TDiff

Finite-element Thermal Transport
with Temperature-dependent Materials

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

TDiff 5.0 Manual - Contents

Section 1. Program function

Section 2. Walkthrough example

Section 3. Physical units and conversion factors

Section 4. Organizing runs

Section 5. Structure of the command file

Section 6. Commands for steady-state solutions

Section 7. Temperature-dependent thermal conductivity

Section 8. Control of dynamic solutions

Section 9. Choosing a time step

Section 10. Material commands for dynamic solutions

Section 11. Diagnostic commands for dynamic solutions

Section 12. Avoiding numerical instabilities in dynamic solutions.

Section 13. Boundary conditions in thermal solutions.

Section 14. Running TDiff interactively

Section 15. Running TDiff from the command prompt

Section 16. TView file menu

Section 17. TView spatial plot menu

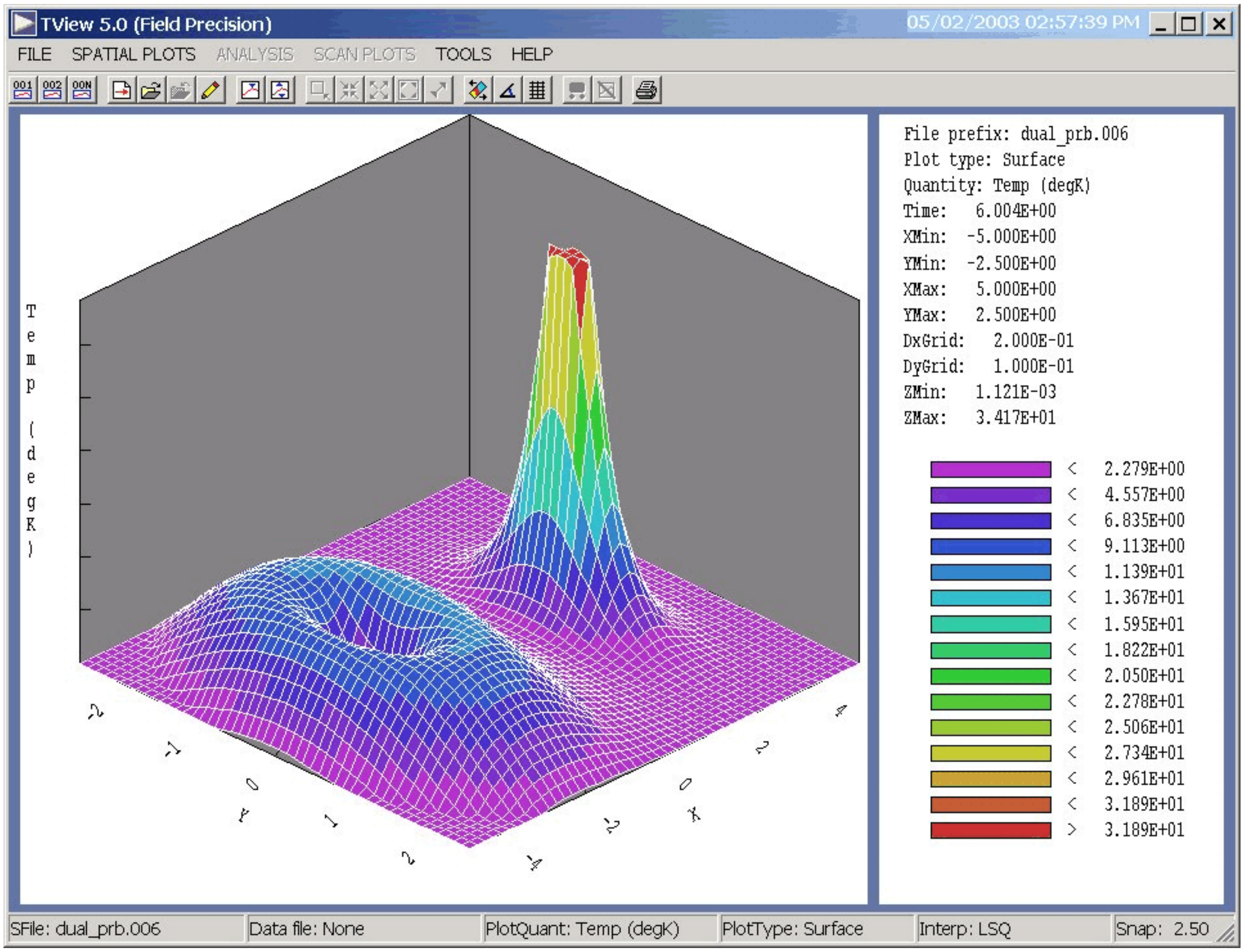
Section 18. TView analysis menu

Section 19. TView scan plot menu

Section 20. TView script operation

Section 21. TView output file format

Section 22. TView QuikPlot utility



```
File prefix: dual_prb.006
Plot type: Surface
Quantity: Temp (degK)
Time:    6.004E+00
XMin:   -5.000E+00
YMin:   -2.500E+00
XMax:    5.000E+00
YMax:    2.500E+00
DxGrid:    2.000E-01
DyGrid:    1.000E-01
ZMin:    1.121E-03
ZMax:    3.417E+01

🟪  <  2.279E+00
🟪  <  4.557E+00
🟦  <  6.835E+00
🟦  <  9.113E+00
🟦  <  1.139E+01
🟦  <  1.367E+01
🟩  <  1.595E+01
🟩  <  1.822E+01
🟩  <  2.050E+01
🟩  <  2.278E+01
🟩  <  2.506E+01
🟨  <  2.734E+01
🟧  <  2.961E+01
🟧  <  3.189E+01
🟥  >  3.189E+01
```

SFile: dual_prb.006 | Data file: None | PlotQuant: Temp (degK) | PlotType: Surface | Interp: LSQ | Snap: 2.50

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

1. Program function

The **TDiff** package handles a wide variety of thermal transport problems in solids or gels. The program solves the two-dimensional thermal diffusion equation in cylindrical and rectangular geometries by finite-element methods . **TDiff** can determine steady-state as well as dynamic solutions. In time-dependent solutions you can define multiple sources with arbitrary temperature histories using a flexible system of tabular function input. There is also the option to define temperature-dependent thermal conductivity and specific heat. In dynamic runs the program produces multiple data files at specified times and a history file of temperature at given locations. Data files can be analyzed with **TView**, an interactive graphical post-processor. The package also contains the standard Field Precision utility **Probe** for inspecting history files.

