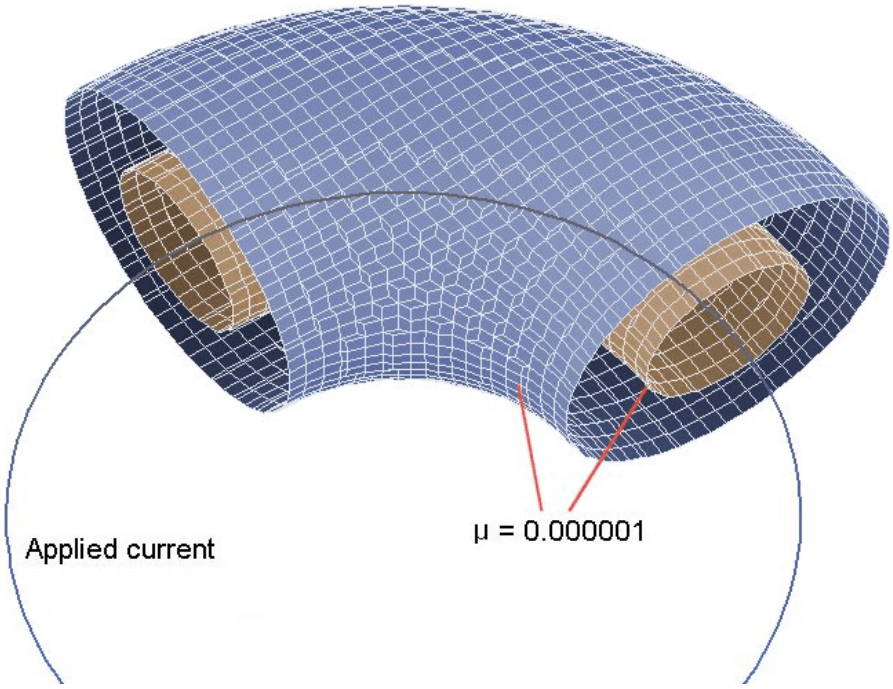

Figure 2.5. Geometry – toroidal conductor with a pulsed azimuthal. current inside a conducting toroidal housing.

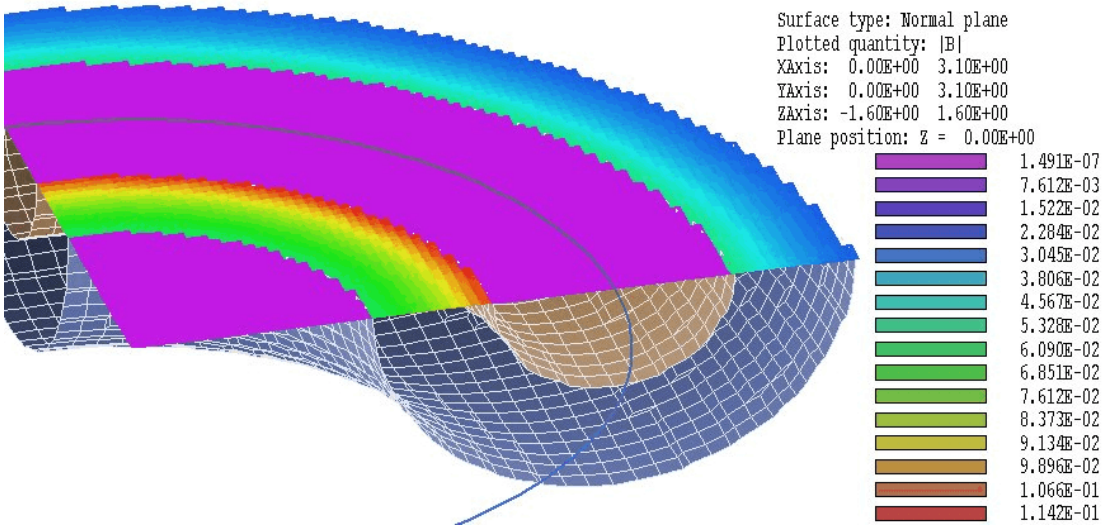

Figure 2.6. Calculated magnetic flux density B for the example of Fig. 2.5.

of a the loop would give a non-physical applied field solution because the drive current would not define a circuit. Sometimes we can omit portions of a coil if the current elements make a negligible contribution to applied fields in the solution volume. For example, consider fields in a cylinder of radius R driven by a line current of infinite length. Because of the scaling of the Biot-Savart equation [Eq. (2.15)], the contribution to the field of elements at an axial distance $z \geq 10R$ is less than 0.1% of the contribution from a nearby element. Therefore, it is sufficient to use a non-circuital line current of approximate length $20R$ to get a good approximation to the applied fields generated by an infinite wire. With careful pre-analysis and judgement, you can substantially reduce the work involved in calculating the applied fields.

We shall next consider conditions for the calculation of the reduced potential ϕ. An underlying assumption in **Magnum** is that the solution volume is surrounded by a set of virtual elements with the property $\mu_r \ll 1.0$. Therefore, the magnetic flux density **B** is constrained to be parallel to any unspecified boundary. Equivalently, the solution volume is automatically surrounded by a virtual volume of perfect conductor. The alternative to the natural boundary condition is to specify a fixed value of reduced potential (typically $\phi = 0.0$) along the surface (Dirichet condition). In this case, the derivative of ϕ parallel to the surface must be zero. This condition implies that the material component of magnetic flux density $\mathbf{B}_m$ is normal to the boundary. If the applied component $\mathbf{B}_s$ is also normal, then the surface constitutes a symmetry boundary. This condition was applied in the example of Section 1.2 to model one-quarter of the quadrupole.

Conditions on the dual potential ψ are the similar to those of ϕ. The region external to an iron region that contacts the solution boundary consists of virtual elements with $\mu_r \ll 1.0$. Therefore, the magnetic flux density **B** in an iron region is parallel to such a boundary. The condition $\psi = 0.0$ implies that the total magnetic flux density **B** is normal to the boundary. The resulting solution is physically-consistent only if the applied field $\mathbf{B}_s$ is also normal to the boundary.

Chapter 3. Organizing Magnum simulations

The **Magnum** package contains three components:

- The program **CPrep** analyzes a script of input specifications for applied-field coils and creates a file of current elements in a standard format.
- The program **Magnum** computes the physical solution
- **MagView** is a dedicated post-processor for analyses of the results.

All three programs can run in two modes: interactively in a window or autonomously in the background under batch file control. The autonomous mode allows automatic processing of large or repetitive data sets.

One input file with a name of the form `FPrefix.CDF` (for *coil definition file*) is required for the **CPrep** preprocessor. Chapter 4 describes the preparation of this file. Three input files may be required for a **Magnum** solution:

- A script that sets control parameters and describes the physical properties associated with regions.
- A list of drive current elements to compute the applied field.
- A **MetaMesh** output file that describes the conformal hexahedron mesh. The file contains node coordinates and the region numbers of elements and nodes.

The **CPrep** output file has a name of the form `FPREFIX.FSC`, where `FPREFIX` is any valid file prefix and `FSC` stands for *free-space coils*. The **MetaMesh** output file always has a name of the form `RUNNAME.MDF`. The **Magnum** script must have a name of the form `RUNNAME.GIN`, where `FPREFIX` is the same prefix as that of the mesh file and `GIN` stands for Ma**G**num **IN**put. **Magnum** issues an error message if any input files are not available in the current working directory. To organize data, the resulting output file name is assigned the name `RUNNAME.GOU`.

For review, a simulation usually consists of the following steps:

- Prepare a **MetaMesh** script (`RUNNAME.MIN`) that defines the solution space. In simulations with conductive or ferromagnetic materials, the conformal mesh describes the division of the solution space into air and material regions. In free-space calculations, the mesh is merely a convenient set of node positions for calculations of applied fields.

- Run **MetaMesh** to create a file `RUNNAME.MOU` of standard mesh information. This file could be used as input for multiple **Magnum** simulations or for other **AMaze** solution programs.

- Prepare a **CPrep** coil definition file (`FPREFIX.CDF`) that defines the geometry and currents of drive coils.

- Run **CPrep** to cut the coils into small current elements and generate a standard element file (`FPREFIX.FSC`).

- Prepare a **Magnum** script (`RUNNAME.GIN`) that sets control parameters and defines the material properties of regions.

- Run **Magnum** to create a solution file `RUNNAME.GOU`. This file contains the following node quantities: spatial coordinates, applied field components ($\mathbf{H}_s$), reduced potential ϕ and dual potential ψ.

- Optionally, prepare a standard **AMaze** analysis script (`ANYNAME.SCR`) to control a **MagView** analysis session.

- Run **MagView** to create plots or to generate numerical data using the information in `RUNNAME.GOU`. The analysis can be performed interactively or automatically under the control of a script.

You may feel that the procedure involves a large number of steps and a considerable amount of preparation. In the long run, the method saves time and aggravation. Splitting complex solutions into several small steps is always a good idea. The scripts form permanent records of the setup and can often be used in different solutions with small modifications. With a good editor, script preparation is usually faster than repetitive menu operations. (Note that the full-featured Programmer's File Editor is included on the **Magnum** distribution disk).

Table 3.1 summarizes the input and output files used in **Magnum**.

Table 3.1. Magnum files

Name form	Function	Status
`RUNNAME.MIN`	Description of simulation geometry, input to **MetaMesh**	*Required*
`FPREFIX.CDF`	Description of applied field coils, input to **CPrep**	*Required*
`FPREFIX.FSC`	Output from **CPrep**, input to Magnum	*Required*
`RUNNAME.MDF`	Output from **MetaMesh**, input to **Magnum**	*Required*
`RUNNAME.GIN`	Description of material properties, input to **Magnum**	*Required*
`RUNNAME.GOU`	Output from **Magnum**, input to **MagView**	*Required*
`ANYPREFIX.SCR`	Analysis control, input to **MagView**	*Optional*
`ANYPREFIX.DAT`	Analysis data output from **MagView**	*Optional*

Chapter 4. Defining applied current with CPrep

4.1. Program function and basic commands

CPrep is a preprocessor that divides coil assemblies into small current elements and records the results in a standard output file. **Magnum** uses this information to calculate applied magnetic fields. Input data is supplied to **CPrep** in the form of a script. The script has a name of the form `FName.CDF`, where the suffix stands for *Coil Definition File*. **CPrep** creates an output file `FName.FSC` that contains a list of current elements. Both files are in text format. You can inspect them with an editor or export information to your own analysis programs

To begin a **Magnum** simulation, you must create an input script that describes applied currents and coil geometries for the system you want to study. The script is a list of commands (similar to a BASIC program) that you write with a text editor. This chapter describes the syntax and available commands in the **CPrep** script.

In this section we shall consider some general features of the coil definition file. The script consists of a set of commands processed in sequence. The file follows the standard format used in all Field Precision programs. Blank lines are ignored. You can include any number of comment lines that begin with an asterisk (*). **CPrep** ignores all information after the *ENDFILE* command, so you can add annotations in any format. Commands are analyzed with a free-form parser. Sets of characters are grouped into words separated by the following delimiters: Space [ ], Comma [,], Tab, Colon [:], Equal sign [=], Left parenthesis [(] and Right Parenthesis [)]. Delimiters can be used in any combination, so you have considerable latitude to choose the appearance of your script. For example, the following commands have the same meaning:

```
Torus 2.0 0.5 10.0 0.0 350.0
Torus = (2.0, 0.5) (10.0, 0.0, 350)
Torus: 2.0, 0.5, 10.0, 0.0, 350
```

The coil definition file consists of a *GLOBAL* section and up to 127 *COIL* sections. Table 4.1 shows the general layout.

Table 4.1. Structure of the CDF file

```
GLOBAL
   (Global commands)
END

COIL
  (Coil 1 commands)
END

COIL
  (Coil 2 commands)
END

...

ENDFILE
```

It is important to understand clearly the definition of the term *coil* as used in **CPrep**. A coil is a set of filamentary wire sections that carry the same current. A coil may be constructed from several *parts*. The complexity of parts varies from a simple line segment to the toroidal helix of Fig. 4.1.

The following chapter describes *COIL* commands to set the current and geometry of coils. In this chapter we shall review the global commands. The *GLOBAL* section must appear at the beginning of the coil definition file It can be followed by several *COIL* sections. The following three commands may appear in any order between the commands *GLOBAL* and *END*. As with all commands in this manual, they are shown in symbolic form and in the form that the might appear in the program.

DUNIT DUnit
DUnit = 1.0E6

CPrep works internally in SI units (lengths in meters). It is often convenient to enter dimensions in alternate units. The quantity *DUnit* is a factor to convert input dimensions to meters. It equals the number of units per meter. For example, to enter dimensions in cm set *DUnit* = 100.0. To enter dimensions in inches, set *DUnit* = 39.37. **CPrep** scans the script file for the *DUnit* command before reading any other commands. The default value is *DUnit* = 1.0.

DS Ds
DS = 0.1

The quantity *Ds* is the approximate length and diameter (Section 2.2) of current elements in units set by *DUnit*. For example if *DUnit* = 39.37, the value *Ds* = 0.25 implies that the element length is about 0.25". Smaller values of *Ds* give more accuracy but result in longer run times. It is good practice to set *Ds* explicitly, but if you don't **CPrep** tries to pick reasonable values for different types of parts. When the *Ds* command appears in the *GLOBAL* section, it sets a default value applied to all coils. When a *Ds* command appears in a *COIL* section, the new value over-rides the default for all elements of that coil.

4.2. Defining coil geometries

A coil is a set of wires carrying the same current. **Magnum** will calculate the magnetic field corresponding to any set of wires, connected or unconnected. It is the user's responsibility to ensure that the coil definitions represent a physically-realizable system. You can define up to 127 different coils, each composed of several parts. Parts may be simple line segments or complex shapes like helices. Up to 2000 parts can be included in a run.

COIL sections follow the *GLOBAL* section. Each coil section begins with the command

```
COIL
```

and ends with the command

```
END
```

CPrep numbers coils in the order they appear in the file and ignores any information after the *COIL* command. You can therefore add descriptive titles to help organize your scripts. For example, the following command is valid:

```
COIL Solenoid 1 (R = 7.8, I = 125 A, N = 50)
```

The commands that appear between *COIL* and *END* set the current and define the shape of the coil. This chapter describes the functions of the coil commands.

CURRENT Current
CURRENT -125.0

A *CURRENT* command must appear in each *COIL* section. The parameter *Current* is the amplitude of the coil current in amperes.

DS DsCoil
DS 0.100

The *Ds* command sets the approximate length of current elements for the present coil. It is useful, for example, if you have a large assembly that contains small coils. If the *Ds* command does not appear in the *COIL* section, the default from the *GLOBAL* section is used.

We next turn to the important topic of coil geometry. The procedures used in **CPrep** are easy to understand if you envision them in terms of physical operations involved in constructing a coil. Fabrication of an assembly proceeds in three stages:

- Build the coil from parts of specified dimensions on a workbench. We shall call the workbench the *assembly coodinate system*.

- If necessary, rotate the parts so that they have the correct orientation relative to the *physical coordinate system*.

- Move parts from the workbench to their final positions in the physical coordinate system.

We use items from a standard parts bin and set the desired dimensions. Simple shapes like line segments may connect any two points in the assembly coordinate system. Complex parts like helices have specific orientations with respect to the workbench. To begin, we shall discuss commands that control orientation and position of the parts.

ROTATION ThetaX ThetaY ThetaZ [RotOrder]
ROTATION 0.0 90.0 45.0 ZXY

This command specifies rotation angles about the Cartesian axes of the assembly coordinate system. Rotations can be performed about the *x*, *y* or *z* axes. The parameters *ThetaX*, *ThetaY* and *ThetaZ* are the respective angles in degrees. The optional string parameter *RotOrder* controls the order in which rotations are performed. The order is important because rotations are not commutative For the rotation in

the example, the 45° rotation is performed about the new *z* direction after the 90° rotation in *x*. *RotOrder* is a string of length 3 that consists of a combination of the characters *X*, *Y* or *Z* (i.e., *YZX*,...). Any number of *ROTATION* commands may appear in a coil section. The rotation is applied to all parts that follow in the coil section until another *ROTATION* command appears.

SHIFT XShift YShift ZShift
SHIFT -10.0 0.0 5.65

This command controls translations from the assembly to the physical system. The three parameters *XShift*, *YShift* and *ZShift* are the components of the displacement vector. Enter the displacements in the units set by *DUnit*. For example, if *DUnit* = 100.0, specify values in cm. Any number of *SHIFT* commands may appear in a coil section. The rotation is applied to parts that follow in the coil section until another *SHIFT* command appears. Translations are applied *after* rotations.

There are twelve parts in the standard bin with which you can construct most practical magnet coils. Additional specialized models are listed in the help file `MAGNUM.HTML`. Lengths should entered in units specified by *DUnit* and angles in degrees.

LINE Xs Ys Zs Xe Ye Ze
LINE 0.0 0.0 0.0 3.56 3.56 0.0

The line segment is the simplest and most versatile part. It is a vector that connects two points in the assembly coordinate system, from (*Xs*,*Ys*,*Zs*) to (*Xe*,*Ye*,*Ze*). A positive value of current implies flow from the *Start* point to the *End*. **CPrep** attempts to cut lines into equal segments with length less than or equal to *Ds.* For short lines, the minimum number of segments is 2. If no *Ds* command appears in the *GLOBAL* section, **CPrep** cuts lines into 10 elements by default.

RECTANGLE Xs Ys Xe Ye
RECTANGLE -5.25 -5.25 5.25 5.25

A *Rectangle* consists of four line segments that define a rectangular coil. A rectangle always lies in the *x*-*y* plane of the assembly coordinate system at *z* = 0.0. Use the *ROTATION* commands to adjust the orientation if necessary. The parameters (*Xs*,*Ys*) and (*Xe*,*Ye*) define the corners of the box in the *x*-*y* plane. A positive value of current

flows in the sense of positive rotation. (If you point the thumb of your right hand along *z*, positive current flows in the direction of your fingers.) As with lines, the minimum number of segments per side is 2 and the default with no *Ds* specification is 10.

CIRCLE R
CIRCLE = 6.0

This command defines a circular coil of radius *R* in a plane normal to the *z*-axis of the assembly frame at z = 0.0. The circle is centered at position (0.0, 0.0) in the *x*-*y* plane. Positive current flows in the direction of positive rotation. The minimum number of elements in a circle is 12. The default value with no *Ds* specification is 16.

ELLIPSE Rx Ry
ELLIPSE = (6.0, 4.0)

This command creates an elliptical coil defined by the equation

$$\left(\frac{x}{R_x}\right)^2 + \left(\frac{y}{R_y}\right)^2 = 1.$$

The ellipse is centered at position (0.0, 0.0) in the *x*-*y* plane of the assembly system and at position *z* = 0.0.Positive current flows in the direction of positive rotation. The minimum number of elements in a circle is 12. The default value with no *Ds* specification is 16.

ARC R ThetaS ThetaE
ARC = 4.0 (0.0, 90.0)

This command defines an arc of radius *R* in the *x*-*y* plane. The start angle (relative to the *x*-axis) is *ThetaS* and the end angle is *ThetaE*. The center point of the arc is at (0.0, 0.0) in the x-y plane at z = 0.0. Positive current flows in the direction of positive rotation..

