BStat

Finite-element Magnetostatics

Field Precision

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

BStat 5.0 Manual - Contents

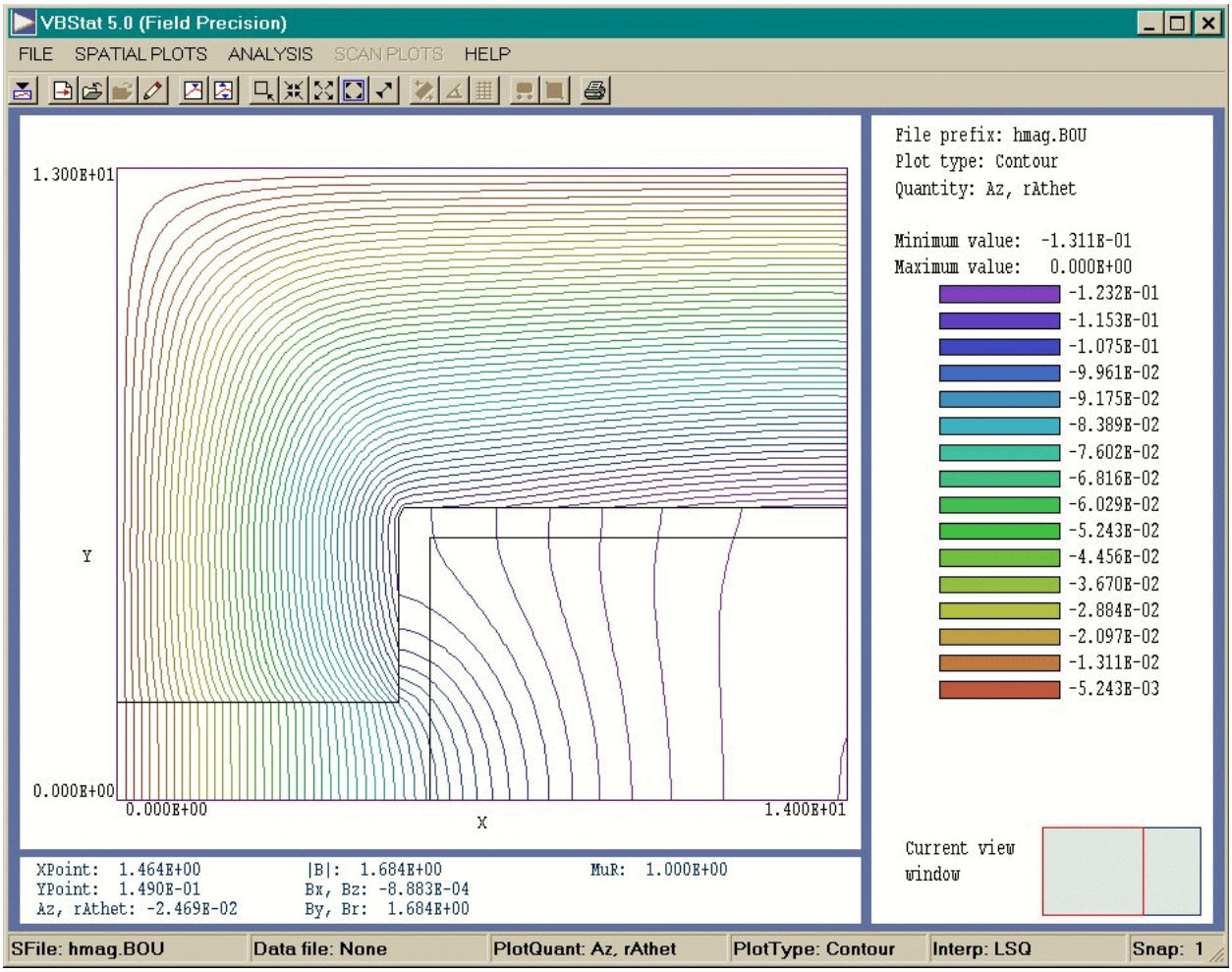

Figure 1. **VBStat** screen display

1. Program function

BStat is a versatile numerical tool for magnet design. The program calculates magnetostatic fields in complex geometries with coils and linear or non-linear materials. **BStat** applies finite-element methods on a variable-resolution conformal triangular mesh. The mesh size is limited only by the installed memory of the computer. The program handles three-dimensional cylindrical problems (azimuthal symmetry) and two-dimensional rectangular problems (variation in *x* and *y* with infinite extent in *z*). Using a simple command language, you can define up to 127 regions to represent materials or coils with different current densities. An interactive graphical analysis program provides full information on the fields and magnetic permeability. Other features include automatic calculation of stored energy, peak field values, magnetic forces and torques. The program makes a wide variety of screen and hardcopy plots.

The walkthrough example in the following section gives an overview of the solution process. Section 3 outlines typical steps in a **BStat** run while Section 4 covers the structure of the command script that controls the program. Sections 5, 6, 7 and 8 give detailed descriptions of script commands to set control parameters and material properties. Section 9 reviews the magnetic field solution technique and the meaning of vector potential boundary conditions. Section 10 covers operation of **BStat** as an interactive Windows program that you can run from the **TC** program launcher. Section 11 explains how to run **BStat** in the background from the Command Prompt or under the control of the **GCon** utility for extended data runs. Sections 12 through 16 address operation of the **VBStat** post-processor used to create plots and to perform quantitative analysis. Finally, Section 17 documents the format of the **BStat** output file.

2. Walkthough example

We shall consider the *HMag* example included in the library of examples to illustrate a complete magnetostatic solution. The simulation illustrates many useful features of **BStat** such as assignment of symmetry boundaries and the ability to generate detailed solutions within a macroscopic solution volume. The *H magnet* configuration is often used in particle accelerators to create dipole fields. The length of the magnet is usually much larger than the transverse dimensions so the two-dimensional planar approximation is valid in regions removed from the ends. Figure 2 shows the simulation geometry, which represents one-quarter of the magnet cross section. By symmetry lines of magnetic flux density are vertical at the magnet midplane. Therefore, we can eliminate the left-hand portion of the magnet and apply a Dirichlet boundary condition (Section 9) along the vertical midplane (left-hand boundary in Fig. 2). Similarly, lines of magnetic flux density **B** are normal at the horizontal midplane; therefore, we can eliminate the bottom half of the magnet and apply a Neumann condition along the bottom boundary. A Dirichlet condition is also applied along the top and right-hand boundaries. The condition is equivalent to the assumption that all magnetic flux is contained within the iron core. In the case of a highly-saturated core it may be necessary to include an air volume around the outside of the solution volume for the correct representation of leakage flux.

In Fig. 2 the dark-blue area (Region 1) is the air gap, the dark-green area is one-quarter of the iron core cross-section, and the light-green area is one-half of the right-hand side of the magnet winding. The current in the coil segment is directed in to the page. The Dirichlet symmetry condition