The walkthrough example in the following section gives an overview of the solution process. Section 3 summarizes units and conversion factors used in **TDiff** solutions. Section 4 outlines typical steps in run while Section 5 covers the structure of the command script that controls the program. Sections 6 and 7 describe script commands to set control parameters and material properties for steady-state solutions. Sections 8 through 11 cover script commands for dynamic solutions. Section 12 reviews special correction features in **Mesh** to avoid numerical instabilities in dynamic simulations. Section 13 discusses the interpretation of boundary conditions in thermal solution. Section 14 covers operation of **TDiff** as an interactive Windows program that you can run from the **TC** program launcher. Section 15 explains how to run **TDiff** in the background from the Command Prompt or under the control of the **GCon** utility for extended data runs. Sections 16 through 20 address operation of the **TView** post-processor used to create plots and to perform quantitative analysis. Finally, Section 21 documents the format of the **TView** output file.

2. Walkthrough example

You can test your installation with the following example that illustrates the steps of a **TDiff** solution. In the discussion we assume that the programs are installed in the directory `\TRICOMP` and that input and output files will be stored in the directory `\TRICOMP\BUFFER`. In preparation move the files `WALKTHROUGH.MIN`, `WALKTHROUGH.TIN`, `WALKTHROUGH.TMP` and `WALKTHROUGH.CND` to `\TRICOMP\BUFFER`. Run **TC** and use the commands at the bottom of the menu to make sure that the program directory is set to `\TRICOMP` and the data directory to `\TRICOMP\BUFFER`.

The simulation illustrates a dynamic calculation with time-dependent applied temperature and temperature-dependent thermal conductivity. Although the example does not represent a specific application, similar setups may be encountered in simulations of medical procedures. Figure 2 shows the geometry. A cylindrical metal probe penetrates a spherical volume of material with a relatively high value of non-linear thermal conductivity *k*. This material is surrounded by a infinite region of poor conductor at ambient temperature (37 °C). The procedure is to raise the temperature of the probe to 95 °C and to determine the range of heated material after a treatment time of 1000 seconds.

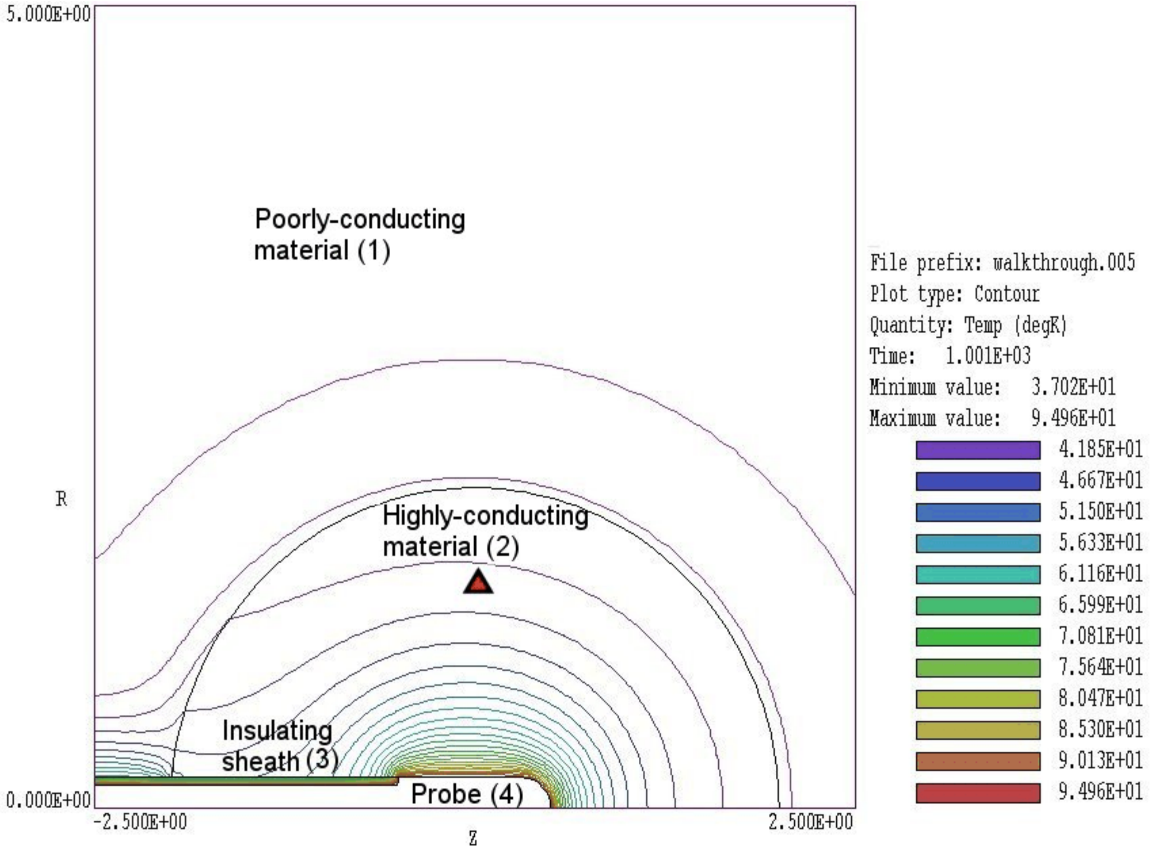


Figure 2. Geometry for the `WALKTHROUGH` example, showing region numbers and the location of the probe (close-up view).