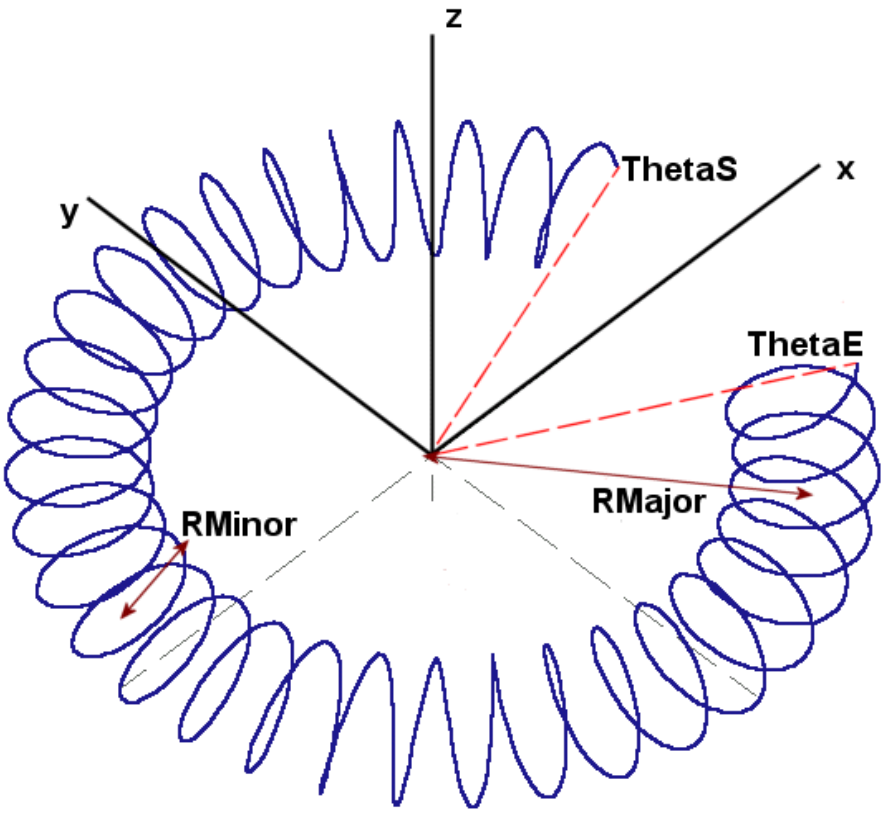

Figure 4.1. Parameters used in the *Torus* model

HELIX R Zs Ze Pitch
HELIX = 2.5 (-15.0, 15.0, 0.500)

The *HELIX* model can be used to create solenoids or twisted wire pairs. The helix is centered at (0.0,0.0) in the *x-y* plane and has a circular cross-section of radius *R*. It extends along *z* from *Zs* to *Ze*. The quantity *Pitch* is the distance along *z* for a full revolution. Therefore, the number of turns is *NTurn* = ABS(*Ze-Zs*)/*Pitch*. By convention the helical coil starts at position (R,0.0) in the *x-y* plane at *z* = *Zs*. The end position in the *x-y* plane is determined by *NTurn*. Use the *ROT ATION* command to adjust the start position. The minimum number of elements per turn is 12 and the default value with no *Ds* specification is 16. By default, the helix has a positive sense of rotation as it moves from *Zs* to *Ze*.

TORUS RMajor RMinor PitchAngle ThetaS ThetaE [ThetaInit]
TORUS = 5.0, 1.0, 12.0, 0.0. 300.0

The toroidal winding creates an azimuthal field. The torus is centered at position (0.0, 0.0, 0.0) with major radius *RMajor* in the *x*-*y* plane. Figure 4.1 defines the parameters *RMinor* and *PitchAngle*. The *PitchAngle* (in degrees) is the rotation about the *z*-axis per turn. The *Torus* starts at angle *ThetaS* (in the *x*-*y* plane relative to the *x*-axis) and ends at *ThetaE*. Current flows in the direction of positive rotation. The minimum number of elements per turn is 12 and the default value is 16. By default, the winding starts at an angle of 0.0° relative to the plane of the minor radius. You can change this value with the optional parameter *ThetaInit*.

The next group of models is useful for constructing volumetric coils with non-zero cross-section area. The models automatically create parallel sets of elements with transverse spacing on the order of *Ds* or *DsCoi*l. Note that volumetric models may generate large numbers of elements. You can ensure that solutions are efficient by using volumetric assemblies only where they are necessary. For instance, suppose you want to find fields near the surface of a circular coil with a rectangular cross section. It would be unnecessary to model the entire circular coil using the *ELBOWR* command if you only needed to know fields at a representative position. Instead, define a short azimuthal section of the coil with the *ELBOWR* model and create the remainder with the *ARC* model. Because of the $1/R^2$ variation in the Biot-Savart integral, the contribution from a distant coil segment has almost no dependance on the coil cross section. All volumetric assemblies have a fixed orientation and position in the assembly coordinate system.

SOLENOID RMin RMax Length
SOLENOID = 5.0 7.5 20.0

The *SOLENOID* model generates elements to represent a solenoid of finite radial thickness. The parameter *RMin* is the inner radius, *RMax* is the outer radius and *Length* is the length. Enter dimensions in meters or units set by *DUni*t. The solenoid consists of a set of circular wires of different radii lying in the *x*-*y* plane and arrayed along z. The wires are centered at (0.0,0.0) in the *x*-*y* plane and extend from -*Leng*th/2 to *Leng*th/2 in the z direction. Figure 4.2 shows the distribution of wires over the cross section. The number of azimuthal sections in each wire is the closest integer to *Circu*m/*DsCoi*l, where *Circum* is average circumference of coils in the set. If *Ds* or *DsCoil* are not specified, the default number of azimuthal sections is 16.

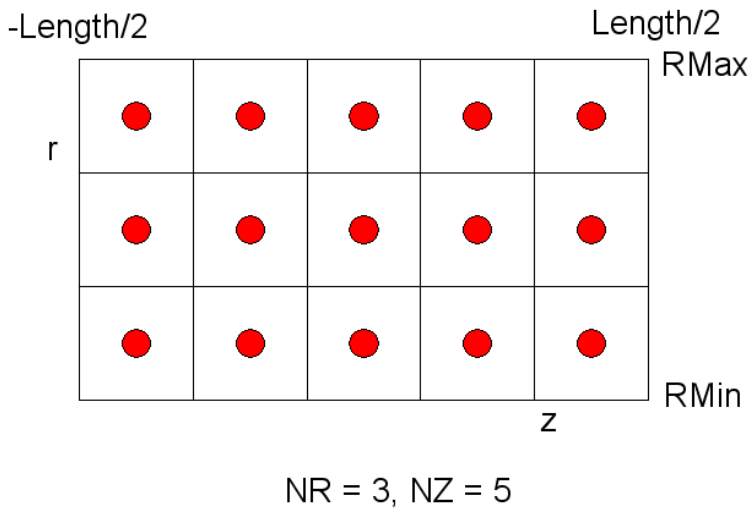

Figure 4.2. Division of the solenoid into wires.

BAR Lx Ly Lz
BAR 0.25 0.25 5.0

A bar is a set of straight wires directed along the *z* axis of the assembly coordinate system that fills a rectangular cross section in the *x*-*y* plane. The parameters are the lengths in x, *y* and *z* in meters or units set by *DUni*t. The wires extend from -*L*z/2 to *L*z/2 in z. The number of elements Nz along *z* equals the nearest integer to *L*z/*DsCoi*l, or *Nz* = 10 if *Ds* or *DsCoil* have not been specified. The array in the *x*-*y* plane is similar to Fig. 3.2 with *Nx* = *L*x/*DsCoil* and *Ny* = *L*y/*DsCoil* (default values: *Nx* = 10 and *Ny* = 10). The quantity *Current* for the coil is interpreted as the total current of the bar. Therefore, each wire carries current *Current*/(*N*x***N*y).

ELBOWR RMin RMax Length Angle
ELBOWR 5.0 6.0 1.0 90.0

The *ELBOWR* model is an elbow with a rectangular cross section. The elbow is equivalent to a *SOLENOID* that extends over a specified angular range rather than 360˚. The parameters *RMi*n, *RMax* and *Length* are identical to those used for the *SOLENOID*. Enter the quantity *Angle* in degrees. The *ELBOWR* extends from the *x* axis of the assembly coordinate system (0.0˚) to *Angl*e.

ROD Radius Length
ROD 0.5 10.0

A *ROD* is similar to a *BAR* but it has an approximately circular cross section. In this model **CPrep** sets up a set of parallel line currents in a hexagonal pattern in the *x*-*y* plane. The hexagons have scale size on the order of *DsCoi*l.

ELBOWC Radius Length Angle
ELBOWC 0.5 10.0 45.0

An *ELBOWC* is an elbow with a circular cross section. The quantities *Radius* and *Length* are the same as those used in the *ROD* model. You can combine *ELBOWC* and *ROD* models to create a continuous set of parallel wires. The *ELBOWC* extends from the *x* axis of the assembly coordinate system to *Angle* (see *ELBOW*R).

The following example (HELMHOLTZ.CDF) demonstrates the use of the *ROTATION* and *SHIFT* commands. We shall set up a Helmholtz coil pair to generate an approximately uniform field in the *x*-direction. To meet the Helmholtz condition, the coaxial coils should be separated by a distance equal to their radius. The two coils have radius R = 10 cm and current I = 400 A and lie in the *y*-*z* plane of the physical coordinate system at positions x = -2.5 cm and x = 2.5 cm. The following *COIL* commands set up the geometry shown in Fig.4.3:

```
COIL (x  =  -2.5 cm)
  Current = 400.0
  Shift = (-2.50,0.00,0.00)
  Rotation = 0.0 -90.0 0.0
  Circle 5.0
END
COIL (x = +2.5 cm)
  Current = 400.0
  Shift = (2.50,0.00,0.00)
  Rotation = 0.0 -90.0 0.0
  Circle 5.0
END
```

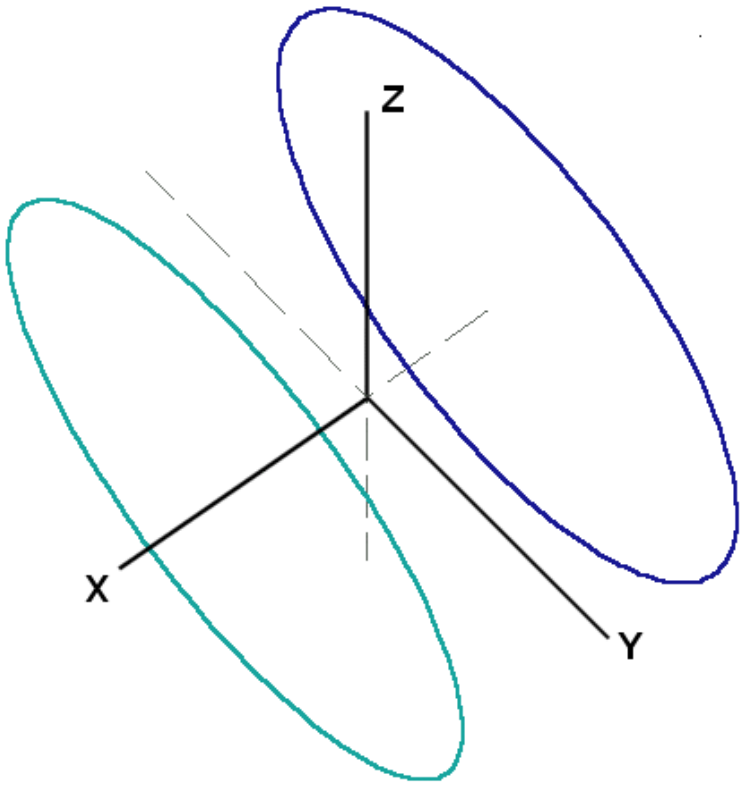

Figure 4.3. Helmholtz coil pair to generate a field B_x

The field at the center of a Helmholtz pair is

$$B_x = \frac{\mu_o I}{1.25^{3/2} R} .$$

For the values used, the predicted field is $B_x = 7.1934 \times 10^{-3}$ tesla. For comparison, the field value calculated in **Magnum** is 7.1958×10^{-3} with a resolution of 126 elements per coil. The error was 0.03%.

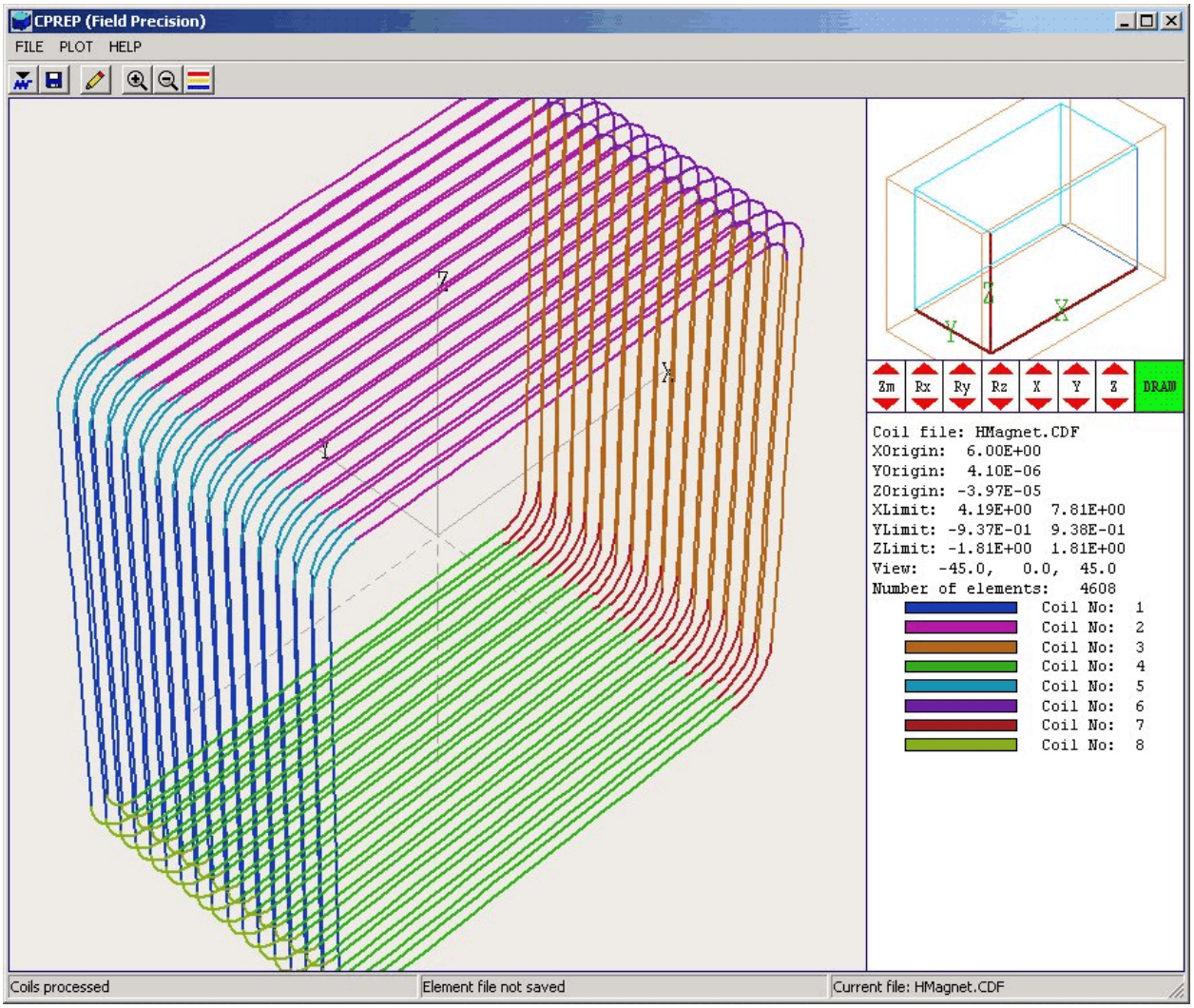

Figure 4.4. Working environment of **CPrep** in the interactive mode

4.3. Plots and listings

This chapter reviews features of **CPrep** in the interactive and autonomous modes. The interactive environment of Fig. 4.4 appears if you run the program as a standard Windows application. The alternative is to run **CPrep** from the command prompt:

```
/ProgPath/CPrep  /DataPath/FilePrefix
```

The quantity *ProgPath* in the command line represents the path to `CPrep.EXE`. You need not include this information if the program is in the current directory or if the directory of executables is on the system path. The quantity *DataPath* is the path to the file `FilePrefix.CDF`.

The main function of the autonomous mode is to perform extended series of runs under batch file control.

In the interactive mode, the **CPrep** *File menu* has the following entries

OPEN COIL FILE

You must open a coil file (`FilePrefix.CDF`) to begin any **CPrep** analysis session. If a previous file has been loaded, the program will ask whether you want to create an element file before loading a new file. The dialog shows all files with the suffix `CDF`. Changing directories in the dialog will change the working directory of the program.

SAVE ELEMENT FILE

Use this command to save the processed elements in a file `FileName.FSC.` The file is created in the working directory.

EDIT COIL FILE

This command loads the active Coil Definition File into an editor where you can make modifications. The compact editor has many of the features of standard Windows programs (*Cut, Copy, Paste, Find,...*). If you make modifications, save the file and re-open it in **CPrep** to see the changes. Close the editor to return to normal program operation.

EDIT FILE

This command calls the editor to work on any text file.

If the Coil Definition File can be processed correctly (*i.e.*, no syntax errors), then **CPrep** makes a default plot and the following commands of the *Plot* menu become active.

DISPLAYED COIL

Activate or deactivate the display of specific coils in the plot.

ZOOM VIEW

Magnify the three-dimensional plot by 33% about the plot origin.

EXPAND VIEW

Expand the three-dimensional plot by 33% about the plot origin.

SET PERSPECTIVE

The quantity *DView* is the distance from the origin to the viewpoint of the three-dimensional plot. Enter *DView* in units set by *DUnit*. A value comparable to the plot limits gives strong perspective. For an orthographic plot, choose a large value for *DView*.

To get a sense of the coil assembly geometry it is useful to move about in three-dimensional space.

The onscreen control bar is the section in the center-right portion of the of the screen(Fig. 4.4). The mouse cursor changes to a cross-hair pattern when you move it into the control region. The controls are identical to those used in **MetaMesh** for projection and surface plots. The commands have the following functions:

Zm

Zoom or expand the view. The approximate size the view volume is shown in the orientation section in the top-right portion of the screen.

Rx, Ry, Rz

Rotate about the *x, y* or *z* axes. The plot orientation is shown in the top-right portion of the screen.

X, Y, Z

Shift the view point along the *x, y* or z axes.

Draw

Update the main plot to reflect the rotation and shift commands. You can also click the right mouse button inside the control region to update the plot.

The information window on the right-hand side of the plot displays the following information:

- The name of the CDF file

- The present location of the plot origin (determined by *X*, *Y* and *Z displacement* parameters)
- The limiting extents of calculated elements
- The view angles: *X*, *Y* and *Z rotate*
- The number of elements
- The color-coding of coils.

When processing a file `FileName.CDF`, **CPrep** makes a listing file `FileName.CLS` of useful information. Chapter 14 describes the format of the element file (`FileName.FSC`) created by **CPrep**.

Chapter 5. Script commands for run control

5.1. Script conventions

Operation of **Magnum** is controlled by a script file with a name of the form `RUNNAME.GIN`. The input script for all **AMaze** solution programs is a text file with data lines containing commands and parameters. Text editing is a critical activity working with Field Precision software, both for preparing input and for inspecting quantitative data output. We supply a good multi-window editor (**Programmer's File Editor**) on the Magnum distribution disk.

The script must end with the *EndFile* command. The programs make no distinction between upper and lower case. Entries on a line can be separated by the following delimiters

```
Space, blank
Comma [,]
Tab
Colon [:]
Equal sign [=]
Left parenthesis [(]
Right parenthesis [)]
```

You can use any number of delimiters in a line. This means that you can add indentations and customize the appearance of the script. For example, the two lines

```
Mu 2 2300.0
Mu(2) = 2300.0
```

have the same effect.