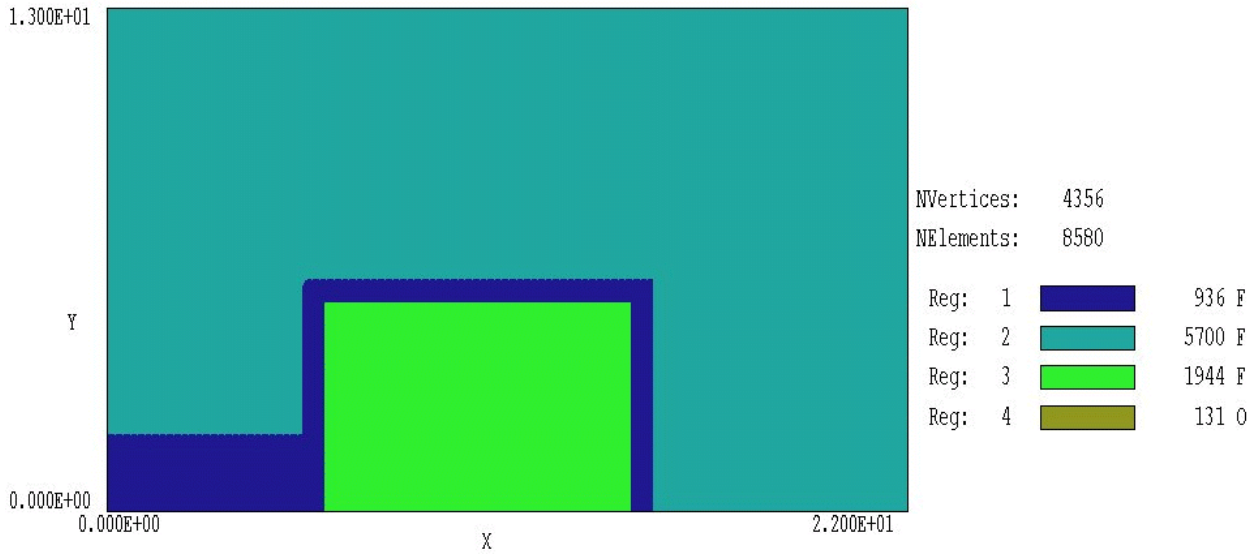

Figure 2. Solution volume and region assignments for the `hmag` example

means that there is a symmetric virtual coil on the left-hand side that carries current out of the page. The fourth region is a set of nodes around the left-hand, top and right-hand boundaries that are assigned the Dirichlet condition (**B** lines parallel). The bottom boundary automatically assumes the Neumann condition (**B** lines normal) so a specification is not needed.

To begin, copy the files `HMAG.MIN`, `HMAG.BIN`, `HMAG2.MIN`, `HMAG2.BIN` and `HMAG2.SCR` to a convenient working directory (*i.e.*, `\TRICOMP\BUFFER`). The first step is to generate a file of geometric information, a required input for **BStat**. Make sure the data directory in the **TC** program launcher is set to your working directory, and then run the **Mesh** program from **TC.** Click on *Load script (MIN)*, pick `HMAG.MIN` in the dialog. and click *OK*. The file has the format described in the **Mesh** manual. It contains a set of vectors that define the shapes in Figure 2. Click on the *Process* command. When mesh generation is complete, click on *Save mesh (MOU)* to create the file `HMAG.MOU`. At this point, you can use commands in the plot menu to inspect the geometry.

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

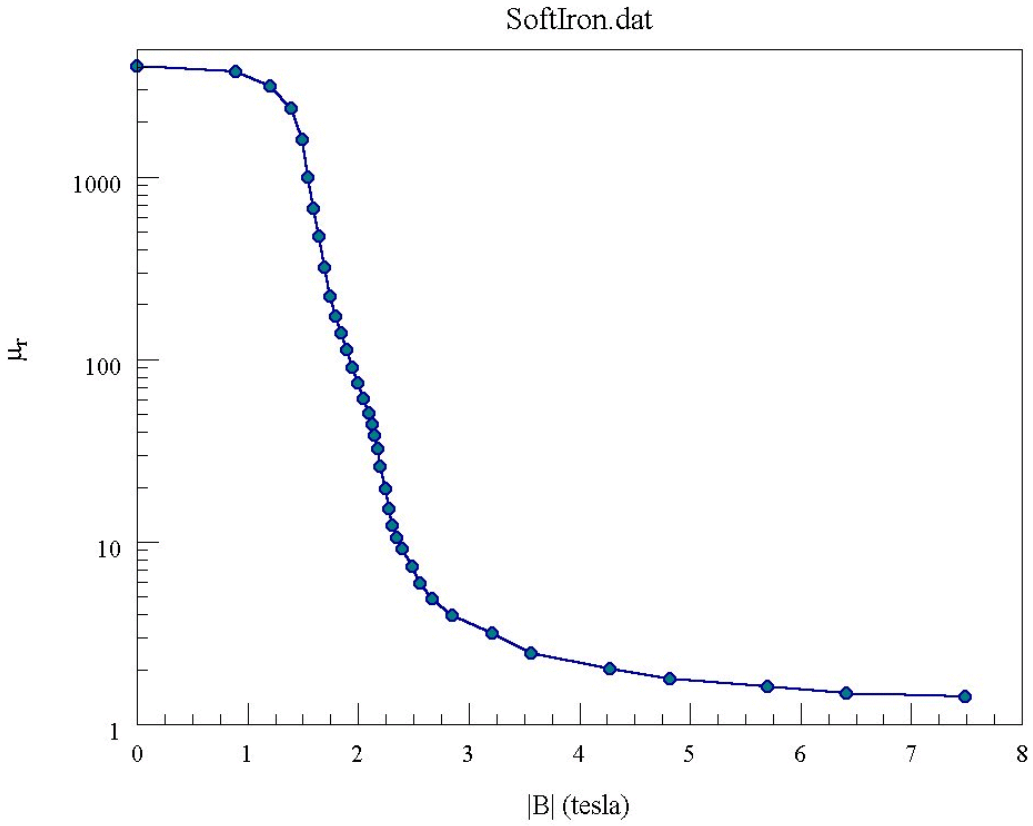

Figure 3. Field-dependent relative magnetic permeability

```
* FILE HMAG.BIN
SET ResTarget 5.0E-8
SET Geometry Rect
SET DUnit 100.0
SET Omega 1.90 1.95
SET MaxCycle 3000
* Main solution volume
REGION 1 Mu 1.0
* Return yoke
REGION 2 Mu Table SoftIron.dat
* Coil
REGION 3 Current -30000.0
REGION 3 Mu 1.0
* Dirichlet boundary
REGION 4 VecPot 0.0
ENDFILE
```

The first group of commands beginning with the keyword *SET* controls the solution process. For example, `SET DUNIT 100.0` specifies that dimensions in the **Mesh** file are given in centimeters. Section 5 describes the *SET* commands in detail. The second group of commands beginning with the word *REGION* defines material properties associated with solution regions. The air and coil regions are assigned a fixed value of relative permeability, $\mu_r = 1.0$. The coil region is also assigned a current of 30,000 A-turns. The total drive current (top and bottom) is therefore 60,000 A-turns. To simulate saturation effects, the iron (Region 3) is associated with a lookup table that defines a field-dependent value of magnetic permeability (Section 7). The file `SOFTIRON.DAT` is in ASCII format and can be inspected with an editor. Figure 3 shows the defined variation of $\mu_r = \mathbf{B}/\mu_o\mathbf{H}$ as a function of $|\mathbf{B}|$. Several tables are supplied with **BStat**, and you can also create your own tables to represent specialized materials. The final command sets the fixed vector potential condition $A_z = 0.0$ along boundaries.

Exit the editor and click the *Start run* command in the *Run* menu. Pick the file `hmag.bin` and click *OK* to start the solution. **BStat** applies an iterative relaxation technique to solve the finite-element equations while adjusting material properties in the iron core following the table `SOFTIRON.DAT`. The solution takes a few seconds on a high-performance personal computer. When the solution is complete, the program creates the file `HMAG.BOU` which contains information on the locations of nodes and the associated value of vector potential.

Run **VBStat** from **TC** to analyze the solution. Click the *Load solution* command in the *File* menu and pick `HMAG.BOU`. The program creates the default plot of Fig. 1 showing contours of A_z. In a two-dimensional solution the contours lie along lines of magnetic flux density. In the *Spatial plot* menu click on *Plot type* and choose *Element*. Next, click on plot quantity and choose *Mu (relative)* to display the plot of Fig. 4. At the given drive current, the major part of the core volume is near saturation except for the top-left and top-right areas.