From **TC** run **Mesh** then process and save the mesh following the instructions in the **Mesh** manual. The program creates the files `WALKTHROUGH.MLS` and `WALKTHROUGH.MOU`. The first file is a listing of run information that you can usually ignore if there are no problems. The second file describes the mesh – it is a listing of node coordinates and region numbers associated with nodes and elements. Like all **TriComp** files, `WALKTHROUGH.MOU` is in ASCII format so you can inspect it with a text editor or transfer the information to your own programs. Before leaving the program, check out the mesh using the plot functions of **Mesh**. The file illustrates good use of variable resolution to achieve high accuracy. It is not possible to simulate an infinite system in a finite-element calculation, but we can come close. Note that there is an extended region of surrounding material modeled at low resolution and that the outer boundary is set to the ambient temperature value of 37 °C.

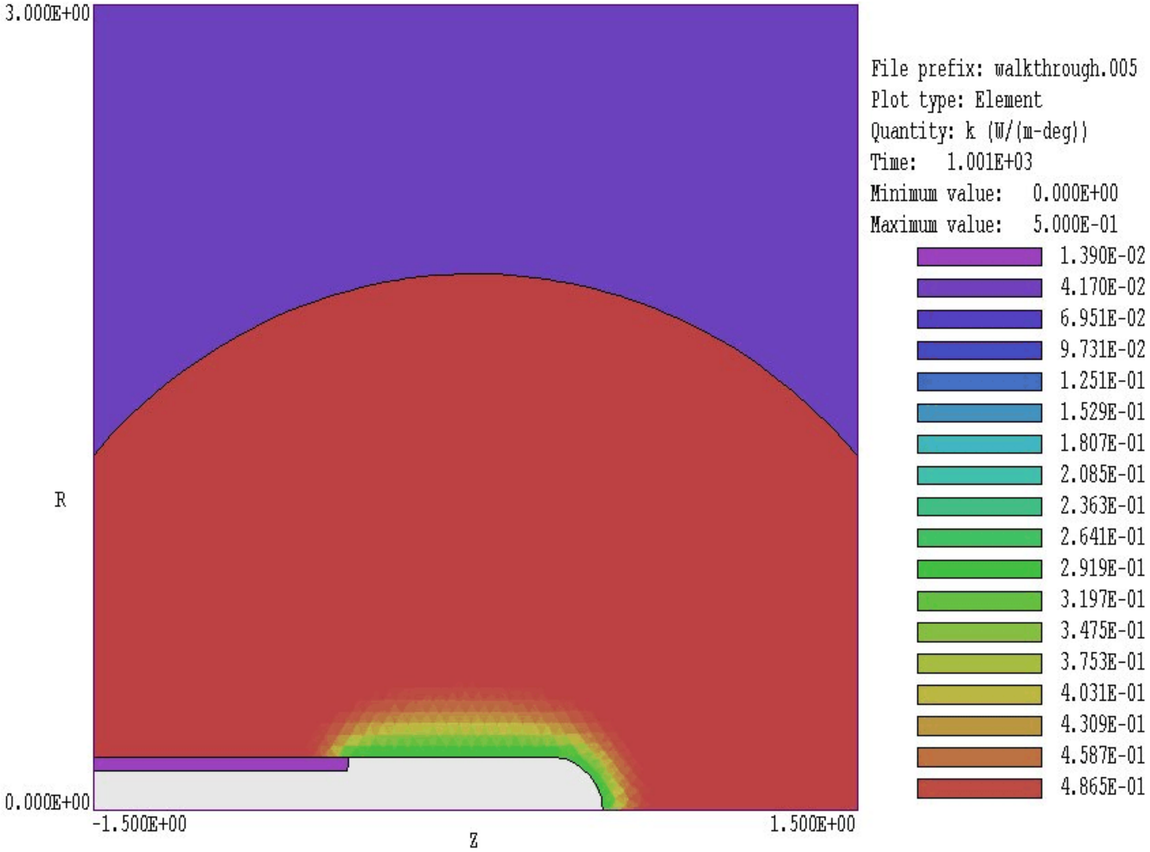


Figure 3. Color-coded element plot of thermal conductivity, `WALKTHROUGH` example.

Next run **TDiff** from **TC**. Pick *START RUN* in the *RUN* menu and choose the script file `WALKTHROUGH.TIN` that will control program operation. **TDiff** takes less than 20 seconds to complete the run, creating the following files:

`WALKTHROUGH.TLS` (Listing of run information for diagnostics)
`WALKTHROUGH.001` (Snapshot of the simulation at 200 s)
`WALKTHROUGH.002` (Snapshot of the simulation at 400 s)
`WALKTHROUGH.003` (Snapshot of the simulation at 600 s)
`WALKTHROUGH.004` (Snapshot of the simulation at 800 s)
`WALKTHROUGH.005` (Snapshot of the simulation at 1000 s)
`WALKTHROUGH.P01` (Time variations of thermal quantities at the position z = 0.0 cm, r = 1.5 cm.

The solution is complete. Before proceeding to analyses we shall take a look at some components of the simulation. In the *FILE* menu pick *EDIT FILE* and load `WALKTHROUGH.CND`. The contents of this file (shown in

Table 1) define a temperature-dependent conductivity. The value of *k* is 0.5 W/°C-m at low temperatures and drops to 0.3 W/°C-m in the range 60 °C to 90 °C. In a biological application the difference could represent chemical changes in tissue at increasing temperature. Next, click on *EDIT INPUT FILES* in the *FILE* menu and pick *WALKTHROUGH.TIN*. The contents are illustrated in Table 2 with added line numbers. At this point, we will not try to understand all the commands in detail but simply point out some features of the script.