Magnum ignores blank lines and comment lines. Comment lines begin with the symbol [*] (asterisk). Most parameters are real numbers. The following formats are valid.

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

The final number is interpreted as 5.0.

Magnum accepts commands in any order. The program reads and analyzes all commands before starting the solution. Generally, it is good practice to put control commands at the beginning and to group commands that set material properties by region. The following example illustrates a complete script for **Magnum**:

```
* File CMagnet.GIN
SOLTYPE Standard
DUNIT 100.0
SOURCE CMagnet.FSC
NCHECK = 25
MAXCYCLE = 2000
RESTARGET 5.0E-7

MU(1) = 1.0
MU(2) = 500.0
POTENTIAL(3) = 0.0
ENDFILE
```

You can place text in any format after the *EndFile* command. You can therefore add extended documentation that can be helpful when you return to a simulation.

5.2. Commands for all calculations modes

This section reviews commands that control program operation. Each command is shown in symbolic form and in the form that it might assume in a program.

SOLTYPE [Standard, Free, AddFree.Mapping]
SOLTYPE = Standard

This required command sets the **Magnum** solution type. Options for the single parameter are the strings *STANDARD, FREE, ADDFREE* and *MAPPING*. A *STANDARD* solution includes the effects of materials, either ideal conductors, permanent magnets or ferromagnetic materials. Usually, the input mesh (MDF) has conformal elements that follow the material boundaries. For solutions with coils the currents defined by the element file (FSC) interact with the materials. In the *STANDARD* mode, **Magnum** generates finite-element

solutions for the reduced potential ϕ (and the dual potential ψ when some materials have $\mu_r > 1.0$). In the *FREE* mode, there are no materials and the magnetic field is created only by applied currents. In this case the function of the mesh is to define node points at which to calculate $\mathbf{H}_s$ from the given current elements. The condition $\mu_r = 1.0$ applies over the full solution volume. The boundaries of the solution volume have no effect on the field values – the solution constitutes a segment of a solution in infinite space. In the *FREE* mode, **Magnum** makes no finite-element calculation and sets the values $\phi = 0.0$ and $\psi = 0.0$ in the output file. The *ADDFREE* mode is used to superimpose a free-space field on a bounded finite-element solution. This mode can be used for systems that combine pulsed fields (affected by conducting boundaries) with steady-state fields. Finally, the *MAPPING* mode superimposes data from multiple magnetic-field solution, either from **Magnum** or the 2D TriComp programs **BStat**, **PerMag**, **Pulse** or **Nelson**.

For clarification, the following paragraphs describe typical calculations in the four modes.

Standard Mode

The **MetaMesh** script `RUNNAME.MIN` describes a conformal mesh where regions represent magnetic materials such as poles and permanent magnets. The mesh generator creates the file `RUNNAME.MDF`. If the simulation includes coils, the file `SOURCENAME.CDF` defines the coil geometries. From this input **CPrep** generates the file `SOURCENAME.FSC`. The three files required by **Magnum** for a solution with coils are `RUNNAME.MIN`, `RUNNAME.GIN` and `SOURCENAME.FSC`. **Magnum** computes the node values H_{xs}, H_{ys}, H_{zs}, ϕ and ψ. The quantities are used in **MagView** to determine total fields resulting from coils and materials.

Free Mode

In *FREE* mode calculations usually the input mesh is non-conformal (box elements) and contains a single region to represent vacuum. Nonetheless, you could employ a conformal mesh with multiple regions if you wanted to use the capabilities of **MagView** to find field energy in or flux through shaped objects. A current element file `SOURCENAME.FSC` is required in this mode. **Magnum** computes nodal values of H_{xs}, H_{ys}, H_{zs} and sets $\phi = 0$ and $\psi = 0$ at all points (*i.e.*, no material contributions).

AddFree Mode

A run in the *ADDFREE* mode is similar to one in the *STANDARD* mode. An additional current element file, FREESOURCENAME.FSC, is required. After performing the standard finite-element calculation, **Magnum** supplements the values of H_{sx}, H_{sy} and H_{sz} with free-space field components calculated from the current elements of FREESOURCENAME.FSC.

Mapping Mode

A *MAPPING* run combines field values from one or more magnetic field solutions into a single solution file. The calculation requires a mesh file (RUNNAME.MDF) to define node points for the calculations and one or more magnetic-field solution files: ANYNAME.GOU (3D) or ANYNAME.BOU (2D). The program performs interpolations within the solution files at positions corresponding to the nodes of RUNNAME.MDF. The output file with the name RUNNAME.GOU is in standard **Magnum** format.

The following program control commands may appear in all modes.

DUNIT DUnit
DUNIT = 100.0

Internally **Magnum** uses SI units, with lengths in meters (m), magnetic flux density **B** in tesla, and magnetic field intensity **H** in amperes/meter (A/m). Use the *Set DUnit* command to work in alternate length units. The real-number quantity is the number of working units per meter. For example, to signal that you used dimensions of centimeters in the **MetaMesh** script, set *DUnit* = 100.0. Alternatively, if the mesh dimensions are in inches, use *DUnit* = 39.37. The quantity *DUnit* is recorded in the output file and is used in **MagView** for the input and output of positions. Default: *DUnit* = 1.0.

FORMAT [Text, Binary]
FORMAT = Text

By default, **Magnum** creates output files FPREFIX.GOU in compact binary format. Binary output files are required if you want to use **MagView** to analyze the solution. We have included an option to create output files in text format to make it easier to port results to your own analysis programs. The string parameter can assume the values *BINARY* or *TEXT*. Chapter 14 describes the file structure for both formats.

5.3. Commands to control field-solution runs

The *SOURCE* command may appear in *STANDARD, FREE* and *ADDFREE* calculations.

SOURCE SourceFileName
SOURCE = CMagnet.FSC

This command specifies an element file that lists drive currents for the applied fields. The parameter *SourceFileName* is the full name of the file. The program issues an error message if the file is not available in the working directory. The currents are used to generate applied fields in the *STANDARD* and *FREE* modes. In the *ADDFREE* mode, the currents are used as the source terms for the finite element calculation of ϕ (and possibly ψ). The command is invalid in a *MAPPING* calculation.

The following commands are valid only in STANDARD and ADDFREE calculations.

RESTARGET Restarget
RESTARGET: 5.0E-6

The numerical calculation of fields with magnetic materials requires the solution of a large set of coupled linear equations, one for each active node in the solution volume. **Magnum** uses an iterative technique based on corrections that reduce the error in the reduced or dual potential at a point compared to predictions from values at neighboring nodes. The *residual* is an average of the relative errors over all nodes in the solution space during an iteration. This command sets a target value for the residual. The program stops if the error drops below the value. For good accuracy, the relative residual should be less than 10^{-6}. If the value of *ResTarget* is too low, the program may not converge because of roundoff errors. In this case, you can terminate a solution manually if you are running **Magnum** in the interactive mode by using the *Stop and save run* or *Abandon run* menu options. Default value: *ResTarget* = 5.0×10^{-7}.

MAXCYCLE MaxCycle
MAXCYCLE = 2500

The integer parameter *MaxCycle* is the maximum number of iteration cycles. **Magnum** saves the solution and stops when it reaches *MaxCycle* even if *ResTarget* has not been attained. Default value: *MaxCycle* = 2500.

NCHECK NCheck
NCHECK = 20

The operations to calculate *ResTarget* take time; therefore, **Magnum** usually does not make a check on every iteration cycle. The integer parameter is the number of relaxation cycles between convergence checks. Default value: *NCheck* = 50.

OMEGA Omega [OmegaDual]
OMEGA: 1.92

The real-number parameter *Omega* is the over-relaxation factor used to correct potential errors during the solution for the reduced potential. This quantity may have a significant effect on the run time. If the *Set omega* command does not appear, **Magnum** automatically picks values for the reduced-potential calculation that vary with the iteration cycle following the Chebyshev acceleration prescription. With this command, you can set a value manually. The value must be between 0.0 and 2.0 for numerical stability. Higher values generally give faster convergence. You can optionally include an independent value for the relaxation factor to use for the dual-potential calculation (inside iron regions). Generally, *OmegaDual* should be smaller than Omega for convergence. The default is *OmegaDual* = 0.6.

OMITIRON

The dual-potential calculation does not affect values in the reduced-potential calculation. In other words, it is not necessary to calculate fields inside iron regions to obtain correct field values in air. If you do not need accurate field values within iron structures, you can reduce run time by skipping the dual potential calculation with this command.

NGAUSS NGauss
NGAUSS = 3

The parameter *NGauss* controls the number of intervals in Gaussian-quadrature integrations to determine element matrices. Under normal circumstances, it should not be necessary to make any change from the default value of *NGauss* = 4. Lower values decrease the program setup time but may reduce accuracy.

The following command is valid only in *ADDFREE* mode calculations.

SOURCEFREE SourceFreeFileName
SOURCEFREE = HelmholtzBias.FSC

This command may appear only in runs in the *ADDFREE* mode and usually appears in conjunction with a *SOURCE* command. The file lists drive currents for free-space fields that will be added to $\mathbf{H}_s$ values in the output file after the finite-element calculation of ϕ (and possibly ψ) have been completed.

5.4. Mapping commands

The following six commands control field interpolations and may appear only in the *MAPPING* mode.

MAP2 MFileName [MScale]
MAP2 = SOL_LENS.BOU (0.479)

This command specifies the name of an output file from a 2D TriComp magnetic-field calculation that will be mapped into the space of the mesh described by `RUNNAME.MDF`. The string *MFileName* is the full name of a file available in the working directory. When the optional real-number parameter *MScale* appears, transferred magnetic field values are multiplied by the value. Output files from the following programs are recognized: **BStat** (`MPREFIX.BOU`), **PerMag** (`MPREFIX.POU`), **Nelson** (`MPREFIX.NOU`) and **Pulse** (`MPREFIX.POU`). A maximum of 5 *MAP2D* commands may appear in the script.

In the 2D mapping process, **Magnum** loops through all nodes of the mesh defined by `RUNNAME.MDF`. Coordinates in the 3D mesh are transformed to those in the 2D solution according to parameters set by the *SHIFT2* and *ROT2* commands. A field interpolation appropriate to the solution symmetry is performed in the 2D space to find values of B_x, B_y and B_z. The program then sets values of the magnetic intensity at the node: H_{sx} = *MScale* B_x/μ_o, H_{sy} = *MScale* B_y/μ_o, H_{sz} = *MScale* B_z/μ_o. Values of the reduced and dual potentials are set equal to zero, $\phi = 0$ and $\psi = 0$. As a result, region divisions in the mesh have no effect on field values calculated in **MagView**. (Note that region divisions may be useful to define shapes for volume or surface integrals.) In the event of an invalid interpolation (*i.e.*, point outside the solution boundaries), **Magnum** sets $H_{sx} = 0.0$, $H_{sy} = 0.0$ and $H_{sz} = 0.0$ and writes a warning message in the listing file `RUNNAME.GLS`.

ROT2 Rx Ry Rz
ROT2 = (0.0, 45.0, 90.0)

This command rotates the 2D file with respect to the 3D mesh. Specify the angles R_x, R_y and R_z in degrees. The rotation is performed in the order $R_x \rightarrow R_y \rightarrow R_z$. The default values are $R_x = 0.0$, $R_y = 0.0$ and $R_z = 0.0$. In the default case, the *z* axes of the 2D and 3D spaces are coincident. For a cylindrical 2D solution, the *r* direction lies in the *x-y* plane. For a planar solution, the *x* and *y* directions in the 2D and 3D spaces are coincident and $B_z = 0.0$.

SHIFT2 Sx Sy Sz
SHIFT2 = (10.0, 0.0, -20.0)

This command shifts the 2D solution with respect to the 3D mesh. Enter displacements in units by *DUnit*. Note that shifts are performed *after* rotations.

MAP3 MFileName [MScale]
MAP3 = CMAGNET.GOU

This command specifies the name of a **Magnum** binary output file (`MPREFIX.GOU`) that will be mapped into the space of the mesh described by `RUNNAME.MDF`. The string *MFileName* is the full name a file available in the working directory. When the optional real-number parameter *MScale* appears, transferred magnetic field values are multiplied by the value. Five *MAP3* commands may appear.

In the 3D mapping process, **Magnum** loops through nodes defined by `RUNNAME.MDF`. Coordinates in the mesh are transformed to those in field solution according to parameters set in the *SHIFT3* and *ROT3* commands. The field interpolations in the 3D space incorporate information on material properties and ϕ and ψ and yield total values of B_x, B_y and B_z. The program then sets values of the magnetic intensity at the node: $H_{sx} = MScale\ B_x/\mu_o$, $H_{sy} = MScale\ B_y/\mu_o$, $H_{sz} = MScale\ B_z/\mu_o$. Values of the reduced and dual potentials are set equal to zero, $\phi = 0$ and $\psi = 0$.

ROT3 Rx Ry Rz
ROT3 = (0.0, 45.0, 90.0)

This command rotates the 2D file with respect to the 3D mesh. Specify the angles R_x, R_y and R_z in degrees. The rotation is performed in the order $R_x \rightarrow R_y \rightarrow R_z$. The default values are $R_x = R_y = R_z = 0.0$.

SHIFT Sx Sy Sz
SHIFT3 = (10.0, 0.0, -20.0)

This command shifts the 2D solution with respect to the 3D mesh. Enter displacements in units by *DUnit*. Note that shifts are performed after rotations.

Table 5.1. Allowed commands in the solution modes

Command	Standard	Free	AddFree	Mapping
DUNIT	●	●	●	●
FORMAT	●	●	●	●
SOURCE	●	●	●	
RESTARGET	●		●	
MAXCYCLE	●		●	
NCHECK	●		●	
OMEGA	●		●	
OMITIRON	●		●	
NGAUSS	●		●	
SOURCEFREE			●	
MAP2				●
ROT2				●
SHIFT2				●
MAP3				●
ROT3				●
SHIFT3				●
MU	●		●	
POTENTIAL	●		●	
PERMAG	●		●	

Chapter 6. Commands to set material properties

Material properties in magnetostatic solutions with linear materials are relatively simple so only a few commands are required. Both symbolic and sample forms given. These commands apply only in the *STANDARD* and *ADDFREE* modes.

MU RegNo MuR
MU(4) = 523.0

This command sets the relative magnetic permeability for all elements associated with a filled region. The integer parameter is the number of a region defined in **MetaMesh.** The second parameter is the value of relative magnetic permeability μ/μ_o (dimensionless). To model a highly-conducting material (*i.e.*, magnetic field excluder), assign a small value of magnetic permeability (*i.e.*, 10^{-4}) to the region. Note that excessive small or large values of μ_r (*i.e.*, 10^6) may slow the solution convergence. The default for all unspecified regions is μ/μ_o = 1.0.

POTENTIAL RegNo Phi
POTENTIAL(5) = 0.0

The main use of this command is to implement symmetry boundaries where the magnetic flux density is normal to the surface. The value applies to both ϕ (in the reduced potential calculations) and to ψ (in the dual potential calculation). The material contribution to the magnetic flux density is normal to a boundary with ϕ = *constant*, $\mathbf{B}_m$(parallel) = 0.0. The total magnetic flux density is normal to a boundary with ψ = 0.0, **B**(parallel) = 0.0. You can pick any value for ϕ or ψ if there is a single symmetry boundary in a solution. In this only relative differences in the potential are significant. You must exercise caution when there are two boundaries. Setting fixed values may introduce artifices. In the example of Section 1.2, the symmetry of the fields dictates that the line integral of **B** and $\mathbf{B}_m$ between the planes x = 0.0 and y = 0.0 is always zero. Therefore, it is valid to set ϕ = 0.0 and ψ = 0.0 on both boundaries.

PERMAG RegNo Br Ux Uy Uz
PERMAG 6 1.12 0.00 0.00 1.00

This command defines an ideal permanent magnet material. Four real-number parameters follow the region number. The quantity B_r is the remanence magnetic field in tesla. The quantities [U_x, U_y, U_z] define a vector that points along the axis of magnetization (easy axis). **Magnum** automatically normalizes these numbers to define a unit vector.

An ideal permanent magnet is one in which the state of domain orientation does not depend on external influences such as the dimensions of air gaps or the presence of coils and other permanent magnets. In this case the normal demagnetization curve (**B** versus μ_o**H**) is a straight line with a slope of -45°. Modern materials like neodymium-iron and samarium-cobalt closely approximate this condition. The model applies to conventional magnet materials (like Alnico) only for moderate values of coercive force ($|\mathbf{H}| \le |\mathbf{H}_c|$). For a discussion of the properties of permanent magnets, see S. Humphries, **Field Solutions on Computers** (CRC Press, Boca Raton, 1997), Sects. 9.6 and 9.7 and S. Humphries, **Principles of Charged Particle Acceleration** (Wiley, New York, 1986), Sect. 5.8. The later book can be download at no charge from our Internet site at http://www.fieldp.com/cpa/cpa.html.

Figure 6.1 shows a two-dimensional benchmark calculation performed with **Magnum**. The example of a ring dipole is taken from J.L. Warren, et.al.**, Reference Manual for the Poisson/Superfish Group of Codes** (Los Alamos National Laboratory, LA-UR-87-126, 1987), pg. 10-17. The theoretical value for the dipole field on axis is 1.0 tesla. The value is closely approximated in two-dimensional numerical calculations by the Los Alamos **Pandira** code and the Field Precision **PerMag** code. The **Magnum** calculation gives 1.003 tesla. Figure 6.2 shows a more interesting calculation where the three-dimensional capabilities of Magnum are required, a ring dipole with finite length along *z*. The plane shows color coded values of |**B**| in the magnet midplane.

The following Internet sites provide useful information on permanent magnet design:

http://www.magnetsales.com/Design/DesignG.htm
http://www.magnetweb.com/design2.htm
http://www.magnet.au.com/fluxgraphs-entry.html
http://www.intemag.com/designtools/designguide.asp

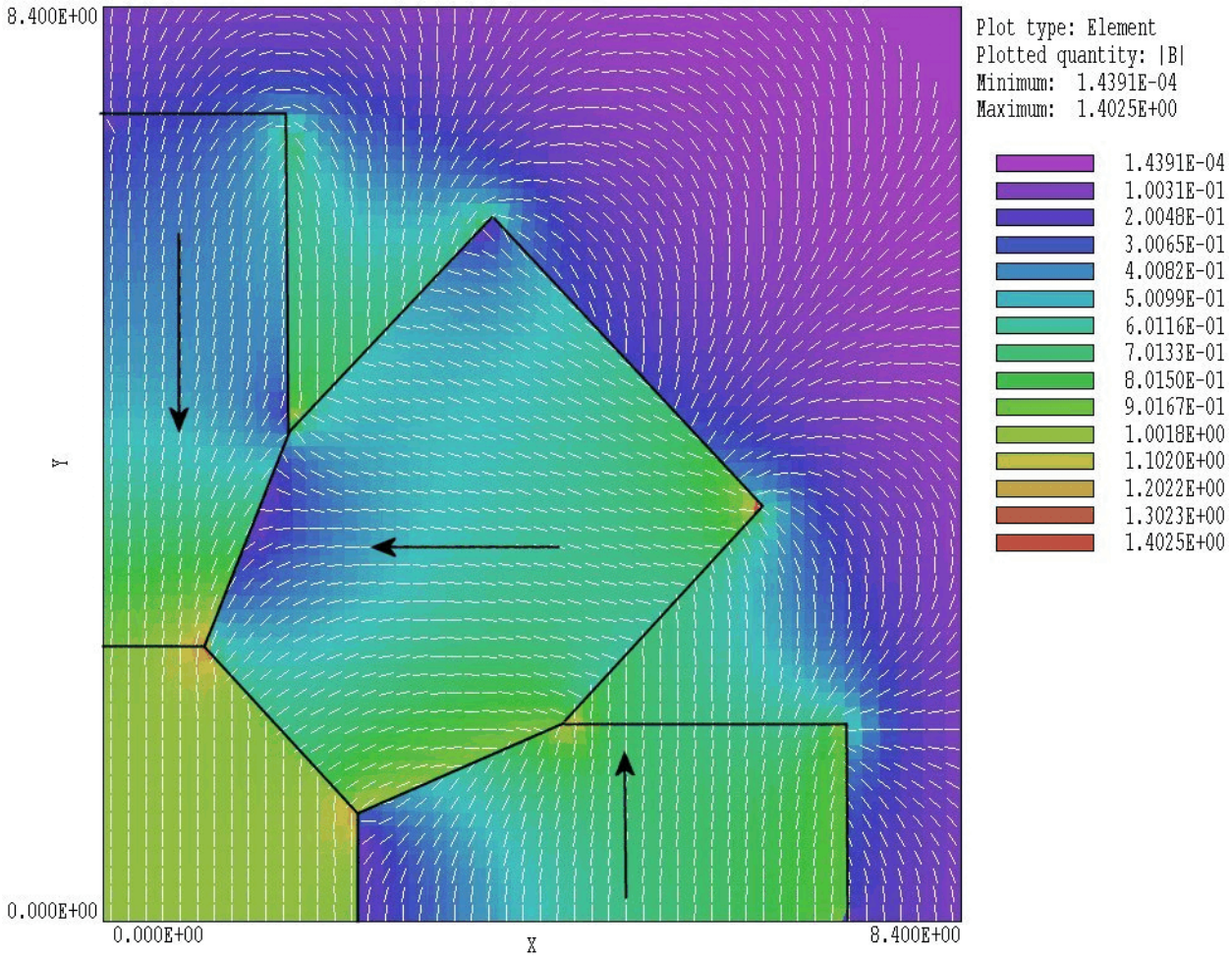

Figure 6.1. Two-dimensional **Magnum** calculation of a permanent-magnet ring dipole (single quadrant with symmetry boundaries). Vector lines with color-coding by |**B**|. Black arrows show magnetization vectors.