Version 5.0 of **VBStat** incorporates a feature that we can use to generate a more detailed picture of saturation near the pole tip. Run **Mesh** and process the file `HMAG2.MIN`. The solution covers a subset of the volume of `HMAG.MIN` near the air gap. Save the mesh file and return to **BStat**. Use the internal editor to inspect the file `HMAG2.BIN`. The file has the following content:

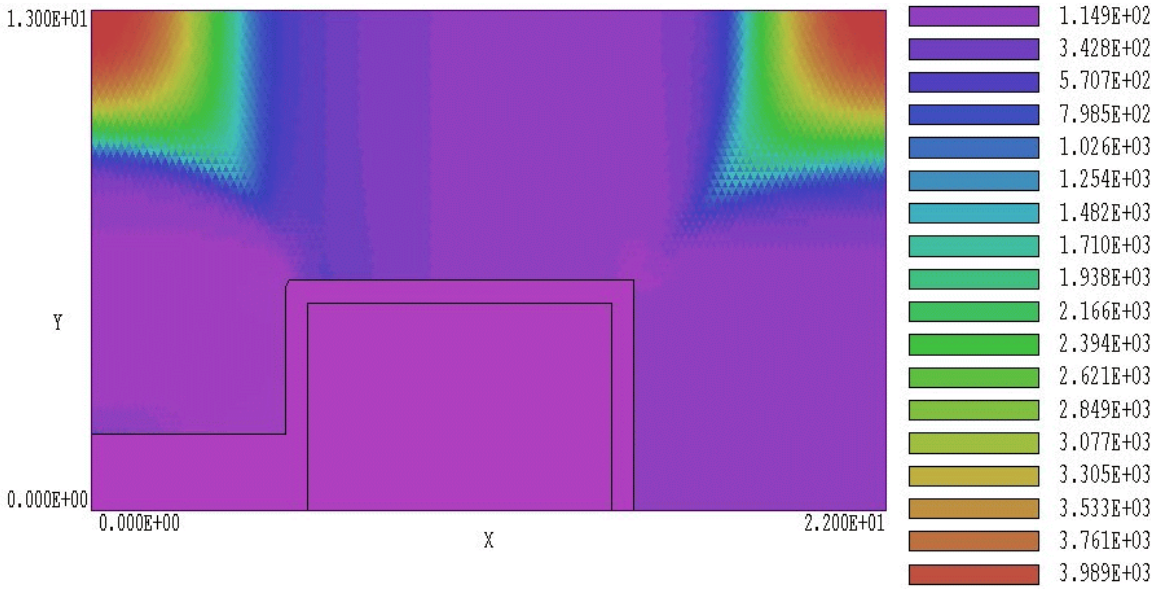

Figure 4. Variation of μ_r in the solution `hmag`

```
* FILE HMAG2.BIN
SET ResTarget 1.0E-9
SET Geometry Rect
SET DUnit 100.0
SET Omega 1.90 1.95
SET MaxCycle 5000
SET Boundary HMag
SET MatAvg 0.10 25 150
* Main solution volume
REGION 1 Mu 1.0
* Return yoke
REGION 2 Mu Table SoftIron.dat
ENDFILE
```

Material properties for regions inside the solution volume are the same as those in `HMAG.BIN`. The main difference is the command `SET Boundary HMag`. This command specifies that the outer boundary of the solution volume should be set to fixed values of A_z (Dirichlet condition) determined by interpolating in the solution space of `HMAG.BOU`. Run the **BStat** solution and then load the file `hmag2.bou` into **VBStat**. The program displays the microscopic view of core saturation of Fig. 5. To complete the session, we shall run a script that performs automatic calculations. To inspect the file content, click on *Edit script* in the *File* menu and pick the file `HMAG2.SRC`. The file has the following content:

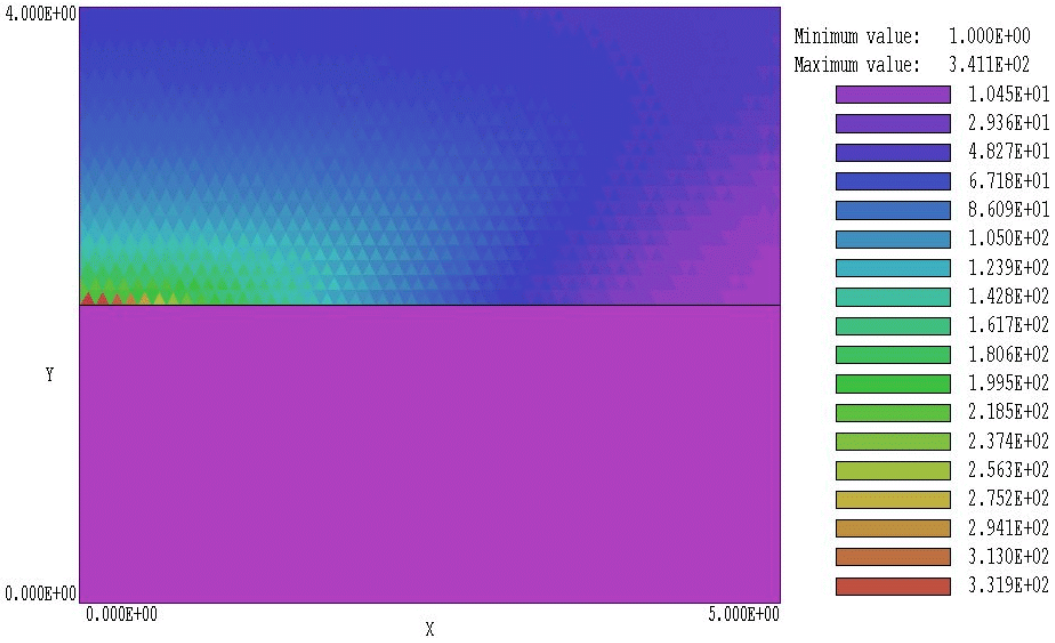

Figure 5. Detail of the variation of μr in solution `hmag2`

```
INPUT HMAG2.BOU
OUTPUT HMAG2
NSCAN 20
SCAN 0.0 0.0  5.0 0.0

ENDFILE
```

The first command ensures that the file `HMAG2.BOU` is loaded while the second command opens the data listing file `HMAG2.DAT`. The *Scan* command writes a set 21 data lines listing the vertical component of magnetic flux density along the horizontal midplane of the air gap. The following is an excerpt from the listing:

```
Scan between points
    XStart:  0.000E+00   YStart:  0.000E+00
      XEnd:  5.000E+00     YEnd:  0.000E+00

   X           By, Br
=======================
   0.000E+00   1.687E+00
   2.500E-01   1.687E+00
   5.000E-01   1.687E+00
   7.500E-01   1.687E+00
   1.000E+00   1.686E+00
   1.250E+00   1.685E+00
   1.500E+00   1.684E+00
   1.750E+00   1.682E+00
   2.000E+00   1.679E+00
   2.250E+00   1.676E+00
   2.500E+00   1.672E+00
   2.750E+00   1.666E+00
   3.000E+00   1.659E+00
   3.250E+00   1.650E+00
   3.500E+00   1.638E+00
   3.750E+00   1.620E+00
   4.000E+00   1.599E+00
   4.250E+00   1.573E+00
   4.500E+00   1.540E+00
   4.750E+00   1.497E+00
   5.000E+00   1.456E+00
```

Using the two solutions we have generated, you can experiment with the extensive plotting and analysis functions of the **VBStat** postprocessor. We also recommend that you run some of the prepared examples supplied with **BStat** to understand the program capabilities and the nature of two-dimensional magnetostatic solutions. The following chapters give detailed information that will help you prepare input files for your own applications.

3. Organizing runs

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

BStat requires a minimum of two input files.

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

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

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

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

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

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

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

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

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

4. Structure of the command file

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

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

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

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

The bottom number is interpreted as 5.0.