Table 1. Definition of temperature-dependent conductivity – contents of the file WALKTHROUGH.CND

```
* Walkthrough example, decreasing thermal
conductivity
  0.0000   0.5000
 20.0000   0.5000
 40.0000   0.5000
 59.0000   0.5000
 60.0000   0.5000
 62.0000   0.4978
 64.0000   0.4914
 66.0000   0.4809
 68.0000   0.4669
 70.0000   0.4500
 72.0000   0.4309
 74.0000   0.4105
 76.0000   0.3895
 78.0000   0.3691
 80.0000   0.3500
 82.0000   0.3331
 84.0000   0.3191
 86.0000   0.3086
 88.0000   0.3022
 90.0000   0.3000
 92.0000   0.3000
 95.0000   0.3000
100.0000   0.3000
ENDFILE
```

Table 2. Control script WALKTHROUGH.TIN

```
01: SET MODE TVar

02: SET DUNIT 100.0
03: SET GEOMETRY Cylin
04: SET SAFETY 5.0
05: SET NCHECK 20
06: SET TMAX 1001.0
07: SET DTMAX 0.50
08: SET DTMIN 0.01

09: REGION(1) COND = 0.05
10: REGION(1) DENS = 1100.0
11: REGION(1) CP = 5000.0
12: REGION(1) INIT = 37.0
13: REGION(2) COND TABLE WALKTHROUGH.CND
14: REGION(2) DENS = 1100.0
15: REGION(2) CP = 5000.0
16: REGION(2) INIT = 37.0
17: REGION(3) COND = 0.010
18: REGION(3) DENS = 800.0
19: REGION(3) CP = 2500.0
20: REGION(3) INIT = 0.0
21: REGION(4) FIXED TABLE WALKTHROUGH.TMP
22: REGION(5) FIXED 37.0

23: DIAG DTIME 200.0
24: DIAG HISTORY 0.0 1.5

25: ENDFILE
```

Line 01 is required input. In this case it specifies that the run addresses a dynamic solution. Lines 02 through 08 are control commands. Line 02 indicates that coordinates in the **Mesh** `MOU` file are in centimeters while Line 03 specifies that the solution has cylindrical symmetry. The commands of Lines 04 through 08 control the adjustable time step (Section 9). Lines 09 through 22 set material properties for regions of the solution space (thermal conductivity, mass density, specific heat and initial temperature) . Figure 2 shows the numbers associated with regions. Line 13 states that the thermal conductivity of the spherical region (Region 2) should be determined from the tabular file `WALKTHROUGH.CND`. Line 21 states that the time-dependent temperature of the probe (Region 4) should be determined from the file `WALKTHROUGH.TMP`.The final two commands control code diagnostics.

The command of Line 23 specifies data dumps of complete spatial information at intervals of 200 seconds. Line 24 sets a temperature probe at the position $z = 0.0$ cm, $r = 1.5$ cm.

Next run **TView** and click on *LOAD FIRST SOLUTION FILE* in the *FILE* menu. Pick the file `WALKTHROUGH.001.` The program loads the first data dump (created at 200 seconds) and displays a contour plot of temperature. To see the final temperature distribution at 1000 seconds, click on *LOAD SOLUTION FILE NUMBER* and enter 5 in the dialog. You can use the *ZOOM WINDOW* command in the *SPATIAL PLOTS* menu top reconstruct the plot of Fig. 2. Next, pick *PLOT TYPE* in the *SPATIAL PLOTS* menu and specify *ELEMENT*. In this plot the temperature is displayed through color-coded elements. Pick *PLOT QUANTITY* in the *SPATIAL PLOTS* menu and specify k (thermal conductivity). You should see the plot of Fig. 3. Note that a sheath of low conductivity has formed around the probe inhibiting thermal conduction.

In addition to plots, **TView** can perform several quantitative calculations. In the *FILE* menu click on *OPEN DATA LISTING FILE* and accept the default file prefix `WALKTHROUGH`. The results of all subsequent operations will be recorded in the ASCII file..Next, click on *REGION PROPERTIES* in the *ANALYSIS* menu. Move the mouse to a point inside the probe and left-click. **TView** performs an analysis of the probe volume and surface. Complete information is recorded in the data file and a summary given on the screen. After closing the data file with the *CLOSE DATA LISTING FILE* command, you can inspect it with the *EDIT DATA FILE* command. The entry for the region analysis is:

```
 --- Region Information ---
Region No:   4
Volume:   5.059E-07 m3
AvgT:   9.500E+01 deg
PeakT:   9.500E+01 deg
ZPeak:   0.000E+00
RPeak:   0.000E+00
Flux:   1.356E+00 W
```

The quantity *Flux* is the integral of $k\nabla_n T$ over the region surface. An analysis of Region 2 (the surrounding sphere) gives *Flux* = 0.21391 W, a net influx of energy. In the steady state this figure should be close to zero because the power entering from the probe surface should equal the power lost through the boundary with Region 1. Therefore, Region 2 is still heating at $t = 1000$ seconds. To confirm this conclusion, we can look at