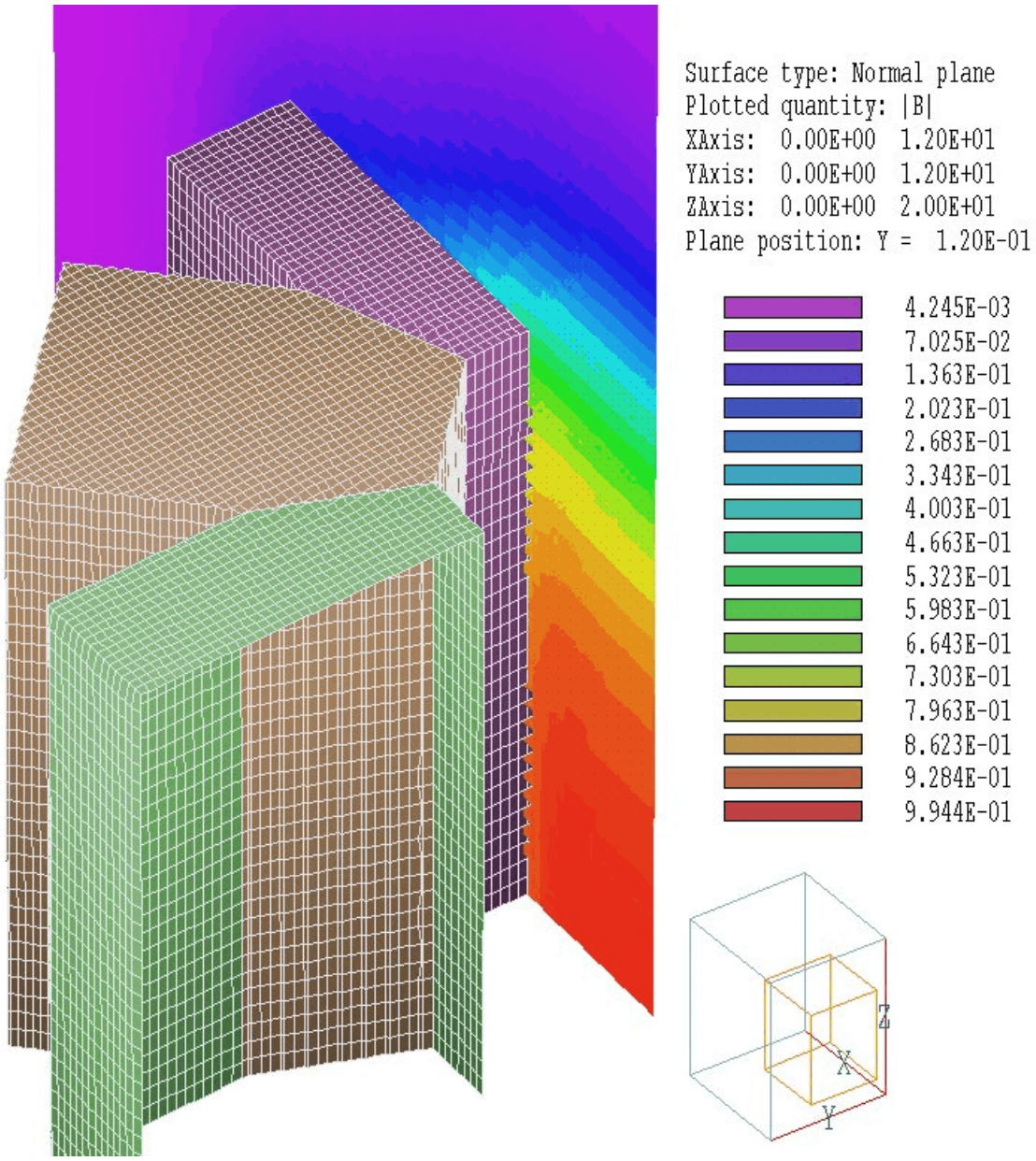

Figure 6.2. Three-dimensional **Magnum** calculation of a finite-length permanent-magnet ring dipole. Color coding shows variations of |**B**| in the plane $y = 0.0$.

Table 6.1. Properties of some permanent magnet materials					
Material	**B_r (tesla)**	**$\mu_o H_c$ (tesla)**	**Material**	**B_r (tesla)**	**$\mu_o H_c$ (tesla)**
17 % Cobalt steel	0.95	0.015	NdFeB 27	1.05	1.00
36% Cobalt steel	0.95	0.024	NdFeB 27H	1.085	1.00
Alnico 1	0.72	0.044	NdFeB 28	1.08	1.01
Alnico 12	0.58	0.095	NdFeB 30	1.10	1.06
Alnico 2	0.72	0.055	NdFeB 30H	1.12	1.07
Alnico 2 (sintered)	0.69	0.052	NdFeB 32	1.16	1.08
Alnico 4	0.55	0.070	NdFeB 32H	1.15	1.09
Alnico 5	1.25	0.055	NdFeB 35	1.23	1.13
Alnico 5 (sintered)	1.09	0.062	NdFeB 35H	1.21	1.16
Alnico 6	1.00	0.075	NdFeB 37	1.24	1.18
Alnico 8	0.82	0.165	Neo Flex6	0.545	0.49
Alnico 8 (sintered)	0.74	0.150	Platinum-cobalt	5.90	0.36
Carbon steel	1.00	0.005	Remalloy or Comol	1.05	0.025
Ceramic 1	0.23	0.186	Silmanal	0.055	0.60
Ceramic 5	0.38	0.24	SmCo 16	0.80	0.76
Ceramic 8	0.385	0.295	SmCo 18	0.85	0.80
Chromium steel	0.97	0.0065	SmCo 20	0.905	0.83
Cunico	0.34	0.066	SmCo 22	0.95	0.86
Cunife (wire)	0.54	0.055	SmCo 22B	0.96	0.78
Flexible regular	0.16	0.137	SmCo 24	0.995	0.72
HF1	0.22	0.20	SmCo 26	1.05	0.92
HF2	0.245	0.22	SmCo 28	1.075	0.84
HF3	0.265	0.22	Tungsten steel	1.03	0.007
Hyflux Fine powder	6.60	0.039	Vectolite	1.60	0.10
NdFeB 24	0.98	0.95	Vicalloy 1	0.88	0.030
NdFeB 26	1.04	1.00	Vicalloy 2 (wire)	1.00	0.051

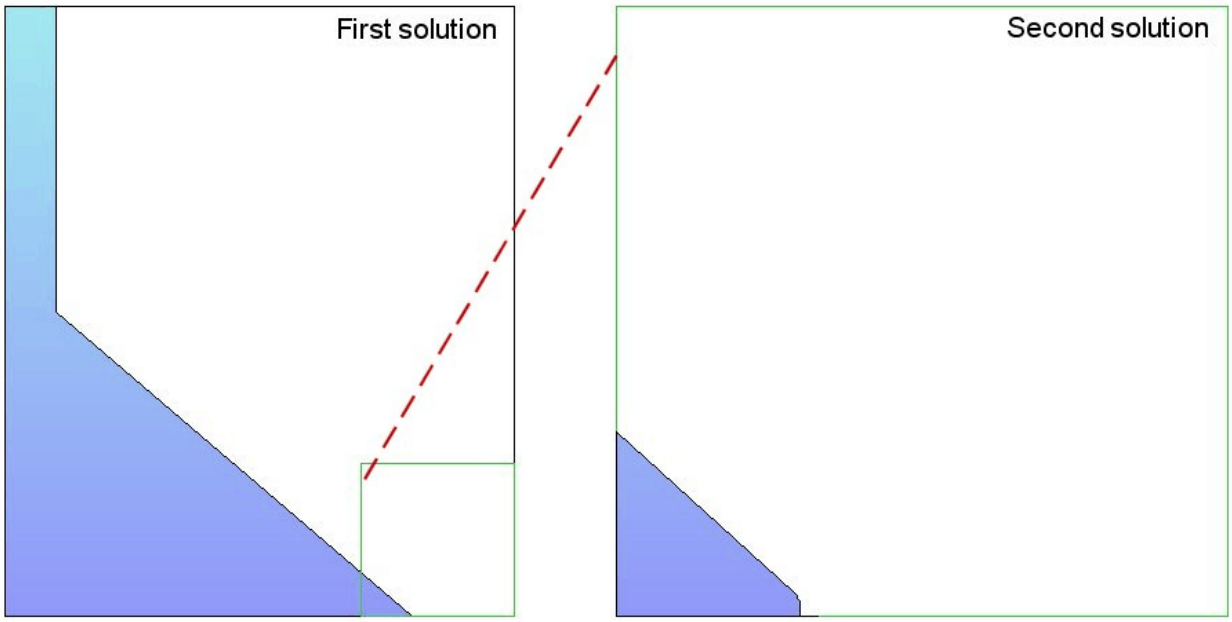

Figure 7.1. Creating a microscopic solution using the *BOUNDARY* command

Chapter 7. Modeling small details in a large-scale solution

The **AMaze** solution programs incorporate a useful capability that enables accurate calculations of fields near small features in a large solution space. Suppose we wanted a precision calculation of the electrostatic fields on a field emission tip. The radius of the tip is much smaller than the scale size of the electrodes that create the field and hold the tip. We need to do a large-scale calculation to find the macroscopic fields (Solution 1), but we would need very small elements near the tip to resolve the curvature. One approach is to use variable mesh resolution (**MetaMesh** manual, Section 4.2) to create small elements near the tip. A limitation to this technique arises from the structured mesh used in the **AMaze** programs – the region of small elements extends the full length of the solution volume.

Figure 7.1 illustrates an alternate approach. We create a second solution that covers a small subregion of the original solution (green outline). The microscopic solution contains any magnetic materials that are located within the subregion. The difference is that the material shapes may be resolved in much greater detail. The question is how to incorporate the

macroscopic fields correctly into the microscopic solution. One resolution is to enclose the second solution within a Dirichlet boundary. Values of the potentials ϕ and ψ on the boundary are calculated by interpolation at the corresponding point in the macroscopic solution (dashed red line). The total solution will be approximately correct as long as new features (such as the rounded tip) are well-removed from the variable-potential boundary.

Definition of the variable Dirichlet boundary is controlled by the following command that appears in the control script of Solution 2:

BOUNDARY FPrefix [BndScale]
BOUNDARY FEmitMacro

The quantity *FPrefix* (a string) is the prefix of the output file for Solution 1. The quantity *BndScale* (a real number) is an optional scaling factor applied to potential values transferred from Solution 1 to Solution 2. The default is *BndScale* = 1.0.

In response to the *BOUNDARY* command, **Magnum** loads Solution 1 into memory and sets up the apparatus for making second-order interpolations of reduced potential ϕ and dual potential ψ in the solution. Then, the program loops through all nodes of Solution 2. If the node is on the boundary of the solution box, **Magnum** marks it as a fixed-potential point and sets the two potentials equal to the interpolated value calculated at the position in Solution 1. **Magnum** issues an error message under the following conditions: 1) the output file for Solution 1 (`FPrefix.GOU`) is not available in the working directory, 2) Solution 2 does not fit completely inside Solution 1, or 3) an interpolation failed. The program makes no further validity checks. You must ensure that the geometry of Solution 2 represents a correct microscopic section of Solution 1.

The procedure is best illustrated with an example. The runs *BOUNDTEST01* and *BOUNDTEST02* are included in the example library. The files `BOUNDTEST01.MIN` and `BOUNDTEST02.GIN` describes a large-scale solution, while `BOUNDTEST02.MIN` and `BOUNDTEST02.GIN` describe a microscopic solution, a subset of the first. Figure 7.2 shows the large-scale benchmark solution, an offset cylindrical conductor inside a conducting pipe. The solution covers a volume extending from -3.1 cm to 3.1 cm in *x* and *y* and from 0.0 cm to 15.0 cm in *z*. We want to find accurate field values in the small gap region shown as a dashed line in Fig. 7.2. We create a solution that extends from 1.0 cm to 3.1 cm in *x*, from -1.0 cm to 1.0 cm in *y* and from 10.0 cm to

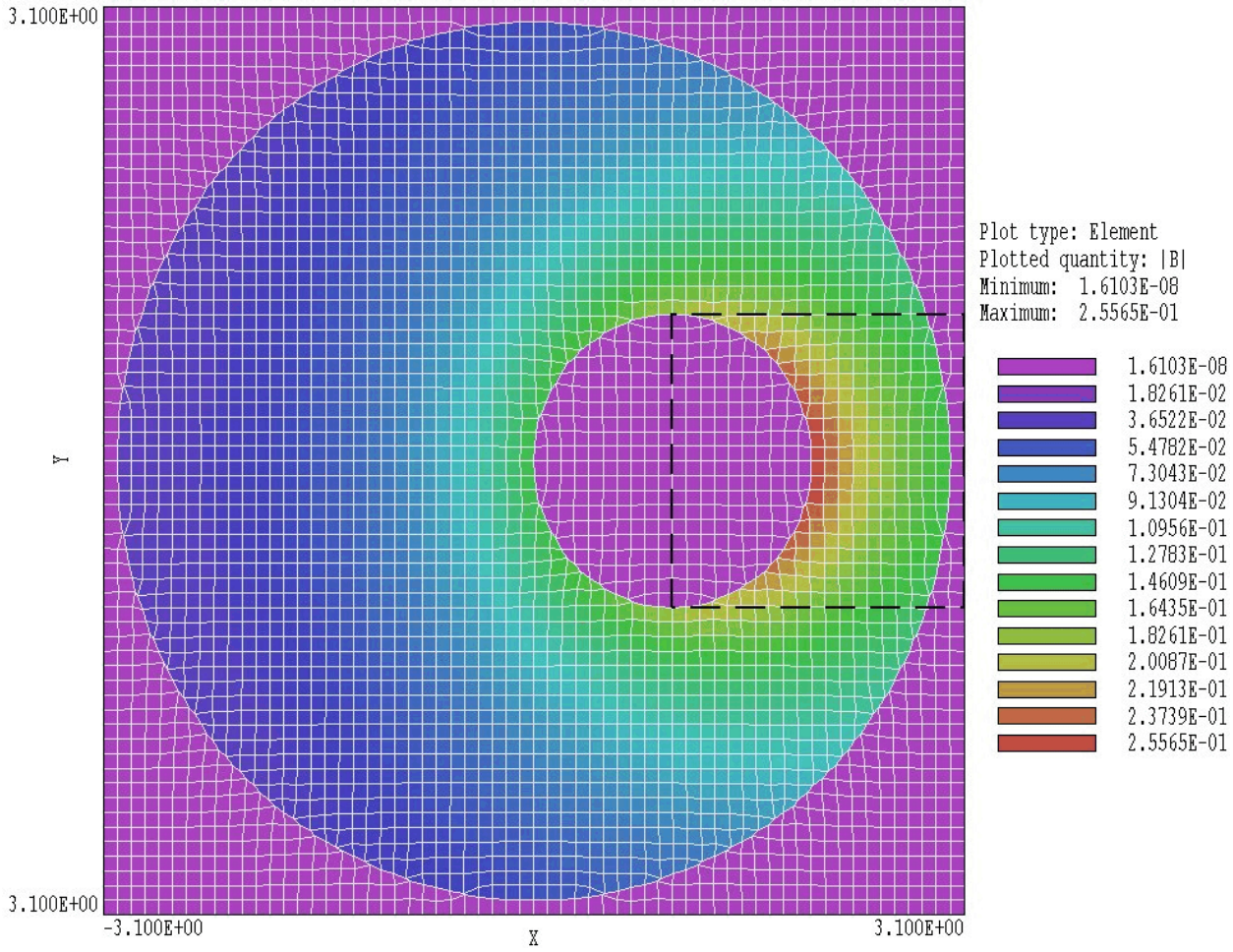

Figure 7.2. Cross-section of solution *BOUNDTEST01* in the *x-y* plane.

12.0 in *z*. To create the mesh, we simply change the limits and resolution in the *GLOBAL* section of *BOUNDTEST01* to:

```
Global
  XMesh
     1.00   3.10  0.05
  End
  YMesh
    -1.00   1.00  0.05
  End
  ZMesh
    10.00   12.00  0.05
  End
End
```

All the *PART* sections remain the same – **MetaMesh** ignores any sections of parts that lie outside the solution box. The **Magnum** script file `BOUNDTEST02.GIN` is almost identical to `BOUNDTEST01.GIN`:

```
* File BOUNDTEST02.GIN
SOLTYPE Standard
RESTARGET 5.0E-7
DUNIT 100.0
OMEGA 1.95
MAXCYCLE 500
SOURCE BOUNDTEST.FSC
MU(1) = 0.00001
MU(2) = 1.0
MU(3) = 0.00001
NCHECK 10
BOUNDARY BOUNDTEST01
ENDFILE
```

The exception is the line:

```
BOUNDARY BOUNDTEST01
```

Note that the solution *BOUNDTEST02* reads the same source file (`BOUNDTEST.FSC`) as *BOUNDTEST01.* The applied field calculation is the same for both runs except that the Biot-Savart integral is performed on a finer mesh in *BOUNDTEST02.*

Figure 7.3 shows the resulting calculation. Field scans along the *x* axes of *BOUNDTEST01* and *BOUNDTEST02* give almost identical values, confirming the validity of the method. Note that there may be small inaccuracies in regions where physical objects intersect a Dirichlet boundary (top and bottom in Fig. 7.3). A more practical application would be to generate a global solution and then to add small details to the microscopic solution, such as the object illustrated in Fig. 7.3.

In response to the *Superposition* command, **Magnum** superimposes values from a large-scale solution (Solution 1) on a small-scale solution (Solution 2). The following statement may appear in the command script for Solution 2.