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

```
* FILE MAG_LENS.BIN
* Solenoid with flux return
* Half length of solenoid coil: 16 cm
* Current density for 0.1 tesla: 7.9365E4
```

```
* Coil current: 1.2698E4 A
SET ResTarget 5.0E-8
SET MaxCycle 1000
SET Geometry Cylin
SET DUnit 100.0
SET Omega 1.90 1.95
REGION 1 Mu 1.0
REGION 2 Mu 1000.0
REGION 3 Current 1.2698E4
REGION 4 VecPot 0.0
ENDFILE
```

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

```
SET
REGION
```

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

```
SET GEOMETRY CYLIN
```

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

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

```
Region   RegNo    Keyword    Value
```

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

```
REGION 2 VecPot 0.0
```

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

5. Commands for general program control

BStat script commands divide into two groups: those that control the program operation and those that set the physical properties of regions. We begin with the program commands. They begin with the keyword *Set*. Each command is written as it might appear in the `FName.BIN` file.

SET DUNIT 1.0E4

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

SET GEOMETRY CYLIN

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

SET OMEGA 1.85 1.90

This command controls the *over-relaxation factor* for the solution, a number in the range 1.0 to 2.0. The two parameters are *OmegaMin* and *OmegaMax*. The over-relaxation factor rises from *OmegaMin* to *OmegaMax* during an adjusted number of cycles. The default values are *OmegaMin* = 1.90 and *OmegaMax* = 1.95. Generally, high values give faster convergence. Lower the over-relaxation factor if the solution does not converge.

SET MAXCYCLE 200

The maximum number of cycles in the iterative solution. The default value is 2500.

SET RESTARGET 5.0E-6

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

SET MATAVG 0.2
SET MATAVG 0.2 25
SET MATAVG 0.2 25 250

This command affects only runs where there are non-linear materials described by *B*-μ tables. It sets parameters that control how the value of magnetic permeability is adjusted during the relaxation process. The three parameters are *Avg* (real), *NAdj* (integer) and *NMatMin* (integer). The last two numbers are optional. When **BStat** updates μ in an element, it takes an average of the old value and the new value determined from the present value of $|\mathbf{B}|$. The parameter *Avg* is the weighting of the new value. It must be in the range 0.0 to 1.0. Higher values may give a faster solution. Reduce the value if the solution does not converge. The default is *Avg* = 0.05. The second quantity, *NAdj*, is the number of relaxation cycles between adjustments of μ. The default is *NAdj* = 50. Finally, the quantity *NMatMin* is the number of relaxation cycles before the correction of μ values begins. Initially, the magnetic permeability in variable elements is set to the value at $|\mathbf{B}|$ = 0.0. If the permeability is adjusted before the magnetic solution has relaxed, the false differences in magnetic flux density may result in large variations of μ or invalid interpolations. Use a higher value of *NMatMin* for slowly-converging solutions. The default is *NMatMin* = 100.

Note: In solutions with non-linear materials, **BStat** reports the quantity *dGamma* in addition to the relative residual during the relaxation process. This quantity is a measure of the fractional change in μ averaged over the solution volume. It should have a value small compared to unity ($\leq 10^{-3}$) for an accurate solution.

SET Interp Linear

Set the interpolation method for tables of non-linear materials and permanent magnets. The options are *Spline* (default) and *Linear*. Use *Linear* for noisy or discontinuous data.

6. Commands for material properties

The **BStat** command set for defining the material properties of regions is simple. The three commands to set the physical properties of materials all start with the word **Region** and have the same format.

Region RegNo Keyword Value

Here, *RegNo* (an integer) is the Region Number defined in the **Mesh** input file. *Keyword* is one of the words listed below and *Value* is a real number.

REGION 4 VECPOT 0.0

The keyword *VecPot* designates that the vector potential of the region (or the stream function rA_θ for cylindrical problems) is fixed and will not change during the relaxation. The command sets the amplitude of A_z (rectangular simulations) in tesla-m or rA_θ (cylindrical simulations) in tesla-m^2. The default value in all fixed regions is 0.0. The most common use of this command is to set Dirichlet boundary conditions. See Sect. 9 for a discussion of the physical meaning of vector potential values on boundaries.

REGION 2 MU 2500.0

This command sets a constant value of the relative magnetic permeability ($\mu_r = \mu/\mu_o$) in the region (linear magnetic material).

REGION 7 CURRENT 24000.0

This command sets the region current in amperes. **BStat** distributes the current uniformly over the region cross section. To define variations of current density, split an area into several regions. Both the *Mu* and *Current* commands can apply to the same region.

REGION 2 MU TABLE SOFTIRON.DAT

This is an alternate form of the *Mu* command to set the relative magnetic permeability of non-linear magnetic materials. The command parameters consists of the region number, the keywords *Mu* and *Table*, and the name of a material file (*i.e.*, `SOFTIRON.DAT`). The file must be in the working directory. It contains a table consisting of sets of values of *B*(tesla) and μ_r over the range of magnetic flux density that will be encountered in the solution. The following section describes permeability tables.

7. Modeling non-linear magnetic materials

BStat can represent isotropic non-linear magnetic materials like soft iron and ferrites. Typical applications include modeling saturation effects in magnet poles and computing forces in motors and solenoids. The saturation properties of materials are entered as tables of values of the magnitude of magnetic flux density (in tesla) and the corresponding relative permeability. On each iteration cycle of the relaxation solution, **BStat** calculates the amplitude of magnetic flux density for each element with a tabulated material. The program then uses the interpolated value of the magnetic permeability to update the coupling coefficients for all connected vertices. The two relaxation processes proceed in tandem, leading to self-consistent values of the magnetic permeability and vector potential. For maximum speed, **BStat** updates coupling coefficients only in non-linear regions and skips the process altogether when no tables have been entered.

Note that **BStat** solutions are limited to static fields and to magnetic materials that are 1) isotropic (no special axis in space) and 2) reversible. Reversible (or *soft*) magnetic materials have a narrow hystersis curve. The **TriComp** solution program **PerMag** treats static fields with hard materials and permanent magnets. The **Nelson** program finds solutions with AC magnetic fields and eddy currents. Finally, the **Pulse** program solves the general problem of magnetic field diffusion into resistive materials with arbitrary temporal variations of coil current.

The key to flexible modeling of non-linear materials is program input through *tabular functions*. A tabular function is a text file consisting of up to 128 entry lines of values of magnetic flux density and corresponding relative magnetic permeability μ/μ_o. You can prepare files with a text editor or spreadsheet. You can also use published data or digitized experimental traces. The example below shows a portion of the tabular function for soft iron supplied with the **BStat** package. Note that the file syntax conforms to the rules of the command script. The free-form parser accepts real numbers in any format with a choice of delimiters. You can add documenting comment lines starting with asterisk. The end of the data is marked with the *EndFile* command. You can also add any text after *EndFile*.

```
* File SOFTIRON.DAT
* Generic soft iron table
* Adapted from Superfish/Poisson
* Los Alamos National Laboratory
* Checked for cubic splines, 11/95
    0.0000    4075.45105
    0.8944    3768.28312
    1.2000    3166.80993
    1.4000    2380.64690
       ...
    2.8498    3.94592075
    3.2074    3.15384615
    3.5644    2.45501166
    4.2782    2.02377622
    4.8134    1.76596737
    5.7052    1.60606061
    6.4186    1.48298368
    7.4887    1.42983683
ENDFILE
```

Although the lines of the example are ordered by increasing value of *B*, this ordering is not required. **BStat** sorts the list before use and records the final order in the file FNAME.BLS. Furthermore, the intervals need not be uniform. Note that the entries in SOFTIRON.DAT are clustered near the saturation transition. The maximum number of tables for all regions is 32. A table requires a minimum of 5 entries. You must ensure that the tabular functions extend over the full range of magnetic flux density that will be encountered in the solution. The interpolation routines return 0.0 for values of |**B**| that are off the table.