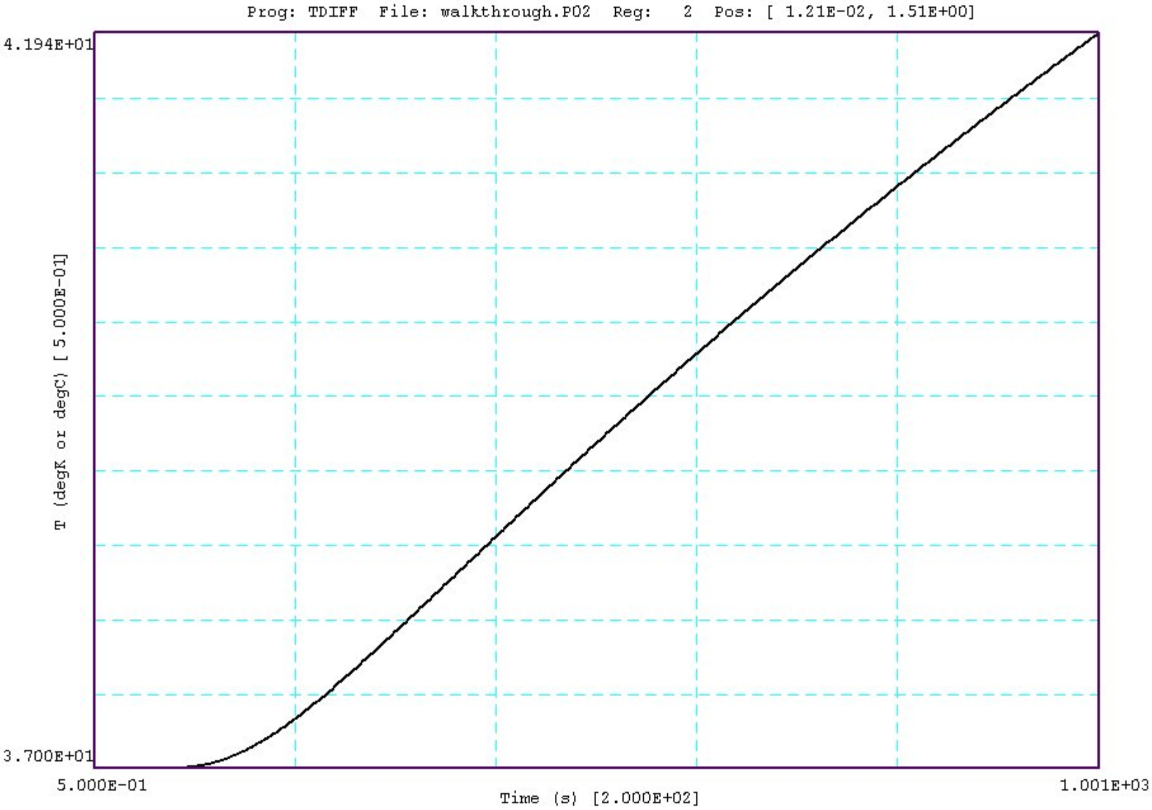


Figure 4. Plot of the time-variation of temperature at the probe position created by the **Probe** program.

the record of the probe at the position shown in Fig. 2 (colored triangle). From **TC** run **Probe**, click on *LOAD PROBE FILE* and choose `WALKTHROUGH.P01`. You should see the plot of Fig. 4. The temperature in the outer portion of Region 2 is still rising at t = 1000 seconds and will take several thousand seconds to approach an equilibrium.

3. Physical units and conversion factors

All thermal quantities, time and length are in standard SI units. Coordinates entered in alternative units must be converted to meters using the `DUnit` command. The quantity *DUnit* is the number of unit lengths per m. For example, if dimensions are in cm, set *DUnit* = 100.0.

Length: meters
Time: seconds
Temperature: °K or °C
Material density: kg/m^3
Specific heat: J/(°K-kg)
Thermal conductivity: J/(m-s-°K)
Thermal sources: W/m^3

Conversion Factors			
Quantity	**From**	**To**	**Multiply by**
Specific heat	cal/(°K-gm)	J/(°K-Kg)	4180.0
Density	gm/cm^3	kg/m^3	1000.0
Thermal conductivity	cal/(cm-s-°K)	J/(m-s-°K)	418.0

4. Organizing runs

The **TDiff** package consists of a program that computes the physical solution (`TDIFF.EXE`), a dedicated post-processor for thermal analyses (`TVIEW.EXE`) and a utility program to plot history files from any Field Precision program that generates initial value solutions (`PROBE.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 **TView** 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.

TDiff requires 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.

Runs with time-dependent sources and applied temperatures or temperature-dependent conductivity or specific heat may require additional files to define the variations.

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.TIN` where `FPREFIX` is the same prefix as that of the mesh file. **TDiff** 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 **TDiff** run consists of three components that involve 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 properties of materials. Each run includes the following steps.

■ Prepare a **Mesh** input script 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.TIN`) using the *Create command file* menu entry in **TDiff** or your own text editor. The allowed file commands are described in this manual.

■ Optionally, prepare data files to define time and temperature variations. The ASCII files may consist of experimental data and may be created with a text editor or spreadsheet.

■ Run **TDiff** to create one or more output files. Files with names of the form `FPREFIX.001` are data dumps at specific times. The files are in ASCII format and contain information on the mesh geometry, the physical properties of regions, and values of computed quantities at the nodes. In thermal solutions the region properties are the thermal conductivity, mass density and specific heat. The computed quantities are the temperature at nodes and the conductivity and specific heat in elements. Data dumps can be analyzed with the **TView** program. Files with names of the form `FPREFIX.P01` are probe records of temperature at specific locations. The contents of these files can be plotted with the **Probe** program.

■ Analyze the solution using **TView**. or **Probe**. You can also transfer the information in data dumps or history files to your own analysis programs.