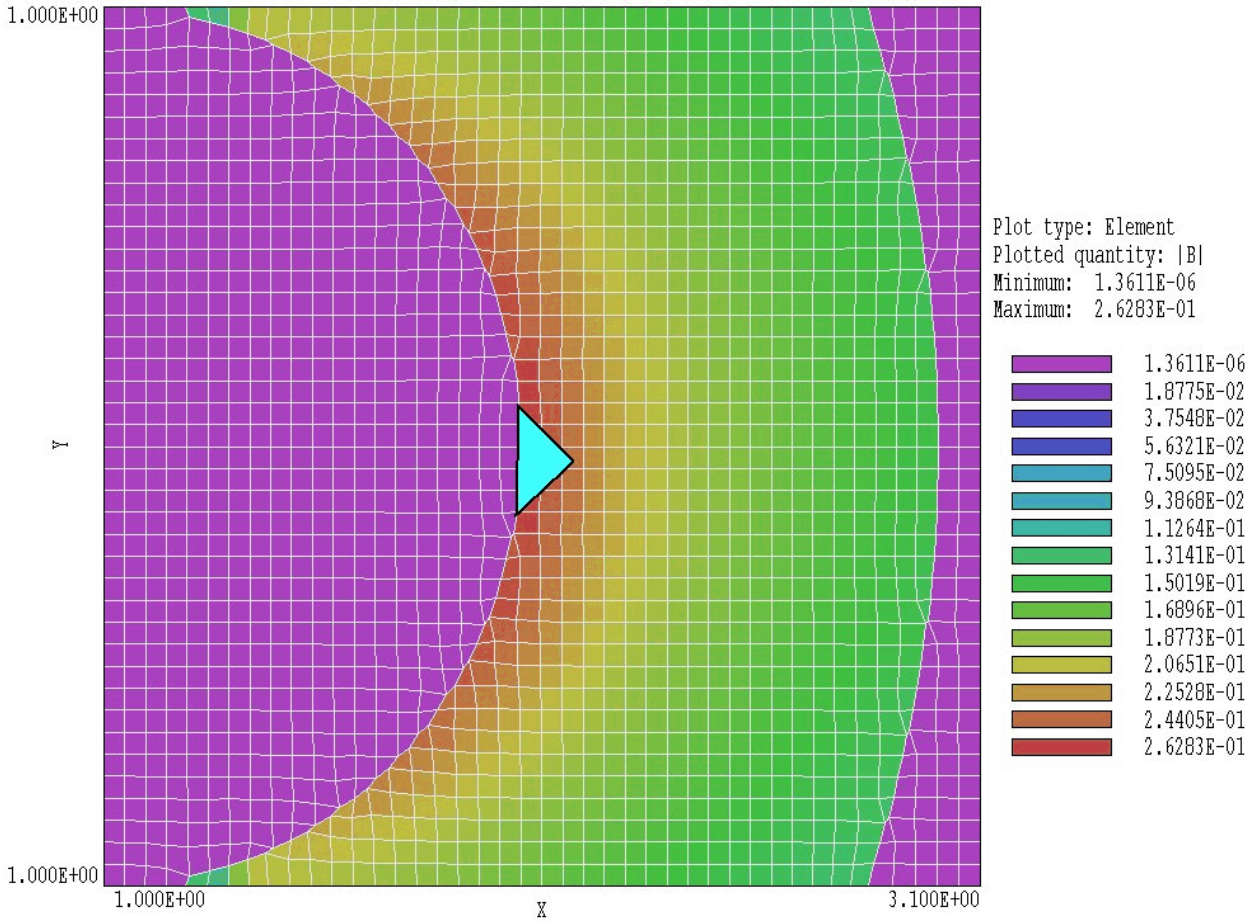

Figure 7.3. Cross-section of solution *BOUNDTEST02* in the *x-y* plane. The blue object illustrates a possible small-scale feature that could be added to the simulation.

SUPERPOSITION FPrefix [SuperScale]
SUPERPOSITION UniField

The quantity *FPrefix* (a string) is the prefix of the output file for Solution 1. The quantity *BndScale* (a real number) is a scaling factor applied to potential values transferred from Solution 1 to Solution 2. The default is *BndScale* = 1.0.

When the *SUPERPOSITION* command is issued, **Magnum** opens the file `FPrefix.GOU` after Solution 2 has been completed. The program performs an interpolation in the space of Solution 1 to determine the reduced potential ϕ_1, applied magnetic field H1 and dual potential $\psi 1$ at each node location in Solution 2 and adjusts values in the solution according to

$$\phi_2' = \phi_2 + SuperScale \times \phi_1 ,$$

$$\boldsymbol{H}_2' = \boldsymbol{H}_2 + SuperScale \times \boldsymbol{H}_1 , \qquad \textbf{(1)}$$

$$\psi_2' = \psi_2 + SuperScale \times \psi_1 .$$

before writing the output file.

You must ensure that the superposition is physically correct. In magnetostatic solutions the presence of perfect conductors and ferromagnetic materials in Solution 2 can make significant local changes to the macroscopic field of Solution 1, so that a simple superposition would be invalid.

Chapter 8. Running Magnum interactively

The program MAGNUM.EXE can be run in interactively in a window. In this mode you can perform several solutions in a session and temporarily leave the program to work on other tasks. You can launch the program from **AMaze** or create you own shortcuts. Figure 8.1 shows the program window.

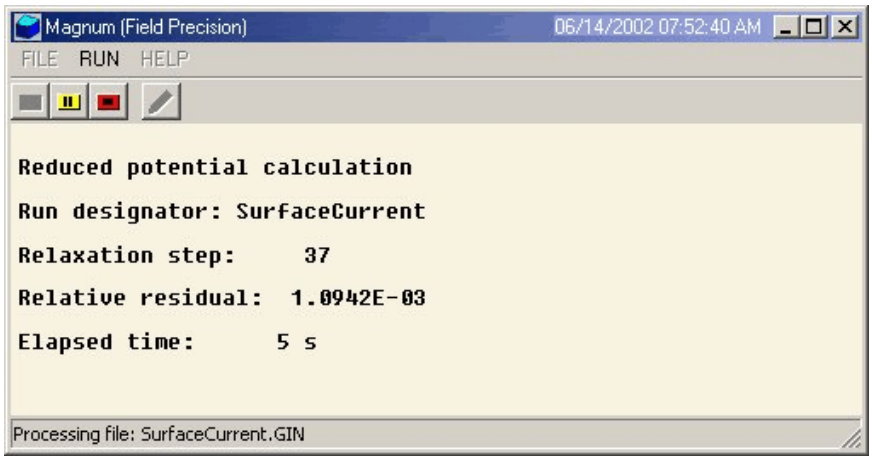

Figure 8.1. **Magnum** display during a solution

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

EDIT SCRIPT FILE
EDIT LISTING FILE
EDIT FILE

The commands call up the internal editor to inspect or to modify text input and output files for the solution program. With the *Edit script file* command you can work on files with names of the form FPREFIX.GIN. With the *Edit listing file* command you can pick files with names of the form FPREFIX.GLS. The *Edit file* command shows all available files. Choosing a file from an alternate directory does not change the working directory of the program.

The *Run* menu has three commands.

START RUN

Pick an input file with a name of the form `FPREFIX.GIN` to start a solution. The working directory changes if you pick a file from an alternate directory. The run begins if the file `FPREFIX.MDF` is available in the working directory. **Magnum** displays timer bars to show progress when the program is loading a mesh file, calculating applied fields, calculating elements matrices and then generating coupling coefficients of the linear equation set. When these tasks are completed, the program begins the relaxation solution for the reduced potential and reports process in the text window (Fig. 8.1). An additional relaxation solution may occur for the dual potential.

PAUSE RUN

The intensive calculations of **Magnum** 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 during the relaxation process and restart it later without loss of data.

STOP RUN

This command terminates the program during the relaxation process and saves output data. For example, you may want to stop a run at a moderate value of convergence to check whether the problem has been correctly defined.

The *Help* menu has a single command.

INSTRUCTIONS

Displays the HTML help file using your default browser. In order for this command to operate correctly, the file `MAGNUM.HTML` must be in the same directory as `MAGNUM.EXE`.

Chapter 9. Automatic runs under batch file control

9.1 Command line operation

Batch file control is a useful option for running large technical programs like **Magnum**. You can prepare scripts to organize complex operations. The sequenced programs run automatically in the background or overnight.

To make a single **Magnum** simulation in the background, go to the Command Prompt from Windows and log to the data directory that contains the required `MDF`, `FSC` and `GIN` files.. For example, suppose the data files `SEPMAGNET.MDF`, `SEPMAGNET.FSC` and `SEPMAGNET.GIN` are stored in `\AMAZE\BUFFER` and that the program `MAGNUM.EXE` is in the directory `\AMAZE`. From `\AMAZE\BUFFER` type

```
..\MAGNUM SEPMAGNET <Enter>
```

The program runs silently, writing detailed information in the listing file `SEPMAGNET.GLS`. If the solution is successful, the program writes the output file `SEPMAGNET.GOU` in the data directory. During lengthy runs you can perform other tasks in Windows. Note that the response of the computer may be considerably slower because **AMaze** programs seek to use the full power of the CPU.

9.2. Batch files

The main function of the command mode is autonomous operation under batch file control. As an example, assume you have prepared the input files `SEPMAGNET01.MDF,...,SEPMAGNET08.MDF` and `SEPMAGNET01.GIN,...,SEPMAGNET08.GIN` in the directory `\AMAZE\BUFFER`.. Next you create the following batch file `SMAGRUN.BAT` in the data directory using a text editor.

```
@ECHO OFF
ECHO Main septum magnet data run
START ..\METAMESH.EXE SEPMAGNET01
START ..\MAGNUM.EXE SEPMAGNET01
START ..\METAMESH.EXE SEPMAGNET02
START ..\MAGNUM.EXE SEPMAGNET02
 ...
```

```
START ..\METAMESH.EXE SEPMAGNET08
START ..\MAGNUM.EXE SEPMAGNET08
```

Type

```
SMAGRUN <Enter>
```

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

9.3. Introduction to GCon

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. For example, Windows 2000 runs a set of programs in the proper sequential order for any valid batch format. Windows 95 starts all programs simultaneously unless you use the command form:

```
START /WAIT  ProgName
```

Windows ME does not recognize the `START` command at all. And so on.... To ensure consistent batch file operation we supply the utility program **GCon** with all our software. To avoid problems we advise running batch scripts from **GCon** rather than from the Command Prompt.

GCon handles the following subset of DOS commands. Additional capabilties are described in the file `GCON.HTML`.

CD

Function: Displays the name of or changes the current directory.
Example: `CD C:\AMaze\Buffer`

COPY ... TO ...

Function: Copies a file to another location.
Example: `COPY Diode.min TO C:\AMaze\Diode.min`

ERASE

Function: Deletes a file.
Example: `ERASE diode.mls`

REM

Function: Displays the line, no action taken

MOVE ... TO ...

Function: Moves a file from one directory to another directory.
Example: `MOVE Diode.mou TO C:\AMaze\Diode.mou`

RENAME

Function: Renames a file or files.
Example: `RENAME diode.mou diode_compare.mou`

RUN

Function: Runs a specified program or command.
Example: `RUN C:\tricomp\mesh diode`

As an example, the following script creates a mesh and an electrostatic solution using the input files `DIODE.MIN` and `DIODE.HIN` in the directory `C:\EStudies\Inputs` and stores the results in the directory `C:\EStudies\Outputs`

```
REM Sample GCon script
CD C:\EStudies\Inputs
RUN C:\AMAZE\METAMESH Diode
RUN C:\AMAZE\HIPHI Diode
MOVE Diode.mou TO
C:\EStudies\Outputs\Diode.mou
```

9.4. Running GCon

GCon is simple to run. Start the program and click on the command *Run GCon script*. The dialog shows all files with the suffix `GCN`. The working directory is changed if you change directories in the dialog. Pick a batch file and click *OK*. The program executes the commands in the file in sequence and displays the current status.

The *Stop on error* command toggles a switch that determines whether the program will stop the batch process is there is an error in one of the batch commands. The command *Instructions* in the *Help* menu displays a help file in your default browser. The function requires that the file `GCON.HTML` is in the same directory as `GON.EXE`.

Chapter 10. MagView - file operations menu

The **MagView** post-processor is designed for the analyses of magnetostatic simulations. The program has the following popup menus: *File operations*, *Analysis*, *Plane plots*, *Slice plots*, *Surface 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 operations* menu.

LOAD SOLUTION FILE (GOU)

The *Load solution file* command displays a dialog (Fig. 10.1) with a list of solution files with names of the form `FPREFIX.GOU`. 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 analysis and plot menus become active.

SOLUTION FILE INFORMATION

The command displays information on the currently-loaded data file (Fig. 10.2).

LOAD COIL FILE (FSC)

You can include applied field coils in urface plots. The *Load coil file* command displays a dialog with a list of current element files with names of the form `FPREFIX.FSC`. Changing the directory in the dialog changes the program working directory. Pick an available file and click *OK*.

CLOSE COIL FILE

This command removes applied field coils from slice and surface plots

COIL FILE INFORMATION

The command displays information on the currently-loaded current element file.

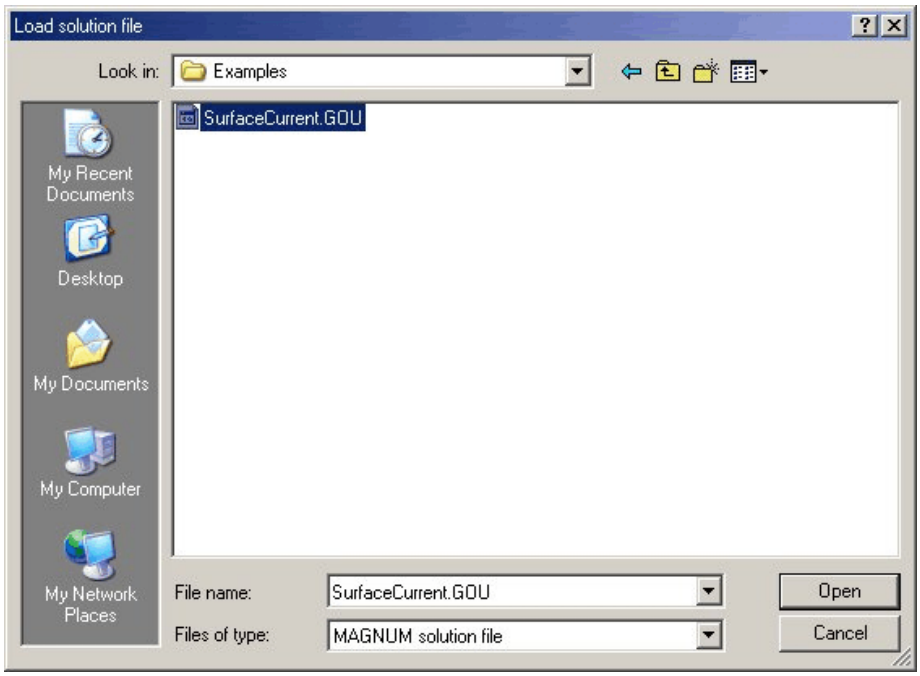

Figure 10.1. Load solution file dialog

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 **MagView**. 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. Section13.4 describes the uniform script command language used in all **AMaze** postprocessors. 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. 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 reference list of allowed commands. 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.

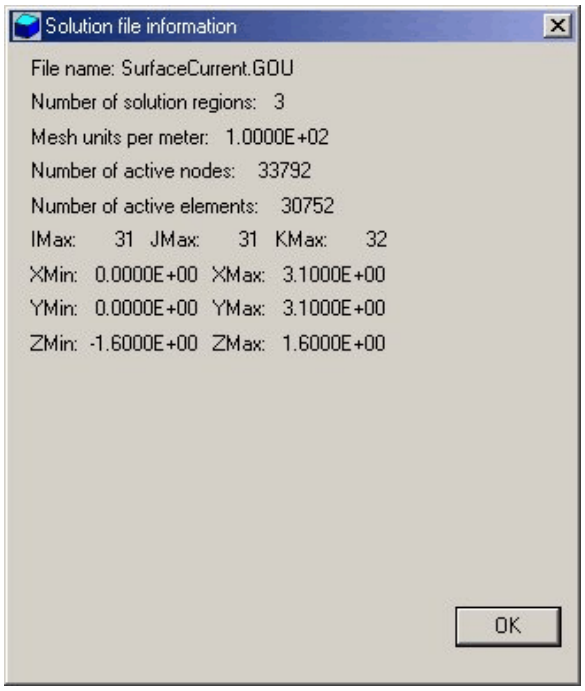

Figure 10.2. Solution file information display

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.

OPEN DATA 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 FILE

Use this command if you want to start a new file to record data. The data file is automatically closed when you exit the postprocessor. Otherwise, you must close the file before using the *Edit data file* command or loading the file into another program. Failure to close the file may result in a Windows Resource Sharing Error.

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.

The *Help* menu shows program information and contains the following command:

INSTRUCTIONS

Displays the HTML help file using your default browser. In order for this command to operate correctly, the file `MAGNUM.HTML` must be in the same directory as `MAGVIEW.EXE`.

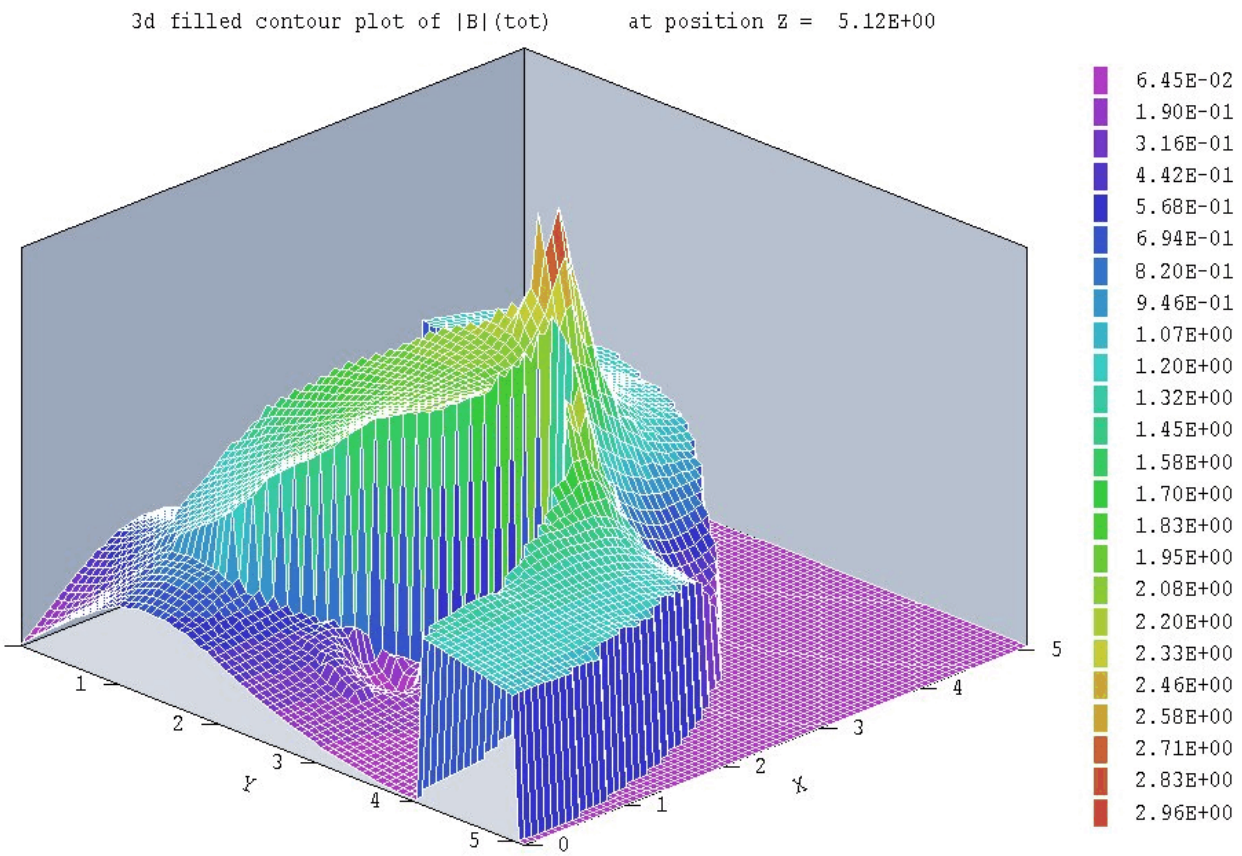

Figure 11.1. Plane plot of |**B**| - type 3D filled contour

Chapter 11. MagView - plane plots

Plane plots are two-dimensional plots that show the variation of quantities over a plane normal to one of the Cartesian axes. Plane plots provide simple and quick views of the solution space. For a plane plot **MagView** creates a rectangular mesh of values over a specified planar region and then generates plots in a variety of styles. No attempt is made to coordinate the plot mesh with the structure of the simulation conformal mesh. Slice plots, discussed in the next chapter, are also two-dimensional in a plane normal to a Cartesian axis. The difference is that slice plots are built using the computational mesh. They show precise region boundaries and field values in true scale. Slice plots require more computational effort than plane plots.