BStat uses cubic splines for interpolation of the tables. This method minimizes recalculation time and gives smooth interpolations that aid convergence. It is important to note that the quality of the interpolation depends on the nature of the numerical data. The magnetic permeability and its first derivative should vary smoothly over the table range. The cubic spline routines will faithfully match values at the entry points and preserve continuity of the first derivative. If the data has discontinuities, the cubic spline values may be quite inaccurate between data points. You can check the fidelity of the interpolations by inspecting the listing file FNAME.BLS. After listing the sorted table, **BStat** records a sample set of interpolated values.

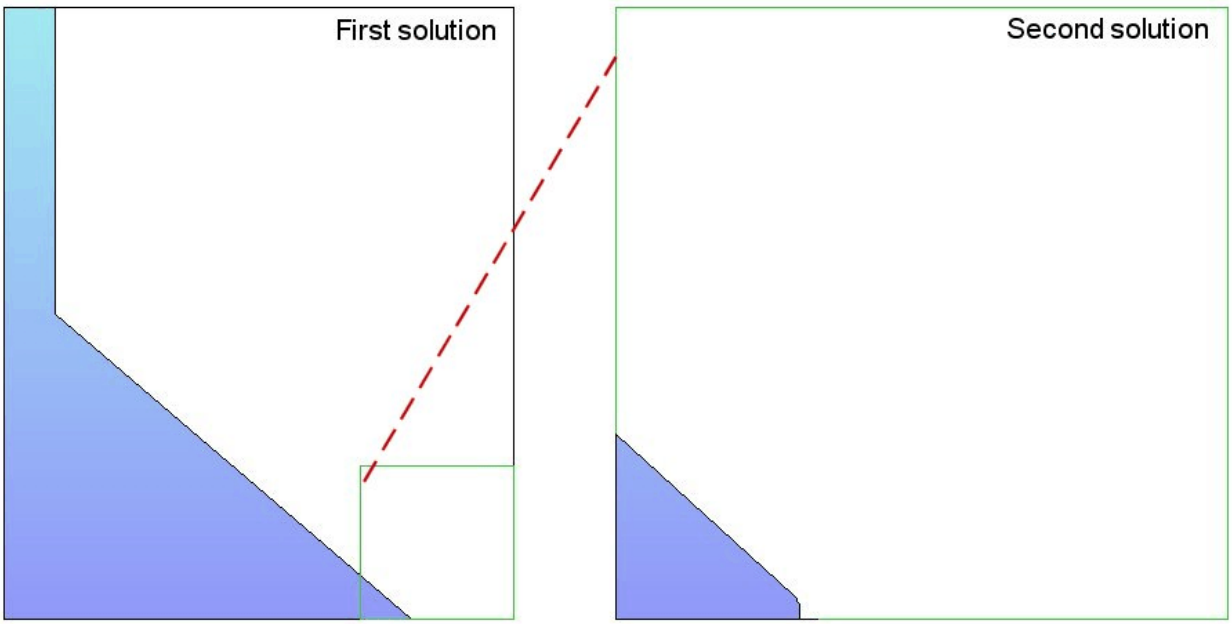

Figure 6. Interpolated Dirichlet boundary for a microscopic solution.

8. Boundary and superposition commands

Version 5.0 of **BStat** and the other **TriComp** solution programs incorporate a powerful new feature that enables accurate calculations of fields near small features in a large solution space. Figure 6 illustrates the procedure for an electrostatic solution. Suppose we wanted a precision calculation of the 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 (**Mesh** manual, Section 6) to create small elements near the tip. A limitation to this technique arises from the structured mesh used in **TriComp** – the region of small elements must extend the full length of the solution volume.

Figure 6 illustrates an alternate approach. We create a second solution that extends over a small subregion of the original solution (green outline). The microscopic solution contains any electrodes that may enter the subregion. The difference is that the electrode 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 in a variable-potential Dirichlet boundary. Values of ϕ 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.

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

SET BOUNDARY FPrefix [BndScale]
SET BOUNDARY FEmitMacro

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 vector potential values transferred from Solution 1 to Solution 2. The default is *BndScale* = 1.0.

BStat issues an error message under the following conditions: 1) the output file for Solution 1 (`FPREFIX.BOU`) is not available in the working directory, 2) Solution 2 does not fit completely inside Solution 1, 3) Solution 1 and Solution 2 have different symmetries, or 4) 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.

A node in Solution 2 is on the boundary under two conditions:

- It has indices $K = 1$, $K = K_{max}$, $L = 1$ or $L = L_{max}$. In solutions with cylindrical symmetry, points on the axis ($L = 1$ and $r = 0.0$) are automatically not set to the Dirichlet condition.

- One of the adjacent elements has *RegNo* = 0. This condition means that the boundary of Solution 2 need not be rectangular.

Figure 7 illustrates the boundary point criteria for an electrostatic solution. Solution 1 (*BoundaryVal01*) represents electric fields between two spherical-section electrodes separated by a 3.00" gap. We want to find the fields on a small dielectric body inserted at the center of the gap. Solution 2 (*BoundaryVal02*) is a spherical region (radius 1.00") at the midpoint. The microscopic solution contains a dielectric sphere of radius 0.10". Note the assignment of potential values on the circular outer boundary of Solution 2. The lower boundary of the cylindrically-symmetric solution has not been set to the Dirichlet condition; therefore, the potential can adjust to the presence of the dielectric.