5. 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, Forward slash,
Colon, Tab, Equal sign
```

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 accepts commands in any order. The following example illustrates a complete control file for **TDiff**:

```
* Run DUAL_PRB
SET MODE TVar
SET SAFETY 10.0
SET NCHECK 25
SET TMAX 10.0
SET DTMIN 1.0E-4
SET DTMAX 1.0E-2
SET DUNIT 100.0
SET GEOMETRY Rect
* Region 1: Left conductive medium
REGION 1 COND   90.0
REGION 1 DENS 1000.0
REGION 1 CP   3500.0
* Region 2: Right conductive medium
REGION 2 COND  150.0
REGION 2 DENS 1000.0
REGION 2 CP   3500.0
* Region 3: Right probe, delayed
REGION 3 FIXED TABLE RIGHT.TMP
* Region 4: Left probe
REGION 4 FIXED TABLE LEFT.TMP
* Region 5: Outer boundary
REGION 5 FIXED 0.0
* Diagnostics
DIAG DTIME 1.0
* Thermocouple in center
DIAG HISTORY 0.0 0.0
ENDFILE
```

In steady-state and dynamic solutions 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 type.

`REGION` commands set physical properties associated with elements and vertices. They 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. The string `Keyword` specifies the physical property. The keyword is followed by one or more numbers. As an example, the command

```
   REGION 2 Fixed 37.0
```

sets nodes with region number 2 to a fixed temperature of 37.0 °C. In **TDiff** several `REGION` commands may refer to the same mesh volume and set different properties.

There is an additional command in dynamic solutions to control data recording. Commands that begin with the keyword `DIAG` are used to specify times for data dumps and locations for probes.

6. Commands for steady-state solutions

TDiff performs a steady-state calculation when the command *SET MODE STAT* appears in the command script. In this case, the program solves the static thermal transport equation

$$\nabla \cdot (k\ \nabla T) = -Q. \tag{1}$$

In the equation k is the thermal conductivity (W/°C-m), T is the temperature (°C) and Q is an optional thermal source (W/m^3). The material density ρ and specific heat C_p do not enter into a steady-state solution. Equation (1) has the form of Poisson's equation. Therefore, **TDiff** uses the same solution techniques applied in the **EStat** program for electrostatics.

Steady-state solutions are straightforward and the associated command set is compact. Specified temperatures and thermal sources have fixed values that do not vary in time. The only advanced technique is the assignment of temperature-dependent k discussed in the next section. The first set of commands control program operation.

SET DUNIT 1.0E4

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

Spatial quantities recorded in the output files `FPrefex.001` are always in meters. The value of *DUnit* is passed to **TView**. In post-processor analyses, spatial quantities in graphs and listing files are always scaled to the **Mesh** units. For example, if the **Mesh** dimensions are in cm and *DUnit* = 100.0 in the **TDiff** command file, the output of spatial quantities in **TView** is in cm.

SET GEOMETRY CYLIN

TDiff handles problems in rectangular or cylindrical geometries. Rectangular systems have variations in *x* and *y* with infinite length in *z*. Cylindrical systems have variations in *r* and *z* with azimuthal symmetry. The command options are *RECT* and *CYLIN*. In cylindrical solutions the program takes the *z*-axis along the **Mesh** *x* direction and the *r*-axis along *y*. In this case the program issues an error message if any node has a *y* coordinate less than 0.0. **TView** automatically adjust calculations and labels to represent the two geometries.

SET OMEGA 1.95

This command sets the *over-relaxation factor* for the Poisson-equation solution, a number in the range 1.0 to 2.0. Generally, high values of *Omega* give faster convergence. Reduce the over-relaxation factor if the solution fails to converge. If this command does not appear, the program uses the Chebyshev acceleration prescription to pick optimal values of the over-relaxation factor.

SET MAXCYCLE 200

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

SET RESTARGET 5.0E-6

TDiff keeps track of the relative error in the temperature during the relaxation 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 NCHECK 20

The operations to calculate *ResTarget* take time; therefore, **TDiff** 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.

The next four commands set the material properties of regions. They start with the word *Region* and have same format.

Region   RegNo   Keyword   Value

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

REGION 4 FIXED 0.0

Sets the region to a temperature that is fixed during the relaxation solution. The parameters following the *Region* keyword are 1) the region number, 2) the keyword *Fixed*, and 3) the temperature in ºC.

The remaining commands refer to thermally conductive materials. Multiple commands may be necessary for a full region definition.

REGION 3 INIT 1000.0

Sets an initial temperature for a conductive region in ºC. The default temperature value at all variable points is 0.0 °C.

REGION 2 COND 3.45

Sets the region thermal conductivity in W/(ºC-m).

REGION 4 SOURCE -4.5E6

Sets a heat source or sink uniformly distributed over the region volume. The units are W/cm^3.

7. Temperature-dependent thermal conductivity

One of the most powerful features of **TDiff** is the ability to handle time-dependent sources and temperature-dependent thermal properties. For example, in a simulation of a biological system the conductivity may decrease as tissue cooks or desiccates. In this case, the Poisson equation assume the non-linear form:

$$\nabla \cdot [k(T)\ \nabla T] = -Q. \tag{2}$$

It is important to realize that non-linear equations do not always have a unique solution. For example, consider a case where a region with a thermal source *Q* has a conductivity *k* that decreases with temperature. A solution may be possible at low values of *Q*, but at higher values an instability may occur. The temperature rises with decreasing conductivity while *k* drops with increasing *T*. The end result is a divergent temperature. In the physical world the system disassembles. In the numerical world the program generates invalid real numbers and crashes. In the example the allowed range of *Q* depends on the functional dependence *k*(*T*). There is no cut-and-dried method to predict stability.

TDiff gives you considerable flexibility to model different materials. The key to this flexibility is program input through tabular functions. A tabular function is a text file consisting of up to 256 data lines containing values for an independent and dependent variable. For example in the definition of a temperature-dependent thermal conductivity, each line has a temperature followed by the corresponding conductivity. You can prepare tabular function files with a text editor or spreadsheet. You can also use published data or digitized experimental traces. The example below shows a tabular function generated with a spreadsheet. The intention was to model a sharp transition in thermal conductivity between two values. Note that the file syntax conforms to the same rules as **TriComp** scripts. The free-form parser accepts real numbers in any format with a choice of delimiters. You can add documenting comment lines starting with an asterisk. The end of the data is marked with the *ENDFILE* command. You can also add any text after *ENDFILE*.

Although the lines of the example are ordered by increasing value of the independent variable, this ordering is not required. **TDiff** sorts the list before use and records the final order in the file `FPREFIX.TLS`. Furthermore, the independent variable intervals need not be uniform. Note that the entries in the example are clustered near the transition temperature. The maximum number of tables for all purposes is 32. A table requires a minimum of 5 entries. ***You must ensure that the tabular functions extend over the full range of temperature that will be***

encountered in the solution. The interpolation routines for *k* return 0.0 for values of the independent variable that are out of the table range. Zero conductivity values can lead to non-physical results or a program crash. Be sure that the table extends below *TInit* and well above the maximum temperature anticipated in the region.