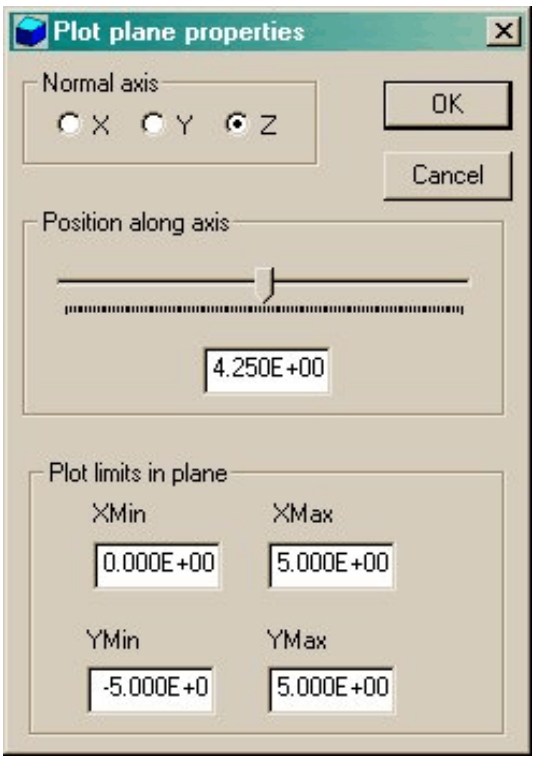

Figure 11.2. Set plane dialog

SET PLANE

This command brings up the dialog of Fig. 11.2 to set the plane for the plot. Set the normal axis with the radio buttons at the top. For example, for a choice of *z* the plot will be created in the *x*-*y* plane. You can use the slider bar to set the position along the normal axis or type a value in the box. The range of the slider bar is automatically set to the limits of the solution volume along the normal axis. The boxes at the bottom determine the plot range in the normal plane. The default settings are the limits of the solution volume. Note that plane plots are constructed to fill the maximum area. They do not preserve scaling in the normal plane.

The following commands are in the *Plot control* popup menu:

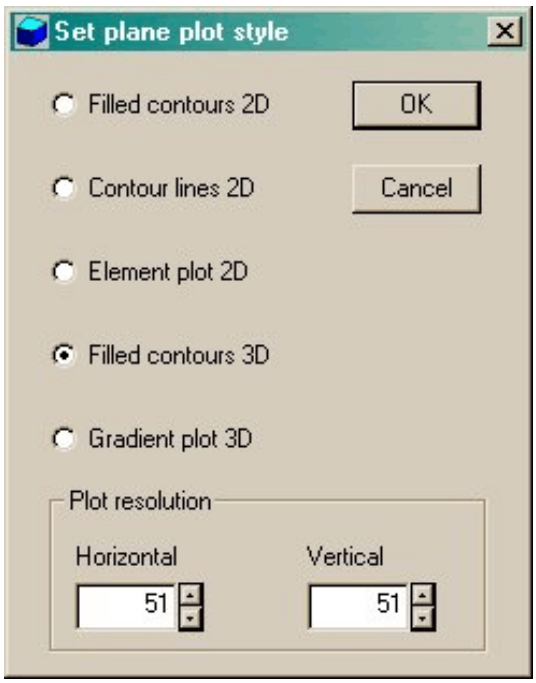

Figure 11.3. Plot style dialog

PLOT STYLE

This command brings up the dialog of Fig. 11.3 to set the plot style. As an example, Fig. 11.1 shows the *Filled contours 3D* plot style. The numbers at the bottom give the resolution of the mesh used to create the plot. Higher values give more detail but require longer regenerate times. The default is a 51 × 51 mesh.

PLOT QUANTITY

Set the quantity to be plotted. The choices are $|\mathbf{B}|$, B_x, B_y, B_z, $|\mathbf{B}_s|$, B_{sx}, B_{sy} or B_{sz}, where the subscript *s* denotes the applied field component. The display of the total magnetic flux density (**B**) and its components depends on the setting of the interpolation mode (see Section 2.3 and Chapter 13). In the *STANDARD* mode, the quantities correspond to the total magnetic flux density calculated using reduced potential values (ϕ) in air and conducting regions and dual potential values (ψ) in iron regions. In the *REDUCED* mode, the program calculates **B** from the reduced potential in all regions. Values of $\mathbf{B}_s$ are always calculated from the reduced potential and do not depend on the interpolation mode.

PLOT LIMITS

Set limits for the plotted quantity. When *Autoscale* is activated, **MagView** automatically sets limits based on the range of values in the solution file.

ROTATE PLOT

This command is active only for the *Filled contours 3D* and *Gradient plot 3D* plot styles. You can rotate the plot in 90° increments for the best view.

The commands of the *Export plot* menu are used to generate hardcopy or to create plot files.

DEFAULT PRINTER

With this command, any **MagView** plot can be ported to any installed Windows printer (including network printers, postscript drivers, PDF drivers...). You can generate colored plots if you have a color printer. Note that the current screen plot is sent to the default Windows printer. If necessary, change the default using the *Settings* command of Windows before issuing the command.

PLOT FILE (EPS)

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). Supply a file prefix in the dialog box. The plot file will be created in the current directory with the name `FPREFIX.PNG`.

COPY TO CLIPBOARD

The current plot is copied to the clipboard in Windows Metafile format. You can then paste the image into compatible graphics software.

With a high-resolution display a screen captures is an easy and effective alternative to the generation of plot files. The freeware utility MWSnap is supplied with **Magnum**. The program PaintShop Pro (http://www.jasc.com) is an inexpensive and versatile option for screen captures and graphical manipulations. We used the program to create the illustrations for this manual.

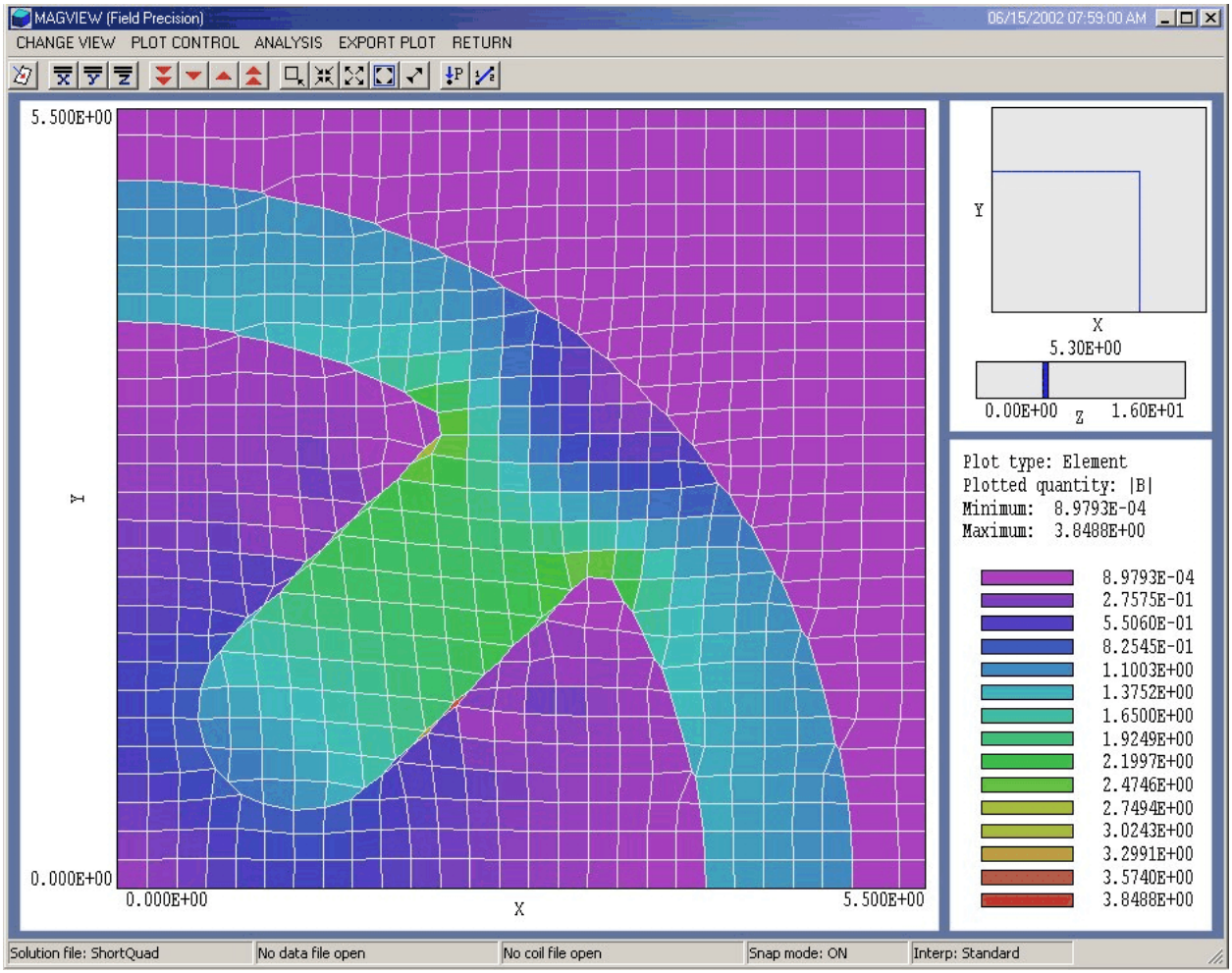

Figure 12.1. **Magnum** working environment in the *Slice plot* menu – illustration of an element plot

Chapter 12. MagView - slice plots

12. 1. Setting the slice view

Slice plots are two-dimensional plots that show the variation of quantities over a plane normal to one of the Cartesian axes. In contrast to plane plots, slice plots are based on the structure of the mesh projected to a slice plane. This projection may be complex for a conformal mesh; therefore, slice plots require more computational effort. To facilitate the process, slices are constructed at discrete locations along the normal axis corresponding roughly to the planes of the foundation mesh. The precise rendering of spatial information allows point-and-click analysis operations (point calculation, line scan, ...) in the slice.

The *Change view* popup menu contains commands to set the slice plane and to adjust the dimensions of the plot.

SET SLICE PLANE

This command calls up the same dialog as the *Set plane* command in the plane plot menu (Fig. 11.2). You can change the normal axis, change the position along the normal axis, and set plot limits in the normal plane.

SLICE NORMAL TO X
SLICE NORMAL TO Y
SLICE NORMAL TO Z

Quick commands to change the normal axis.

JUMP FORWARD
STEP FORWARD
STEP BACKWARD
JUMP BACKWARD

Move along the normal axis by small or large steps. The small step is approximately one layer of the foundation mesh and the large step is 5 layers. The term *forward* implies motion toward higher indices of the normal axis. The slider bar in the orientation area to the right of the plot (Fig. 12.1) shows the current location.

ZOOM WINDOW

As an alternative to the entries in the *Set slice plane* dialog, you can interactively change plot limits in normal plane using the mouse. Choose the command and move the mouse pointer into the plot area. The status bar enters coordinate mode. It shows the current mouse position in the plot and the snap mode (discussed below). Use the left button to pick one corner and then move the mouse to create a view box. Click the left button again, and the plot regenerates. On any coordinate operation, press the *F1* key if want to enter values from the keyboard. Note that the normal plane box in the orientation area to the right of the plot (Fig. 12.1) shows the dimensions of the slice plane and the outline of the current zoomed view.

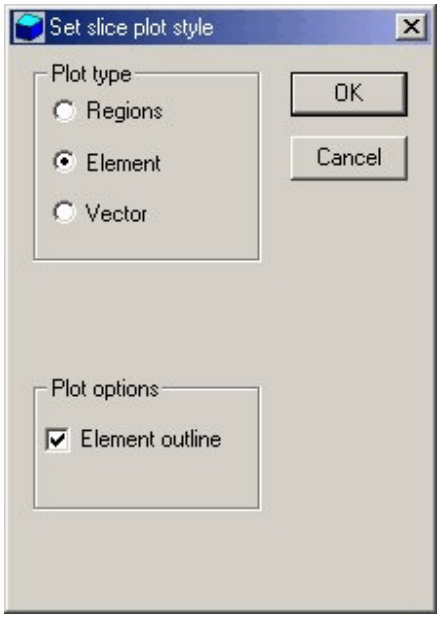

Figure 12.2. Slice plot style dialog

ZOOM IN

Enlarge the plot about the current view center.

EXPAND VIEW

Expand the plot about the current view center.

GLOBAL VIEW

Enlarge the plot boundaries to show the entire normal plane.

PAN

When the plot is zoomed, you can use this operation to shift the current view center. Use the mouse to define a relative starting and ending point for the shift.

12.2. Setting slice plot properties

The commands in the *Plot control* popup menu are used to set the plot style and mouse options.

SET SLICE PLOT STYLE

This command brings up the dialog of Fig. 12.2. The *Region plot* style is a cross-section view of the mesh element divisions color-coded by region. In contrast to the logical plane plot of **MetaMesh**, **MagView** attempts to resolve the exact mesh structure in the plane. In the *Element plot* style the program shows elements color-coded by the value of the present slice plot quantity. The *Vector plot* style shows elements with color-coding according to $|\mathbf{B}|$ or $|\mathbf{B}_s|$ with superimposed vector lines parallel to local direction of $\mathbf{B}$ or $\mathbf{B}_s$ in the plane of the slice (Fig. 12.3). When *Element outline* is active the boundaries of elements are added to the color-coded plot.

PLOT QUANTITY

Use this command to set the quantity for color-coding in element and vector plots. The element plot choices are $|\mathbf{B}|$, B_x, B_y, B_z, $|\mathbf{B}_s|$, B_{sx}, B_{sy} and B_{sz}. For vector plots, a choice of any of the quantities are $|\mathbf{B}|$, B_x, B_y, B_z gives a plot of $|\mathbf{B}|$, while a choice of any of the quantities $|\mathbf{B}_s|$, B_{sx}, B_{sy} and B_{sz} give a plot of $|\mathbf{B}_s|$.

PLOT LIMITS

In the *Autoscale* mode, **MagView** sets the color spectrum so that it spans the range between minimum and maximum values of the current plot quantity in the current slice plane. The values are automatically updated when you change the slice plane or plot quantity. Deactivate the autoscale check box to set the values manually.

VECTOR DENSITY

In vector plots **Magnum** creates vectors on a square array with approximately 35 vectors along the longest axis of the plot. Depending on the resolution of your screen or hardcopy device, you may want to increase or decrease the density of vectors. Enter a number between 10 and 75 in the dialog. To avoid computing vectors in regions of small fields, vectors are plotted only in areas where the field magnitude exceeds the minimum value by 5 percent of the range.

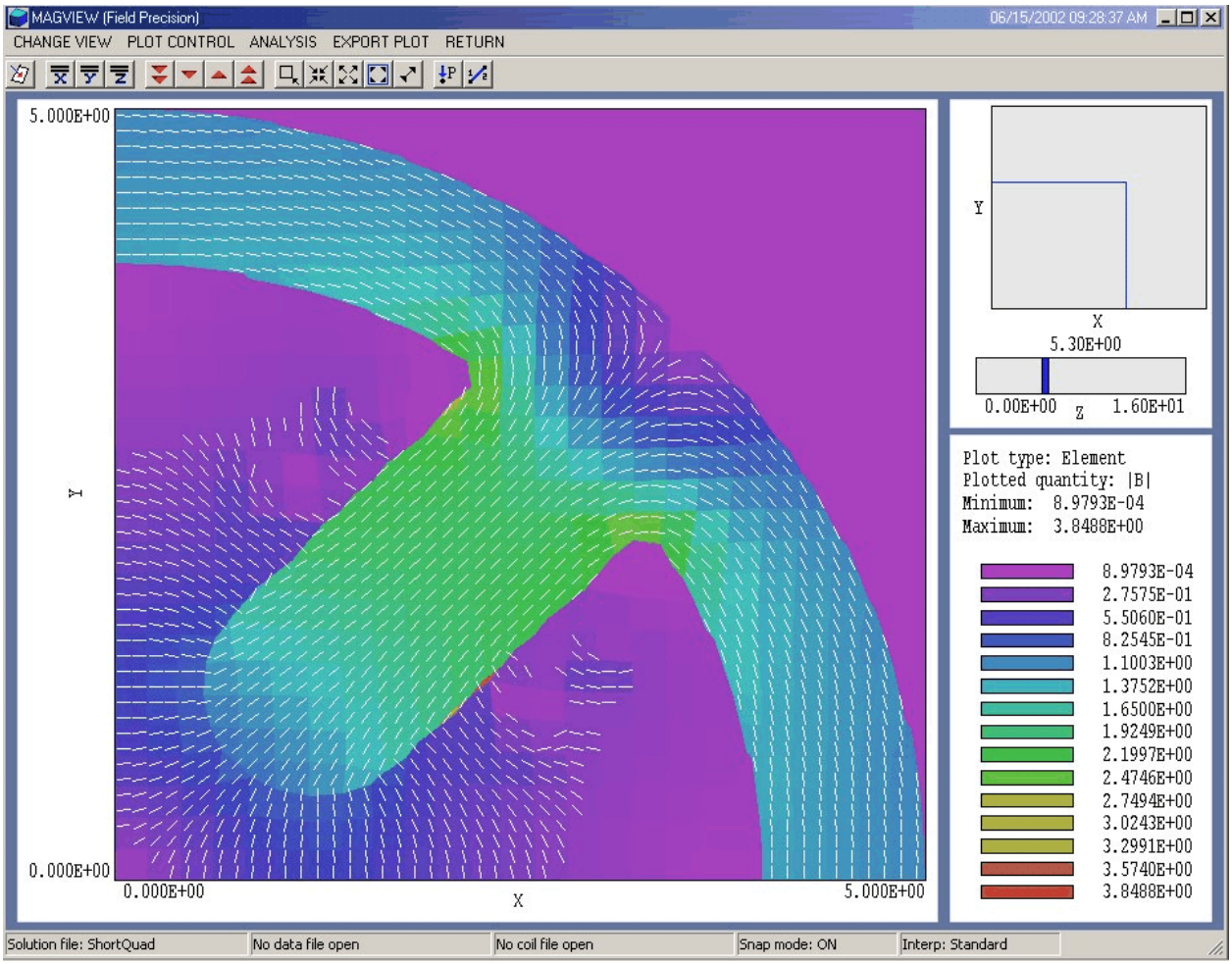

Figure 12.3. **MagView** working environment, vector plot

TOGGLE SNAP MODE

Mouse coordinates for commands such as *Zoom window*, *Pan*, and *Scan in slice* can be entered in two modes. In the normal mode, the returned position corresponds to the mouse position on the screen. In the snap mode, the program picks a point at an even interval close to the mouse position. The returned point depends on the value of the parameter *DSnap*. For example if *DSnap* = 0.1 and the mouse is at position (6.2345,-5.6113), the returned position is (6.2000,-5.6000). The status bar displays the actual or snapped position of the mouse.