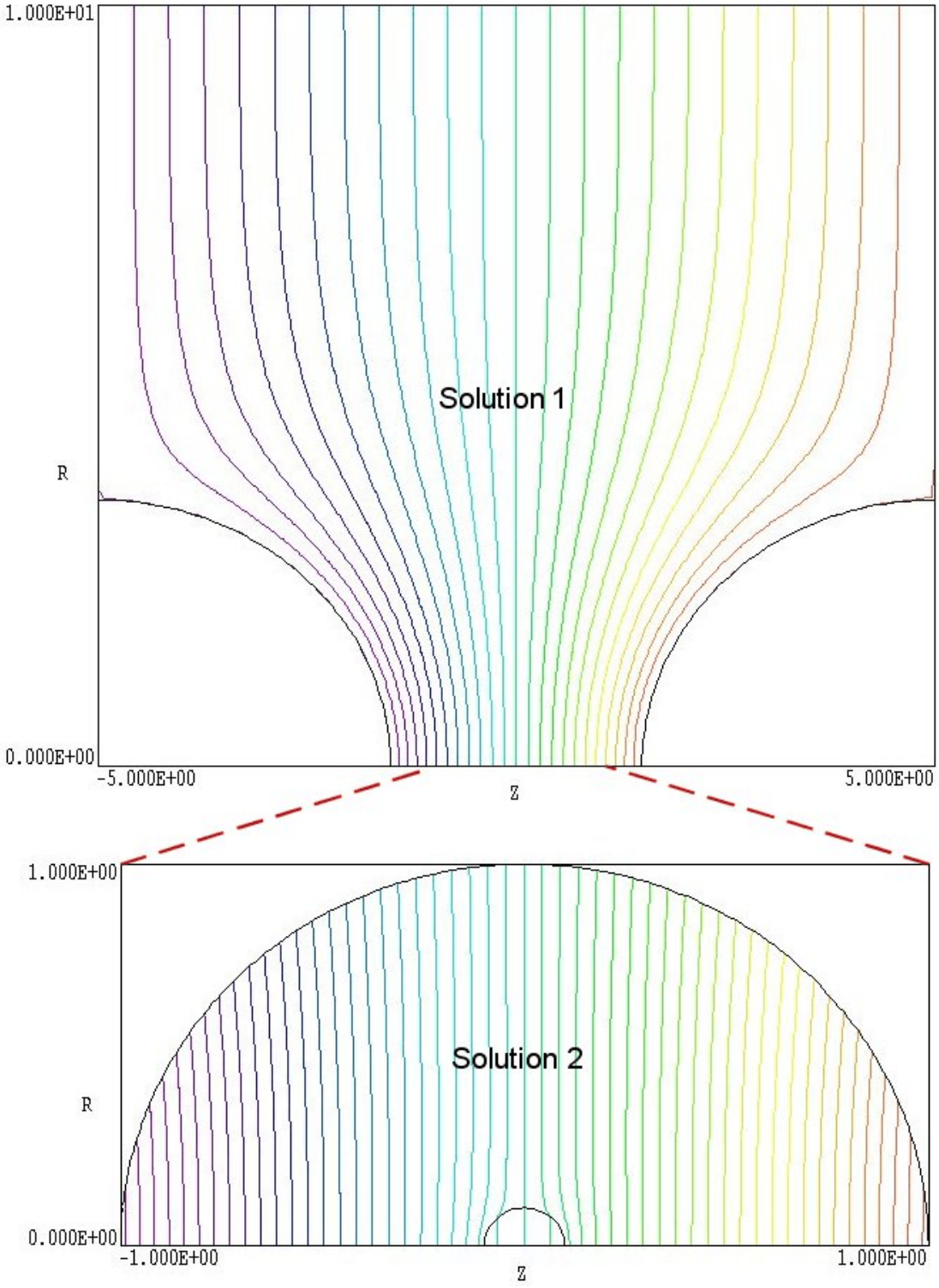

Figure 7. Example of a two-stage electrostatic solution using the *Boundary* command.

In response to the *Superposition* command, **BStat** 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.

SET SUPERPOSITION FPrefix [SuperScale]
SET 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 vector potential values transferred from Solution 1 to Solution 2. The default is *BndScale* = 1.0.

In response to the *Superposition* command, **BStat** opens the file `FPrefix.BOU` after Solution 2 has been completed. The program performs an interpolation in the space of Solution 1 to determine the vector potential or stream function at each node location in Solution 2 and adjusts values according to

$$A'_{z2} = A_{z2} + SuperScale \times A_{z1} , \qquad \textbf{(1)}$$

before writing the output file.

You must ensure that the superposition is physically correct. The procedure is usually invalid in solutions with non-linear materials.

9. Vector potential and boundary conditions

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

In rectangular problems (variation in *x* and *y* with no variation in *z*) **BStat** determines the vector potential A_z created by currents that move in the *z*-direction. The components of magnetic flux density are given by

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

and

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

For problems in cylindrical coordinates, **BStat** calculates the quantity rA_θ, where A_θ is the vector potential created by azimuthal currents. The quantity rA_θ is usually called the stream function and denoted by ψ. The components of magnetic flux density are given by

$$B_r = -\frac{1}{r}\frac{\partial(rA_\theta)}{\partial z} = -\frac{1}{r}\frac{\partial \psi}{\partial z}.$$

and

$$B_z = \frac{1}{r}\frac{\partial(rA_\theta)}{\partial r} = \frac{1}{r}\frac{\partial \psi}{\partial r}$$

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

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

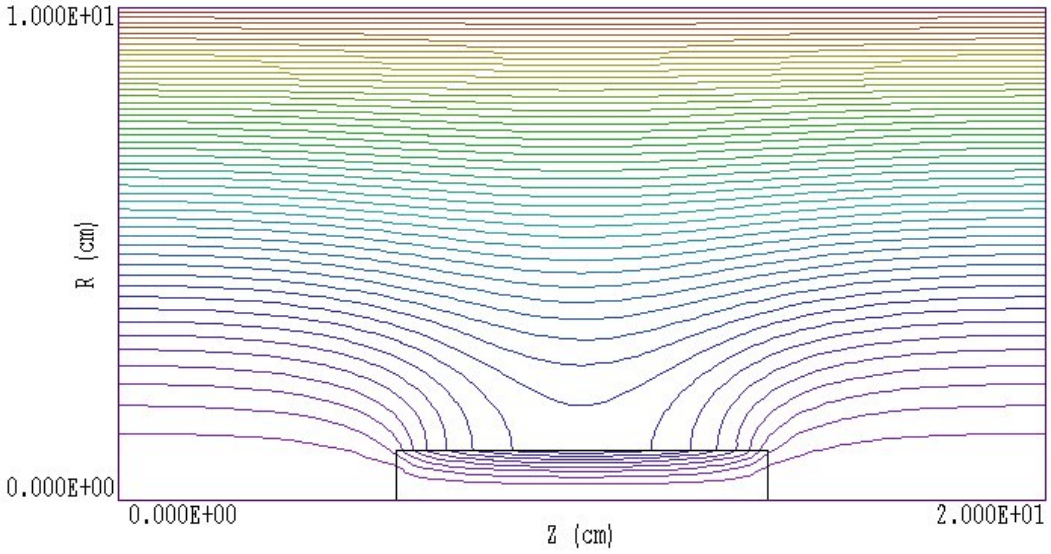

Figure 8. Setting boundary values of the vector potential to create a region of uniform flux. Cylindrical geometry. Vertical: *r* from 0.0 to 0.1 m. Horizontal: *z* from 0.0 to 0.2 m. Lower boundary: $rA_\theta = 0.0$ tesla-m^2. Upper boundary: : $rA_\theta = 5.0 \times 10^{-4}$ tesla-m^2.

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

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

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

As an example, Fig. 8 shows lines of **B** around an iron cylinder immersed

in a uniform solenoidal field of 0.1 tesla. The uniform field was created by setting ψ = 5.0 x 10-4 tesla-m^2 on the outer boundary at r_o = 0.10 m.

With this background, we can now discuss the implications of the two types of boundary conditions used in **BStat**.

Dirichlet

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

Neumann

A Neumann boundary is one where the normal derivative of the vector potential is specified. The boundaries in **BStat** are limited to the special case $\partial A/\partial n = 0$. The special Neumann condition implies that **B** is normal to the boundary. One of the advantages of the finite-element method is that all boundaries that are not fixed automatically satisfy the special Neumann condition, even if they are slanted or curved. Neumann boundaries are often used to reduce computation time for symmetric systems.

10. Running BStat interactively

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

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

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

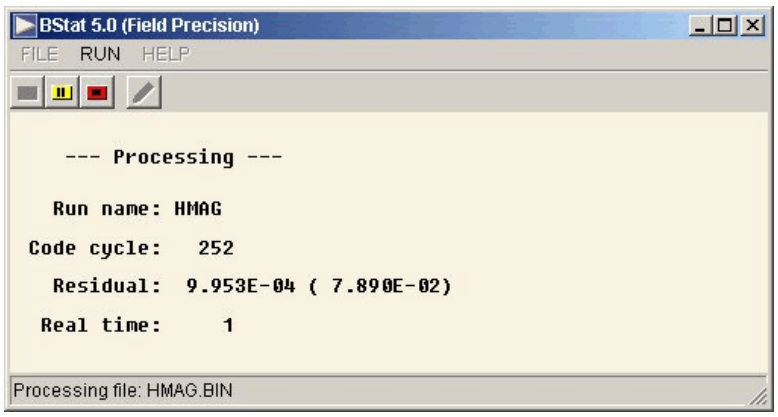

Figure 9. **BStat** in the interactive mode

Create input file

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

Edit input file
Edit listing file
Edit file

The commands call the editor to inspect or to modify ASCII input and output files for **BStat**. Choosing a file from an alternate directory does not change the working directory. The *Edit input file* command shows a list of all files with names of the form `FPREFIX.BIN`. The *Edit listing file* command displays files with names `FPREFIX.BLS`.