```
* TWAVE.CND
* T(deg-C) K(J/(m-s-degC)
* ===================
     0.0   10.00
   200.0   10.00
   290.0   10.00
     3.000E+02      1.000E+01
     3.025E+02      1.019E+01
     3.050E+02      1.076E+01
     3.075E+02      1.177E+01
     3.100E+02      1.331E+01
     3.125E+02      1.551E+01
     3.150E+02      1.854E+01
     3.175E+02      2.266E+01
     3.200E+02      2.815E+01
     3.225E+02      3.538E+01
     3.250E+02      4.472E+01
     3.275E+02      5.653E+01
     3.300E+02      7.105E+01
     3.325E+02      8.828E+01
     3.350E+02      1.079E+02
     3.375E+02      1.290E+02
     3.400E+02      1.502E+02
     3.425E+02      1.699E+02
     3.450E+02      1.859E+02
     3.475E+02      1.963E+02
     3.500E+02      2.000E+02
   360.0  200.0
   400.0  200.0
   500.0  200.0
  1000.0  200.0
  2000.0  200.0
  4000.0  200.0
ENDFILE
```

TDiff 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 dependent quantity and its first derivative should vary smoothly over the table range. The example above illustrates the approach. Note how the transition between the values 10.0 and 200.0 is represented by a continuous variation. You can check the fidelity of the interpolations by inspecting the file `FPREFIX.TLS`. After listing the sorted table, **TDiff** records a sample set of interpolated values.

Temperature-dependent conductivity can be defined by a modified form of the *COND* command.

REGION 2 COND TABLE ALUM.CND

Determines the thermal conductivity of the region as a function of temperature from a tabular function file. The first three parameters are the region number and the keywords *COND* and *TABLE*. The final parameter is the full name of a data file in the working directory. The file may contain up to 256 data lines with the form