SET SNAP DISTANCE

Change the value of *DSnap* from the default value determined by the program.

TOGGLE GRID

A set of dashed grid lines can be superimposed on slice plots. **MagView** automatically chooses intervals and positions so that the lines occur at convenient values along the horizontal and vertical directions (for example, 0.01 rather than 0.01153). Listings of the grid intervals are included in the axis labels. Grids corresponding to the normal plane axes are plotted as solid lines.

12.3. Analyses in a slice

You can determine field values at points and along scan lines with the commands of the *Analysis* popup menu.

POINT CALCULATION

This command is useful for quick checks of field values in the solution volume. After you click the *Point calculation* command, move the mouse into the plot area. The mouse pointer changes to a cross-hair pattern and the status bar enters coordinate mode. Click the left button to specify a point or press the *F1* key to enter the coordinates from the keyboard. Note that the mouse coordinates will shift between discrete values if snap mode is active. **MagView** calculates components of **B** and **H** at the point in the normal plane given by the coordinates. Chapter 13 describes the interpolation method. Figure 12.4 shows the screen information display. The results are recorded if a data file is open. Table 12.1 shows an example of listing file output for a point calculation.

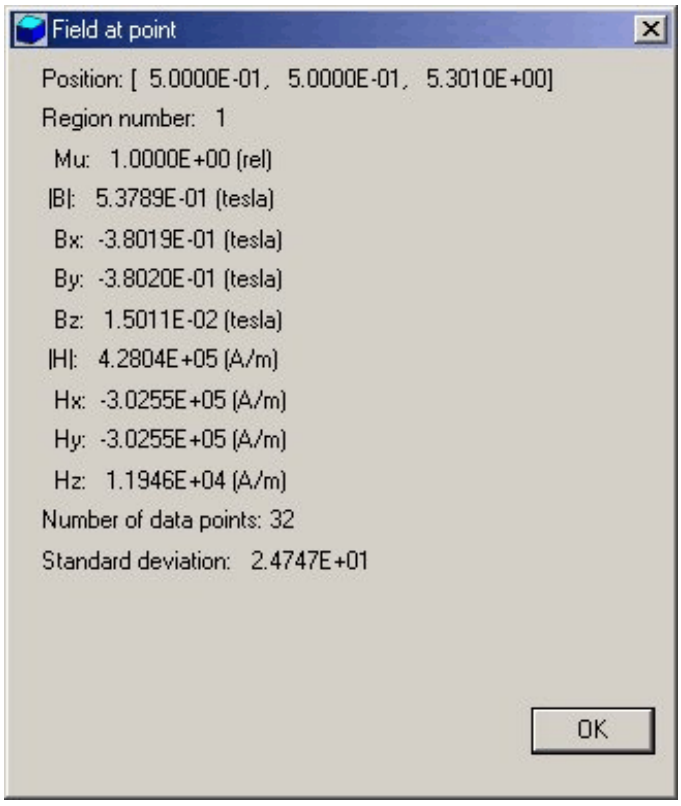

Figure 12.4. Screen listing for point calculation

Table 12.1. Data file listing for a point calculation

```
  --- Point Field Analysis ---
Position: [  5.0000E-01,   5.0000E-01,   5.3010E+00]
Region number:   1
  Mu:   1.0000E+00 (rel)
 |B|:   5.3789E-01 (tesla)
  Bx:  -3.8019E-01 (tesla)
  By:  -3.8020E-01 (tesla)
  Bz:   1.5011E-02 (tesla)
|Bs|:   5.3789E-01 (tesla)
 Bsx:  -3.8019E-01 (tesla)
 Bsy:  -3.8020E-01 (tesla)
 Bsz:   1.5011E-02 (tesla)
Number of data points: 32
Standard deviation:   2.4747E+01
```

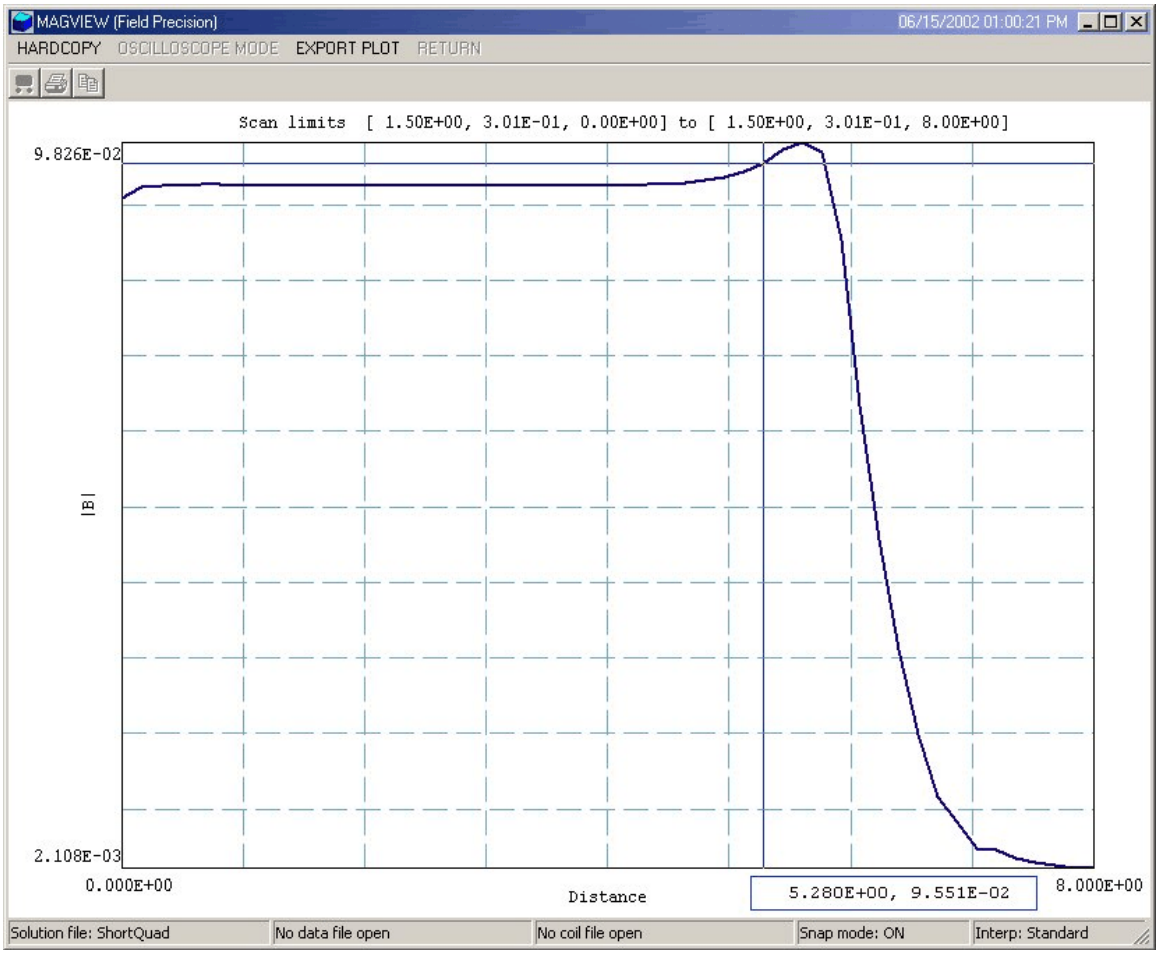

Figure 12.5. Line scan plot in the oscilloscope mode

LINE SCAN

Line scans are one of the most useful **MagView** capabilities. After clicking on the command, supply two points with the mouse to define a scan line (or press the *F1* key to enter coordinates manually). The snap mode is useful in this application (for example, you may want the scan to extend from 0.000 to 5.000 rather than 0.067 to 4.985.) The program calculates a series of values of field quantities in the normal plane at equal intervals along the line. Complete information is recorded if a data file is open. The program also makes a screen plot of the currently-selected quantity versus distance along the scan and activates the *Scan plot* menu (Fig. 12.5). **MagView** 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 pick the quantity that will be displayed in screen and exported plots of line scans. Pick the quantity from the list box and click *OK*. Available quantities are $|\mathbf{B}|$, B_x, B_y, B_z, $|\mathbf{B}_s|$, B_{sx}, B_{sy} and B_{sz}. This setting has no effect on the data file listing which includes all field quantities.

SET NUMBER OF SCAN POINTS

This command sets the number of line scan points in the screen plot and data file listing. The default value is 50 and the maximum number is 250.

In addition to the standard *Export plot* options, the *Scan plot* menu contains the following command:

OSCILLOSCOPE MODE

In oscilloscope mode, a scan plot assumes characteristics of a digital oscilloscope. **MagView** 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 records information when a data file is open. Press the right mouse button to exit the oscilloscope mode.

Click *Return* to exit the scan plot and return to the slice plot.

Table 12.2. Data listing from a line scan

```
 ------- Field scan between points -------
    XStart:   1.5000E+00   YStart:   3.0111E-01   ZStart:   0.0000E+00
      XEnd:   1.5000E+00     YEnd:   3.0111E-01     ZEnd:   8.0000E+00

       X           Y           Z          |B|         Bx          By          Bz
=============================================================================================
  1.5000E+00  3.0111E-01  0.0000E+00  6.9394E-01  9.0825E-02 -6.8797E-01 -1.6390E-03
  1.5000E+00  3.0111E-01  1.6000E-01  6.9275E-01  9.2313E-02 -6.8658E-01  8.9894E-04
  1.5000E+00  3.0111E-01  3.2000E-01  6.9283E-01  9.2423E-02 -6.8664E-01  4.4154E-04
  1.5000E+00  3.0111E-01  4.8000E-01  6.9285E-01  9.2581E-02 -6.8663E-01 -1.1986E-04
  1.5000E+00  3.0111E-01  6.4000E-01  6.9286E-01  9.2658E-02 -6.8664E-01 -7.2919E-04
  1.5000E+00  3.0111E-01  8.0000E-01  6.9294E-01  9.2609E-02 -6.8672E-01  1.5788E-03


                 |H|         Hx          Hy          Hz          MuR      NReg
               ===========================================================================
                 5.5222E+05  7.2276E+04 -5.4747E+05 -1.3043E+03  1.0000E+00    1
                 5.5128E+05  7.3460E+04 -5.4636E+05  7.1535E+02  1.0000E+00    1
                 5.5134E+05  7.3548E+04 -5.4641E+05  3.5136E+02  1.0000E+00    1
                 5.5135E+05  7.3674E+04 -5.4640E+05 -9.5378E+01  1.0000E+00    1
                 5.5136E+05  7.3735E+04 -5.4641E+05 -5.8027E+02  1.0000E+00    1
                 5.5142E+05  7.3696E+04 -5.4647E+05  1.2564E+03  1.0000E+00    1
```

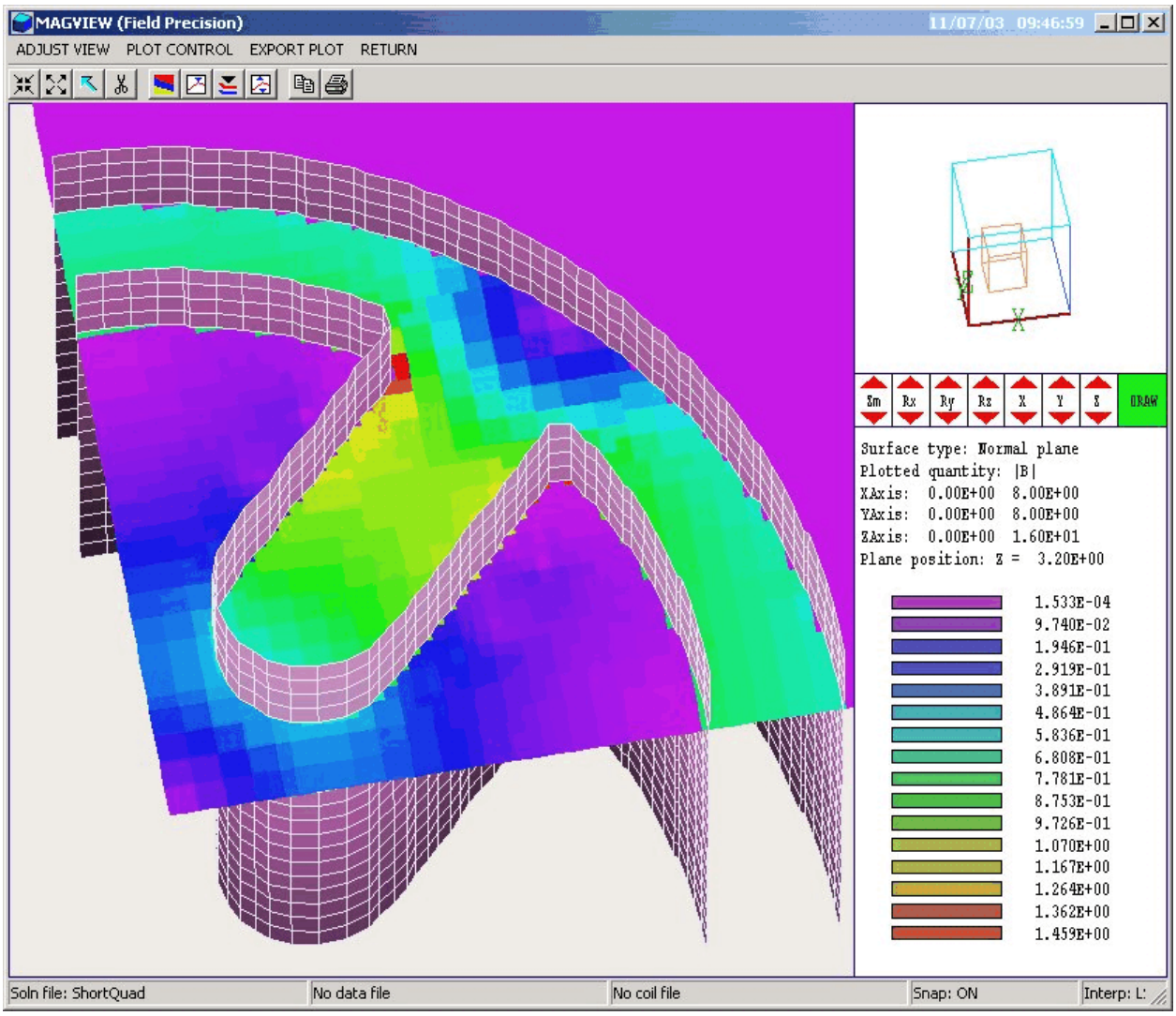

Figure 13.1. MagView working environment for surface plots.

Chapter 13. MagView - surface plots

Surface plots (Fig. 13.1) are three-dimensional views of the solution space. Three types of information can be superimposed: 1) boundaries of regions, 2) computed quantities in a slice plane normal to one of the Cartesian axes and 3) applied field coils. Applied field coils are plotted whenever a current element file is open (Chapter 9). Surface plots are created from the conformal mesh and preserve true spatial scaling.

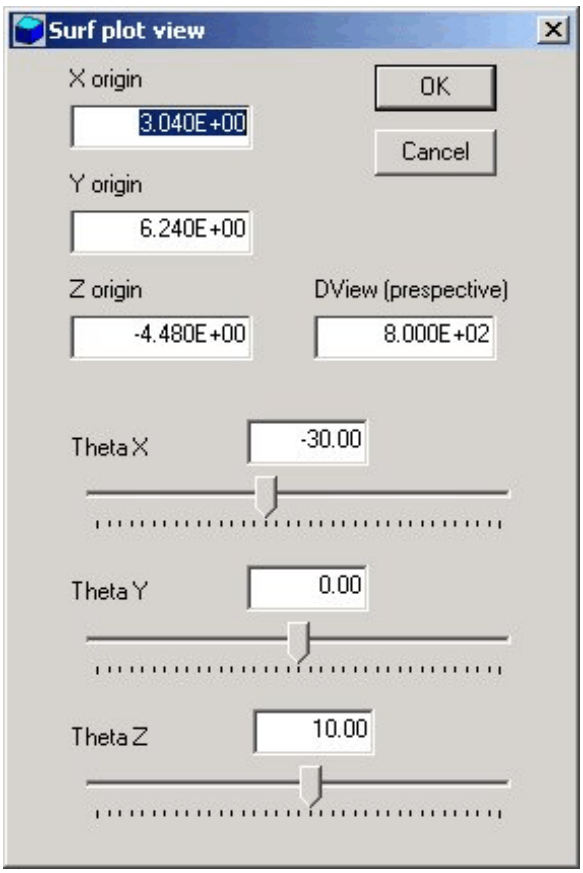

Figure 13.2. Surface view dialog.

The three-dimensional display of the current plot style is controlled through the control bar described in Sect. 4.3 (center-right section of the screen in Fig. 13.1.). Alternatively, you can use the *Set surface view* command in the *Adjust view* menu (Fig. 13.2).

The commands of the *Plot control* popup menu control the appearance of the plot.

SURFACE PLOT STYLE

This command brings up the dialog of Fig. 13.3 that controls the style of three-dimensional plots. The first group sets plot type. The option *Regions only* displays geometric information on chosen material boundaries and applied field coils if a current element file is loaded. The *Normal plane* options superimposes a plot of color-coded field information on the geometric plot (Fig. 13.1). The normal plane plot is

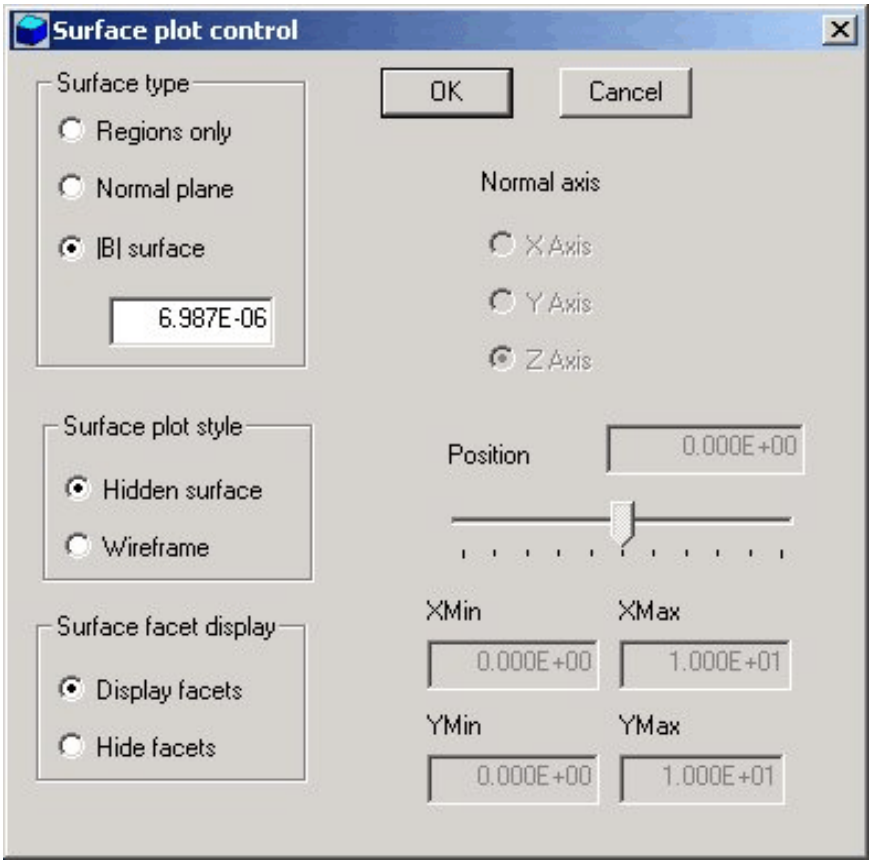

Figure 13.3. Surface plot style dialog.

the three-dimensional equivalent of the plane plots discussed in Chapter 11 except that it preserves true spatial scaling. For normal plane plots the commands to choose the plane on the right-hand side of the dialog become active. You can set the normal axis, pick the plane position along the axis, or set the transverse plane boundaries.