The *Run* menu has four commands.

Start run

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

Pause run

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

Stop run

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

11. Running BStat from the command prompt

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

```
..\BStat SOLENOID <Enter>
```

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

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

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

Type

```
SWRUN <Enter>
```

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

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

12. VBStat file menu

VBStat (for View BStat) is the interactive graphical analysis program for output files from **BStat**. The program automatically adjusts labels and calculated quantities depending on whether the solution is dielectric or conductive:

Program units

Spatial dimensions: meters or units set by *DUnit*.
Magnetic flux density: tesla
Vector potential: tesla-m
Current: amperes
Forces: newtons
Torques: newton-m

VBStat has the following popup menus: *File*, *Spatial plots*, *Analysis*, *Scan plots* and *Help*. When the program starts only the *File* and *Help* menus are

active. You must load a data file in order to create plots or to perform analyses. This section reviews options in the *File* menu.

Load solution file

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

Open data listing file

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

Close data listing file

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

Run script

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

Create script

This command allows you to create script files using the internal editor of the post-processor. A dialog box requests a file prefix. The

resulting script file will be saved as FPREFIX.SCR. Next, the program opens the file in the editor and writes a list of allowed commands for reference. This list follows the *EndFile* command and will be ignored. Enter analysis commands above the *EndFile* command. After saving the file, you can run it using the *Run script* command.

Edit script

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

Edit data file

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

Edit file

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

13. VBStat spatial plot menu

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

Plot type

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

Mesh. Outline of the elements in the computational mesh.

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

Contour. Lines that follow constant values of a calculated quantity. In two-dimensional magnetostatic solutions constant values of A_z or rA_θ are parallel to lines of **B**. Contours are separated by equal increments of magnetic flux. Therefore, in cylindrical coordinates the line spacing increases near the axis even if magnitude of **B** is uniform.

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

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

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

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

Plotted quantity

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

VBStat supports the following plot quantities. Note that available quantities depend on the solution type (dielectric of conductive).

Contour plot quantities

Vector potential (A_z) or stream function (rA_θ)
Magnetic flux density amplitude, $|\mathbf{B}|$
Relative magnetic permeability, μ_r

Element plot quantities

A_z or rA_θ
Magnetic flux density amplitude, $|\mathbf{B}|$
Magnetic permeability, μ
Magnetic field amplitude, $|\mathbf{H}|$

Vector plot quantities

Magnetic flux density, **B**

Surface plot quantities

A_z or rA_θ
Magnetic flux density amplitude, $|\mathbf{B}|$
Horizontal component of flux density, B_x or B_z
Vertical component of flux density, B_y or B_r
Relative magnetic permeability, μ
Magnetic field amplitude, $|\mathbf{H}|$

Plot limits

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

Toggle grid

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

Set contour plot style

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

Toggle element outline

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

Toggle mouse/keyboard

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

Toggle snap mode

The mouse snap mode is a convenient feature of **VBStat**. When snap mode is active, the mouse returns the coordinate values closest to an integer multiple of the quantity *DSnap*. In other words, if *DSnap* = 0.5 and the mouse position is [5.4331,-2.6253], the returned coordinates are [5.5,-2.5]. By default, snap mode is *ON*. Snap mode is automatically turned off for coordinate input to the commands *Point calculation* and *Element properties*. Otherwise, the program would pick a location closest to the snap point rather than the point at the tip of the mouse arrow, giving misleading information.

Set DSnap

Set the distance scale for the mouse snap mode.

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

Zoom window

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

Zoom in

Magnifies the view about the center of the current plot.

Expand view

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

Global view

Returns the plot boundaries to the full solution area.

Pan

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

The following commands control the appearance of *Surface* plots. The commands are active only when a *Surface* plot is displayed. You may notice that there is a delay if you choose the *Surface* option in the *Plot type* command or if you change the plotted quantity when a *Surface* plot is active. To create the plot, **VBStat** must map the current quantity to a rectangular grid, a process that requires a large number of interpolations.

Rotate 3D image

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

View angle 3D

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

Set grid 3D

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

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

Default printer

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

Plot file (PostScript)

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

Plot file (BMP)

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

Plot file (PNG)

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

Copy to clipboard

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

14. VBStat analysis menu

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

Point calculation

VBStat employs a sophisticated interpolation technique that preserves discontinuities at material boundaries. The program gives correct values for magnetic flux density on both sides of a material boundary. Click on the command and then point to any position in a *Mesh*,

Region, Contour, Element or *Vector* plot. Note that snap mode is turned *OFF* for coordinate input. The program writes a subset of interpolated quantities to the window below the plot and also records complete information if a data file is open. To enter point coordinates by keyboard, use the *Toggle mouse/keyboard* command.

Toggle interpolation

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

Line scan

The line scan is one of the most useful functions of **VBStat**. After clicking on the command, supply two points with the mouse in a view of a *Mesh, Region, Contour, Element* or *Vector* plot to define a scan line. The snap mode is useful in this application (for example, you may want a scan to extend from 0.000 to 5.000 rather than 0.067 to 4.985.) The program computes a series of values of field quantities at equal intervals along the line. The information is recorded if a history file is open. The program also makes a screen plot of the currently selected quantity versus distance along the scan and activates the commands in the *Scan plot* menu (see Section 15). The program adds fiducial lines to the plot using intelligent grid selection. This means that the plot is adjusted to fill the screen and grids are drawn at useful intervals (like 0.05 or 2.00).

Scan plot quantity

With this command you can choose the quantity to display in screen and hardcopy plots of line scans. Pick the quantity from the list box and click *OK*. This setting has no effect on the history file listing which includes all field quantities. The available line scan quantities depend on the solution type. The options are: vector potential (A_z or A_θ), horizontal component of magnetic flux density (B_x or B_z), vertical component of magnetic flux density (B_y or B_r), magnetic flux density amplitude ($|\mathbf{B}|$),relative magnetic permeability (μ_r) and amplitude of

the magnetic field (|**H**|).

Set number of scan points

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

Element properties

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

Region properties

To see the calculated quantities associated with a region of the solution space, click the mouse close to any arc or line vector of the region. Partial results are shown on the screen and a complete analysis is included in the data file. **VBStat** calculates volume integrals to derive the following quantities:

1) Magnetic field energy, $\int\int\int dV\ B^2/(2\mu_r\mu_0)$.

2) The value and location of the peak magnetic field, |**B**|.

3) The rigid-body force if the region contains current.

4) Torque about the z-axis arising from coil forces if the solution has planar symmetry. The calculation is performed relative to the current torque axis origin.

The following quantities are calculated with surface integrals:

1) The rigid-body material force if the region has $\mu \neq \mu_o$. Note that coil forces arises from applied currents and material forces arise from alignment of magnetic dipoles to form current layers on surface.

2) Torque about the z-axis arising from material forces for solutions with planar symmetry. The calculation is performed relative to the current torque axis origin.

Line integrals

No line integral quantities are calculated in **VBStat**.

Volume integrals

No input is needed for this command. **VBStat** performs the following analyses:

1) Magnetostatic field energy density, $u = B^2/2\mu_r\mu_o$ over the solution space and region volumes. The output units are J/m in rectangular solutions and J in cylindrical solutions.

2) Coil forces, the volume integrals of $\boldsymbol{j}\times\boldsymbol{B}$ for all regions with non-zero current. The units are newtons/m for rectangular problems and newtons for cylindrical problems.

3) Coil torques, the volume integrals of $\tau_z = \boldsymbol{x} \times \boldsymbol{j} \times \boldsymbol{B}$ for all regions with non-zero current. The vector **x** is taken relative to the torque axis origin. The calculation is perrformed valid only for planar simulations. The units are N-m/m.

4) The location and value of the maximum magnetic flux density in the solution space and in each region.

Torque axis

Set the origin for torque calculations in planar simulations.

Matrix file

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

15. VBStat scan plot menu

The commands of the *Scan plot* menu become active when a plot is created following the line scan command. The commands to export plots were described in Section 12. This section describes unique commands of the Scan plot menu.

Oscilloscope mode

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

Toggle scan symbols

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

Toggle grid

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

Close scan plot

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

16. VBStat script file commands

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

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

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

```
..\VBSTAT GTEST <Enter>
```

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

The following commands can appear in a script:

Input SOLENOID.BOU

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

Output SW02

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

Point 5.65 10.68

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

Scan 0.00 0.00 10.00 0.00

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

Interpolation LSQ

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

Element 5.65 10.68

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

NScan 150

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

Region 5

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

LineInt 0.00 0.00 10.00 0.00

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

VolumeInt

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

Matrix Switch1 10 20 0.00 0.00 5.00 10.00

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

TorqueAxis X0 Y0

Param1: *X* coordinate of the axis for torque calculations, in units set by *DUnit*.
Param2: *Y* coordinate of the axis for torque calculations, in units set by *DUnit*.

Endfile

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

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

```
OPEN OUTPUT COMP
OPEN INPUT SWITCH01.BOU
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH02.BOU
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH03.BOU
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH04.BOU
SCAN 0.00 -50.00 0.00 50.00
ENDFILE
```

17. Format of the BStat output file

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

- Header with general information on the run
- Node and element information
- Region informaton

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

```
--- Run parameters ---
XMin: -1.270003E-01
XMax:  1.270003E-01
KMax:  101
```

```
YMin:  0.000000E+00
YMax:  2.540005E-01
LMax:  101
DUnit:  3.937000E+01
NReg:    5
ICylin:  1
```

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

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

```
--- Vertices ---
  k    l  RgNo RgUp RgDn       x             y           VecPot         MuUp          MuDn
=============================================================================================
   1    1   -2    1    0  0.000000E+00  0.000000E+00  0.000000E+00  1.000000E+00  0.000000E+00
   2    1    2    1    0  1.308568E-02  0.000000E+00  0.000000E+00  1.000000E+00  0.000000E+00
   3    1    2    1    0  2.614603E-02  0.000000E+00  0.000000E+00  1.000000E+00  0.000000E+00
   4    1    2    1    0  3.917244E-02  0.000000E+00  0.000000E+00  1.000000E+00  0.000000E+00
   5    1    2    1    0  5.215985E-02  0.000000E+00  0.000000E+00  1.000000E+00  0.000000E+00
   6    1    2    1    0  6.510645E-02  0.000000E+00  0.000000E+00  1.000000E+00  0.000000E+00
   7    1    2    1    0  7.801303E-02  0.000000E+00  0.000000E+00  1.000000E+00  0.000000E+00
 ...
```

Each data line contains the following quantities:

- The indices of the node (K, L)
- The region number of the node (*RgNo*) and region numbers for two associated elements (*RgUp* and *RgDn*). The upper element lies about the line between nodes (K, L) and (K+1, L) and the lower element lies below the line.
- The coordinates of the element in meters (x, y)
- The vector potential A_z at the node in tesla-m or the stream function rA_θ in tesla-m^2.
- The values of relative magnetic permeability in the upper and lower associated elements.

The region section consists of four title lines following by *NReg* data

lines, one for each region. For a dielectric solution, the region section has the following appearance:

```
--- Regions ---
RegNo Fix                          Mu          Current    Vector Pot        Area
================================================================================
   1    0    0    0    0    0  1.000000E+00  0.000000E+00  0.000000E+00  1.000000E-03
   2    0    0    0    0    0  1.000000E+00  0.000000E+00  0.000000E+00  1.000000E-03
```

A one in the second column designates a fixed-potential region such as a Dirichlet boundary.

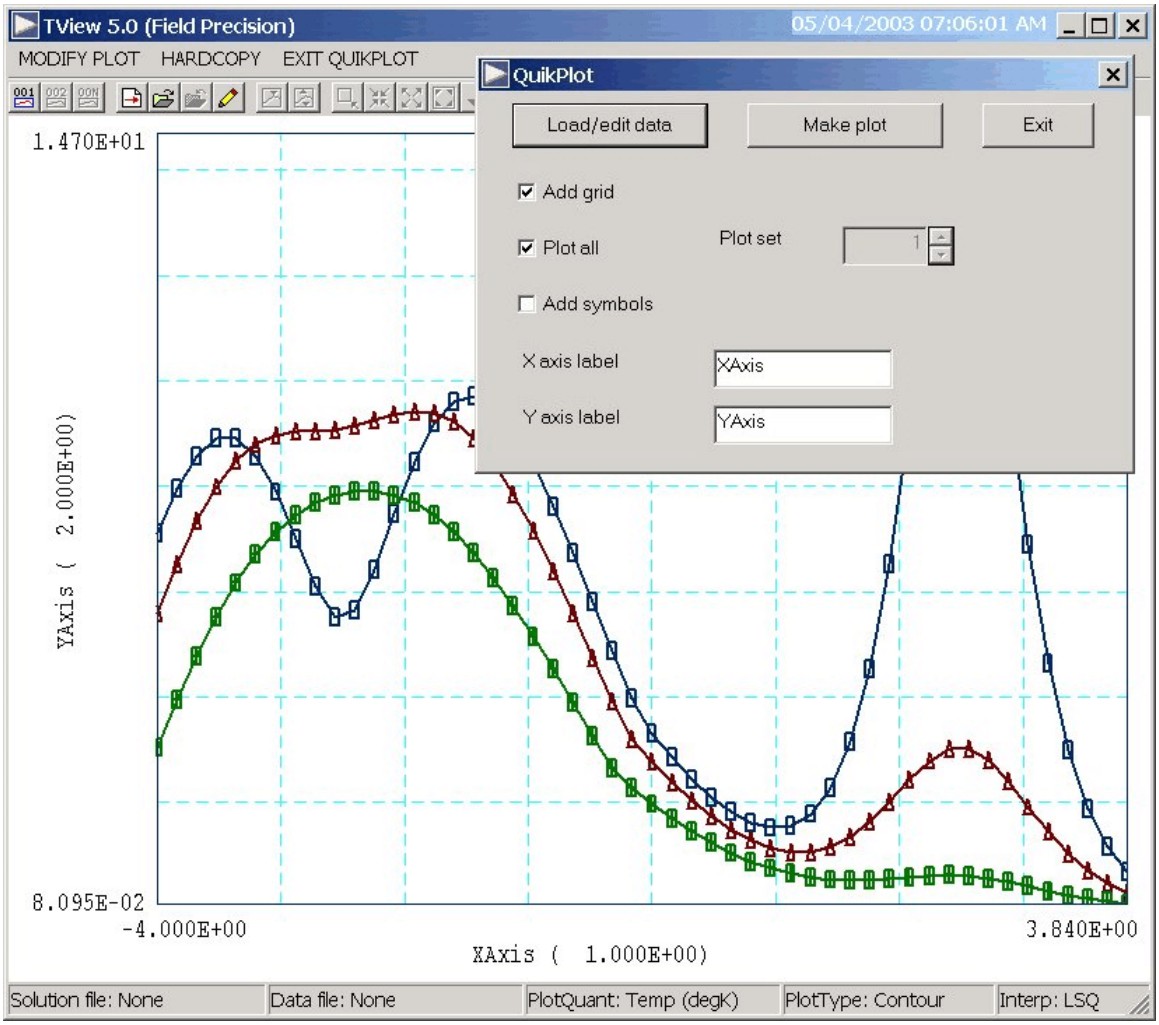

Figure 10. Control dialog superimposed on **QuikPlot** screen.

18. QuikPlot utility

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

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

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

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

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