The third option is a three-dimensional surface with a fixed value of |**B**|. Enter the value in the box below the button. The default value is the average of |B| over the solution volume. The |B| surface plot should be used with caution. The reason is that |B| may be a discontinuous function in solutions where $\mu r \ll 1$ or $\mu \gg 1$. In situations where the surface is not well-defined, the complex process of surface construction may generate errors that crash the program. Note: if the program generates a surface with missing facets, rotate the view slightly.

The radio buttons *Hidden surface* and *Wireframe* control whether the facets of region boundary plots are filled. The wireframe view regenerates faster but may difficult to understand. You can use the wireframe mode to set up a three-dimensional view and then switch to the hidden-surface mode to create the final plot. The radio buttons *Display facets* or *Hide facets* controls whether the edges of facets are shown on region boundaries.

DISPLAYED REGIONS

The command brings up a dialog where you can pick region boundaries to superimpose on a plot. The default is that no regions are displayed.

SURFACE/FIELD CUT PLANES

In a hidden surface plot, internal geometric details or the normal plane plot may be obscured by surrounding region boundaries. This command brings up a dialog that allows you to adjust the displayed portions of region surfaces and normal planes along the *x*, *y* and *z* axes. **MagView** does not display facets that lie outside the limits. With this feature you can create cutaway views. The default is that cut limits are set equal to the dimensions of the solution volume so that all facets are included.

COIL CUT PLANES

Use this command to limit plotted current elements when coils are displayed.

SET PLOTTED QUANTITY

With this command, you can change the quantity used for color-coding in the normal plane plot. The options are $|\mathbf{B}|$, B_x, B_y, B_z, $|\mathbf{B}_s|$, B_{sx}, B_{sy}, B_{sz}, $|\mathbf{B}_m|$, B_{mx}, B_{my} and B_{mz}. The subscript s denotes applied field components and m denotes field components arising from materials. The default quantity is $|\mathbf{B}|$.

SET NORMAL PLOT LIMITS

In the *Autoscale* mode, **MagView** chooses colors so that the spectrum spans the range between the minimum and maximum values of the plotted quantity in the current plane. Scaling values change automatically if you change plot quantities or move to a different plane. To set the range manually, deactivate the autoscale check box and enter minimum and maximum values.

Chapter 14. Analysis

The commands of the *Analysis* popup menu are used to determine numerical values for the solution. In the interactive mode, the menu is active only when a data file is open.

14.1. Point calculation interpolation method

The *Point calculation* command brings up a dialog where you enter the *x*, *y* and *z* coordinates of a general point in the solution space. In contrast to the *Point calculation* command of the *Slice plots* menu, the point is not limited to a slice plane. When you click *OK*, **MagView** displays the information box of Fig. 12.4 and also writes the results to the data file in the format shown in Table 12.1.

MagView must perform several operations to make the calculation:

- Estimate the indices of the element that contains the target point.
- Check this element and its neighbors using a normal coordinate inversion to determine the precise element that contains the target point.
- Collect the nodes of the target element as well as neighboring nodes that border on at least one element with the same region number as the target element
- Fit three-dimensional, second-order functions to the reduced potential ϕ, the applied field values $\mathbf{H}_s$ and possibly the dual potential ψ using a least-squares method.
- Find values of **B** and **H** using applied field values and derivatives of the potential functions.

Although the procedure sounds complicated, **MagView** performs it with high reliability. The second-order least-squares method gives magnetic field estimates that are substantially better than those derived from the direct use of the quasi-linear finite-element form functions. Furthermore, the requirement that the included nodes must be connected to elements of the target region ensures that data points will be collected from one side of a material boundary. Therefore, **MagView** correctly handles the discontinuity in magnetic field at the boundaries of ferromagnetic materials.

14.2. Line scans and matrix files

In the *Analysis* menu scans can be performed along arbitrary straight lines between any two points in the solution volume. When you click the *Line scan* command a dialog appears where you can specify the start and end points. Enter values in units set by *DUnit* (in other words, if you used dimensions of cm in **MetaMesh** and the **Magnum** script contains the command *DUnit* = 100.0, then enter the dimensions in cm). In the interactive mode, **MagView** creates a screen plot of a selected quantity determined by the *Set scan quantity* command. The command has no effect on the data file listing which contains all calculated quantities. Table 12.2 shows an example of a listing.

The *Create matrix file* command controls a **MagView** feature that is useful if you want to write your own analysis routines or port results to mathematical software. In response to the command, **MagView** performs interpolations over a specified box region on a regular grid of coordinates. It is much easier to use results in this form than to deal directly with the conformal mesh. Clicking on *Create matrix file* calls up the dialog of Fig. 14.1. Specify the dimensions of the box along each axis (in units set by *DUnit*) and the number of calculation intervals. To illustrate, calculations are performed at positions with x coordinates given by

$$x = x_{\min} + \frac{n\,(x_{\max} - x_{\min})}{n_x}\,, \qquad (13.1)$$

where n = 0, 1, 2, ... , n_x. For example, if you set x_{min} = 0.5, x_{max} = 1.5 and n_x = 10, the calculations are performed at points with x = 0.5, 0.6, ..., 1.4, 1.5. You can also specify an output file name. The text file is created in the standard AMAze Field Exchange format. The stored quantities are B_x, B_y, B_z, and μ. You can inspect the file with an editor.

14.3. Automatic analyses

In response to the command *Full analysis*, **MagView** performs a volume integral of magnetostatic field energy U_m over all elements of the mesh. **MagView** organizes energy by region and writes the results to the data file in the format shown in Table 14.1. The program will also compute forces on applied field coils if a current element file is open. The calculations are intensive, so there may be a delay for large meshes.

You can use the global energy value to determine the self-inductance of a single applied field coil from the equation

$$U_m = \frac{L\, I^2}{2} \,. \qquad (13.1)$$

In response to the *Region* command, **MagView** performs a volume integral of magnetostatic energy and surface integrals of magnetic flux over a specific region. Surface integral values are given for the total region boundary and also broken down by boundaries with other regions. Depending on how you set up the region geometry, the magnetic flux integrals may be useful for estimating mutual inductances.

14.4. Script operation

You can create a script that will automatically call the commands of the *Analysis* menu. **MagView** scripts have names of the form `SCRIPTNAME.SCR`. You can initiate an automatic analysis using the *Run script* command in the interactive mode or starting **MagView** from a batch file with a command of the form:

```
START \AMAZE\MagView SCRIPTNAME
```

We discussed the following interactive script commands in Chapter 9: *Run script*, *Create script* and *Edit script*. The *Create script* command generates a template file with the content shown in Table 14.1. In this section, we shall discussed the allowed commands in detail.

Table 14.1. Create script - default content

```
* AMaze script file

* Insert commands here...

ENDFILE


      --- Script command summary ----

INPUT FileName
 [Close current solution file and load FileName]
OUTPUT FPrefix
 [Close current data file and open FPrefix.DAT]
NSCAN  100
 [Set the number of points in a line scan]
POINT xp yp zp
 [Point field calculation at the given coordinates]
LINE xp1 yp1 zp1 xp2 yp2 zp2
 [Scan along a line between the given coordinates]
FULLANALYSIS
 [Write analyses for all region to the data file]
REGION RegNo
 [Write an analysis for region RegNo to the data file]
MATRIX FileName XMin XMax NX YMin YMax NY ZMin ZMax NZ
 [Write a matrix of field values to the file FileName]
ENDFILE
 [Terminate the analysis]
```

Symbolic command: `INPUT FPrefix.GOU`
Example: `INPUT WGUIDE1.GOU`
Function: Close the current solution file and load a binary solution file from the current directory
Comments: The solution file must be in the same directory as the script

Symbolic command: `OUTPUT FPrefix`
Example: `OUTPUT WGUIDE1`
Function: Close the current data file and open a new data file.
Comments: The data file has a name of the form `FPrefix.DAT`.

Symbolic command: NSCAN NScan
Example: NSCAN 100
Function: Set the number of intervals for line scans
Comments: The default value is *NScan* = 50, the maximum value is *NScan* = 250.

Symbolic command: POINT xp yp zp
Example: POINT 0.00 0.05 4.67
Function: Perform a point field calculation and write the result to the data file. Enter coordinates in the units set by *DUnit*.

Symbolic command: LINE xp1 yp1 zp1 xp2 yp2 zp2
Example: LINE 0.00 0.00 15.00 12.00 0.00 15.00
Function: Perform (*NScan*+1) calculations along a line in space and write the results to the data file. Enter coordinates in units set by *DUnit*.

Command: FULLANALYSIS
Function: Write volume integrals for the full solution space to the data file.

Symbolic command: REGION RegNo
Example: REGION 7
Function: Write volume and surface integrals for region *RegNo* to the data file.

Symbolic command: MATRIX FileName XMin XMax NX YMin YMax NY ZMin ZMax NZ
Example: MATRIX WGUIDE.MAT 1.00 1.00 10.00 2.00 2.00 12.00
Function: Write solution values computed field values at an array of location to a data file in text format. The file is named *FileName* and is created in the current directory.

Command: ENDFILE
Function: Terminate the analysis

Chapter 15. Structure of Magnum files

15.1. Current-element file (FSC)

The FSC output file from **CPrep** is in text format and is largely self-documenting. Table 15.1 shows a partial listing of an element file consisting of *a*) header information, *b*) coil parameters, and *c*) element properties. The file has the following components:

1) Program title line
2) *Ds*, element resolution
3) *DUnit*, length unit conversion
4) *Frequency*, RF frequency
5) *NCoil*, number of coils
6) *NElem*, number of elements
7) Separator and title for coil information (4 lines)
8) Coil number, current amplitude, current phase (*NCoil* data lines)
9) Separator and title for element information (4 lines)
10) Coil number, *XStart*, *YStart*, *ZStart*, *XEnd*, *YEnd*, *ZEnd*, *IMag* (*NElem* data lines)

Note that the quantities *Frequency* and the phase of coil currents are ignored in the static **Magnum** code. These quantities are included for compatibility with the Field Precision **FSC** code which can calculate radiative fields from specified currents.

15.2. Magnum output file

Magnum creates output files in binary or text format. The mode is controlled by the FORMAT command (Sect. 3.2). The BINARY mode must be used to communicate results to MagView. The text mode is useful to transfer results to your own analysis programs.

As in MetaMesh, nodes are referenced with the indices [*I, J, K*] where *I* (the index along the *x* axis) extends from 0 to I_{max}, *J* (*y* axis) from 0 to J_{max}, and *K* (*z* axis) from 0 to K_{max}. The number of elements is approximately equal to the number of nodes. A single element (in the direction of positive *x, y* and *z*) is associated with each node for storage. Nodes and elements have integer *region numbers* to associate them with structures in the solution space.

Table 15.1. FSC file illustration

```
FSC Current Element File (Field Precision, Albuquerque NM)
Ds:   5.0000E-03
DUnit:   1.0000E+02
Frequency:   0.0000E+00
NCoil:   8
NElem:    960

Coil     I         Phase
 No     (A)        (deg)
==========================
  1  5.0000E+02  0.0000E+00
  2  5.0000E+02  0.0000E+00
  3  5.0000E+02  0.0000E+00
  4  5.0000E+02  0.0000E+00
  5 -5.0000E+02  0.0000E+00
  6 -5.0000E+02  0.0000E+00
  7 -5.0000E+02  0.0000E+00
  8 -5.0000E+02  0.0000E+00

Coil   XStart      YStart      ZStart       XEnd        YEnd        ZEnd        IMag
 No     (m)         (m)         (m)         (m)         (m)         (m)         (A)
================================================================================================
  1  4.0000E-02  1.5000E-02 -3.0000E-01  4.0000E-02  1.5000E-02 -2.9500E-01  5.0000E+02
  1  4.0000E-02  1.5000E-02 -2.9500E-01  4.0000E-02  1.5000E-02 -2.9000E-01  5.0000E+02
  1  4.0000E-02  1.5000E-02 -2.9000E-01  4.0000E-02  1.5000E-02 -2.8500E-01  5.0000E+02
  1  4.0000E-02  1.5000E-02 -2.8500E-01  4.0000E-02  1.5000E-02 -2.8000E-01  5.0000E+02
  1  4.0000E-02  1.5000E-02 -2.8000E-01  4.0000E-02  1.5000E-02 -2.7500E-01  5.0000E+02
  1  4.0000E-02  1.5000E-02 -2.7500E-01  4.0000E-02  1.5000E-02 -2.7000E-01  5.0000E+02
  1  4.0000E-02  1.5000E-02 -2.7000E-01  4.0000E-02  1.5000E-02 -2.6500E-01  5.0000E+02
```

The format of the **Magnum** binary output file is simple and compact, making it easy to transfer information to other programs. The code extract shown in Table 15.2. comprises the entire output algorithm. In the header, the quantities I_{max}. J_{max} and K_{max} are 4-byte integers. *DUnit* is a 4-byte real, and *NCQuant* is a 1-byte integer. Here *NCQuant* is the number of stored quantities for the solution program. In **Magnum**, *NCQuant* = 5 and the recorded node quantities are the reduced potential (ϕ), the components of the applied magnetic field intensity (H_{sx}, H_{sy} and H_{sz}) and the dual potential (ψ). **Magnum** then records the names of the five quantities as strings of length 12. This information is used to create labels in **MagView**.

The next step is to record information for each node. The quantities *RegNo* (the region number of the node) and *RegUp* (the region number of the adjacent element in the direction of increasing *I*, *J* and *K*) are 1-byte integers. The coordinates (*X*,*Y* and *Z*) and the stored quantities are 4-byte real numbers. The final step is to record region information. The quantity *NRQuant*, the number of physical quantities stored per region, is equal to 2 in **Magnum**. The quantities are the relative magnetic permeability μ_r for standard materials and the value of the reduced and dual potentials for

Table 15.2. Code to create the Magnum binary output file

```
! --- Header information
WRITE (OutField) IMax,JMax,KMax,DUnit,NCQuant
  DO N=1,NCQuant
    WRITE (OutField) CQuantName(N)
  END DO
! --- Node and element properties
DO K=0,KMax
  DO J=0,JMax
    DO I=0,IMax
      MC = M(I,J,K)
      WRITE (OutField) &
        C(MC).RegNo,C(MC).RegUp,C(MC).x,C(MC).y,C(MC).z, &
        C(MC).Phi,C(MC).H.x,C(MC).H.y,C(MC).H.z,C(MC).Psi
    END DO
  END DO
END DO
! Region properties
WRITE (OutField) NRQuant
DO N=1,NRQuant
  WRITE (OutField) RQuantName(N)
END DO
WRITE (OutField) NRegMax
DO N=1,NRegMax
  IF (Reg(N).Fixed) THEN
    NFix = 1
  ELSE
    NFix = 0
  ENDIF
  WRITE (OutField) NFix,Reg(N).Mu,Reg(N).Pot
END DO
```

fixed regions such as symmetry boundaries. The region quantity names (RQuantName) are strings of length 12. The variable NFix is a 1-byte integer and the region quantities are 4-byte real numbers.

Table 15.3 shows the code to generate the text form of the Magnum output file, while Table 15.4 extracts from and output file.

Table 15.3. Code to create the Magnum text file

```
! Header information
WRITE (OutField,5000)
WRITE (OutField,5100) OutFieldName,IMax,JMax,KMax,DUnit,NCQuant
WRITE (OutField,5200) (CQuantName(N),N=1,NCQuant)
! Node and element quantities
DO K=0,KMax
  DO J=0,JMax
    DO I=0,IMax
      MC = M(I,J,K)
      WRITE (OutField,5300) &
       C(MC).RegNo,C(MC).RegUp,C(MC).x,C(MC).y,C(MC).z, &
       C(MC).Phi,C(MC).H.x,C(MC).H.y,C(MC).H.z,C(MC).Psi
    END DO
  END DO
END DO
! Region properties
WRITE (OutField,5400) (RQuantName(N),N=1,NRQuant)
DO N=1,NRegMax
  IF (Reg(N).Fixed) THEN
    NFix = 1
  ELSE
    NFix = 0
  ENDIF
  WRITE (OutField,5500) N,NFix,Reg(N).Mu,Reg(N).Pot
END DO
```

15.3. MagView matrix file

The **MagView** program can create text files in the standard AMaze Field Exchange format. The file information can be ported to **OmniTrak** or to your own applications. A sample is shown in Table 15.5. Any number of comment lines starting with "*" (asterisk) can proceed the header. The header consists of two data lines:

```
IMax  JMax  KMax
XMin YMin ZMin XMax YMax ZMax
```

The header is followed by (*IMax*+1)(*JMax*+1)(*KMax*+1) data lines that contain the information

```
Bx  By  Bz  MuR  NReg
```

Data lines are recorded in the order

```
DO K=0,KMax
  DO J=0,JMax
    DO I=0,IMax
      ...
    END DO
  END DO
END DO
```

The field value $B_x(I,J,K)$ is calculated at the position:

```
X(I) = XMin + I*Dx,
Y(J) = XMin + I*Dy,
Z(K) = XMin + I*Dz.
```

where

```
Dx = (XMax-XMin)/IMax,
Dy = (YMax-YMin)/JMax,
Dz = (ZMax-ZMin)/KMax.
```

Table 15.4. Extract from a Magnum text output file

```
AMaze series, Program: Magnum

  File: CMagShort.GOU
  IMax:    112
  JMax:    122
  KMax:      4
  DUnit:   1.0000E+02
  NCQuant:   5

 RegNo  RegUp        X           Y           Z          PhiM
    1      1   -3.5000E-02 -7.0000E-02 -5.0000E-03 -9.3749E+02
    1      1   -3.2500E-02 -7.0000E-02 -5.0000E-03 -9.3544E+02
    1      1   -3.0001E-02 -7.0000E-02 -5.0000E-03 -9.3364E+02

                    Hx          Hy          Hz          PsiM
                8.6877E+02 -3.2528E+02  0.0000E+00 -9.3749E+02
                9.0630E+02 -3.1199E+02  0.0000E+00 -9.3544E+02
                9.4520E+02 -2.9643E+02  0.0000E+00 -9.3364E+02
...

 RegNo  Fixed      MuRel       MagPot
    1      0    1.0000E+00  0.0000E+00
    2      0    5.0000E+02  0.0000E+00
```

Table 15.5. Segment of a MagView matrix file

```
* Magnum matrix file: OMNIDEMOM.DAT
      20      20      20
 -2.5000E+00 -2.5000E+00 -1.0000E+00  2.5000E+00  2.5000E+00  6.0000E+00
  8.5954E-03  8.5955E-03  3.2406E-02  1.0000E+00    1
  7.5006E-03  8.3316E-03  3.1949E-02  1.0000E+00    1
  6.4840E-03  8.1033E-03  3.1586E-02  1.0000E+00    1
  5.5393E-03  7.9124E-03  3.1298E-02  1.0000E+00    1
  4.6535E-03  7.7539E-03  3.1071E-02  1.0000E+00    1
  3.8140E-03  7.6264E-03  3.0889E-02  1.0000E+00    1
    ...
```