```
Temperature(°C,°K )    Conductivity (J/m-s-°K)
```

SET AVG 0.35 20

In a steady-state solution **TDiff** adjusts values of *k* in variable elements during the relaxation process. For a stable system the default adjustment parameters in the program are usually sufficient to achieve convergence. For difficult cases where *k* may change by a large factor over a small temperature range, you can use the *AVG* command to change the adjustment factors. The first real-number parameter *MatAvg* is a smoothing factor to prevent oscillations during the relaxation. Enter a value from 0.0 to 1.0. The conductivity in an element is adjusted according to

$$k_{new} \;=\; (1.0 - MatAvg)\; k_{old} \;+\; MatAvg\;\; k(T_{new}) \;. \qquad \textbf{(3)}$$

Small values ($MatAvg \ll 1$) give strong smoothing that may be required for problem solutions. In this case, be sure to set *MaxCycle* high enough so that the material can reach equilibrium. The default value is *MatAvg* = 0.5. The second parameter *NAvg* (an integer) equals the number of relaxation cycles per material adjustment. The default value is *NAvg* = 10. Lower the value for solutions that converge rapidly.

8. Control of dynamic solutions

TDiff performs a dynamic calculation when the command *SET MODE TVAR* appears in the script. In this case, the program solves the thermal transport equation

$$\rho\ C_p\ \frac{\partial T}{\partial t} = \nabla \cdot (k\ \nabla T) + Q. \qquad \textbf{(4)}$$

In the equation ρ is the mass density (kg/m^3), C_p is the specific heat (J/kg-°C), *k* is the thermal conductivity (W/°C-m), *T* is the temperature (°C) and *Q* is an optional thermal source (W/m^3). Because of increased options the solution set for dynamic solutions is larger. This section reviews allowed `SET` commands.

SET SAFETY 35.0

Sets a safety factor for the automatic time step adjustment. A typical value is 20.0. Higher values improve accuracy at the expense of longer computation times. The default value is 50.0.

SET DTMIN 0.0001

Sets the minimum time interval, over-riding the automatic step selection. Enter the value in seconds. The default value is 1.0×10^{-9} s.

SET DTMAX 0.25

Sets the maximum time interval, over-riding the automatic step selection. Enter the value in seconds. The default is infinity.

SET NCHECK 10

Sets the number of time increments between a recalculation of the automatic time step routine. The default value is 5.

SET TMAX 100.0

Sets the stop time for the run. Enter the value in seconds. All runs begin at $t = 0.0$. The default is 0.0.

SET NSTEP 5000

Sets the maximum number of time steps in the solution. The program stops at either *TMax* or *NStep*, depending on which occurs first. This command can prevent solutions controlled by a batch file from running out of hand. The default is 100,000.

SET GEOMETRY Cylin

See Section 6.

SET DUNIT 100.0

See Section 6.

9. Choosing a time step

In the dynamic mode **TDiff** follows thermal transport by advancing in small time increments. Because time scales may vary several orders of magnitude in numerical solutions of diffusion problems, it is essential to use a variable time step. For example, consider setting an initial temperature on a body in a surrounding medium at ambient temperature. Initially, the full temperature drop occurs across a single mesh element. If the solution space is about 100 elements on a side, the initial time step must be 10,000 times shorter than the time step necessary to model thermal diffusion in the full solution space. To perform this large adjustment, **TDiff** has a routine that estimates the best time step. The procedure relies on a calculation of the root-mean-squared value for the second derivative of temperature over the full solution region. Although the automatic time step feature generally works well, some user intervention may be necessary because of the wide diversity of problems that **TDiff** handles. There are three *SET* commands that allow you to modify the time step: *SAFETY, DTMIN* and *DTMAX*. We will discuss each in turn.

SET SAFETY 15.0

The automatic time-step routine calculates the maximum possible time increment and then divides by the factor *Safety*. Generally, *Safety* should be about 20.0. Higher values improve accuracy but lengthen the computation time. The numerical methods used in **TDiff** are stable for all choices of time step.

SET DTMIN 5.0E-7

The automatic time step feature of **TDiff** can cope with difficult solutions where objects have initial temperatures different from their immediate surroundings. The program gives results that are in excellent agreement with analytic solutions for relaxation of an initial temperature distribution. Depending on the geometry, **TDiff** may have to use extremely short time steps to achieve accuracy early in time. Sometimes, details of the initial local temperature relaxation are not important. To speed calculations, you can over-ride automatic time selection by setting a minimum time step with the `DtMin` command. Once the estimated time step rises above *DtMin*, the automatic routine takes over.

The `DtMin` command is also important for solutions with thermal sources where all nodes start at the same temperature. Because the second derivative of temperature is zero everywhere, there is no way for **TDiff** to calculate a time step. In this case, the program assigns a very small default increment. You can set a better value using `DtMin`. In picking a time step, note that a good run should occupy about 1000 to 3000 steps.

SET DTMAX 1.0E-3

Occasionally, the automatic routine in **TDiff** may pick a time step that is too long for good accuracy. This may happen if there is a local temperature variation in a large solution volume at approximately uniform temperature. In this case, you can clamp the maximum value of time increment using the `DtMax` command. Again, a good solution should run 1000 to 3000 steps.

Ironically, one disadvantage of the stable numerical procedure used in **TDiff** is that it always gives an answer, even if quite inaccurate. You can use the `DtMax` command to check the accuracy of important solutions. First, run the problem using automatic time step selection. Note the final time step and use **TView** to find temperatures at critical positions. Run the solution again, setting *DtMax* to a fraction (0.25 - 0.50) of the final time. Check for significant changes in the predicted temperature.

10. Material commands in dynamic solutions

The following `REGION` commands set the physical properties of materials in dynamic solutions. We shall first cover simple commands and then consider options for time and temperature variations.

REGION 4 FIXED 0.0

Sets the region to a fixed temperature that does not change during the solution. The parameters following the *REGION* keyword are 1) the region number, 2) the keyword *FIXED*, and 3) the temperature in ºC or ºK.

Other commands refer to thermally-conductive materials. Several commands may be necessary for a full region definition.

REGION 6 DENS 5.67

Sets the mass density of the region in kg/m^3.

REGION 3 INIT 1000.0

Sets an initial temperature for a conductive region in ºC. The default temperature value at all variable points is 0.0.

REGION 2 COND 3.45

Sets the region thermal conductivity in J/(s-ºC-m).

REGION 2 CP 3.45

Sets the region specific heat in J/(kg-ºC)

REGION 4 SOURCE -4.5E6

Sets a heat source or sink uniformly distributed over the region volume. The units are W/cm^3.

The following commands apply time or temperature dependance to materials. All commands require a prepared tabular function file in the working directory.

REGION 4 FIXED TABLE PTEMP.TMP

Sets the temperature of a fixed region to a prescribed function of time rather than to a constant value. The numerical values are contained in the named file in the form

```
Time (seconds)    Temperature (°C)
```

Note that the values of the independent variable must extend from t = 0.0 to $t > t_{max}$. The interpolation routines return $T(t) = 0.0$ for values of t out of the table range.

The following commands apply to thermally conductive regions.

REGION 2 CP TABLE M2.SPH

Determines the specific heat of the region as a function of temperature from the named tabular function. The file data lines have the form

```
Temperature(°C,°K )  Specific heat(J/kg-s-°K)
```

Note the values of C_p for elements in the region are determined from the same table. This does not mean that all triangles have the same specific heat because the temperature may vary over the region.

REGION 2 COND TABLE ALUM.CND

Determines the thermal conductivity of the region as a function of temperature from the named tabular function. The file data lines have the form

```
Temperature(°C,°K )    Conductivity (J/m-s-°K)
```

REGION 4 SOURCE TABLE EVAP.SRC

Adds a time-dependent volumetric heat source or sink uniformly distributed over a region. The source variation is contained in the named data files. The data lines have the format

```
Time (seconds)      Source(W/m³)
```

11. Diagnostic commands for dynamic solutions

Diagnostic commands that begin with the keyword *DIAG* control data output from the program from the program in the dynamic mode. **TDiff** produces three types of data, all in ASCII format.

- ■ From 1 to 999 data dump files with the name `FPREFIX.001`, `FPREFIX.002`, ... Each file contains a complete record of spatial information (the mesh geometry and temperature distribution) at a specified time. These files are usually analyzed with the **TView** program.

- ■ A listing file with the name `FPREFIX.TLS` that can be inspected with a text editor. This file contains a variety of information about the **TDiff** run used mainly to check the validity of input parameters and to diagnose problems.

- ■ Optional history files with the names of the form `FPREFIX.P01, FPREFIX.P02`,.... Here, the temperature at one or more probe locations is recorded at each time step of the solution. The information can be inspected with a text editor or the Field Precision utility program **Probe**.

The first three commands control the times for spatial data dumps. The last command sets probe positions for the history file.

DIAG DTIME 0.2

For many runs a convenient way to make spatial data dumps is at uniform time intervals. The `DTime` command sets the approximate interval (in seconds) between dumps. Because **TDiff** uses variable time steps, it is not possible to ensure that the dump occurs at an exact time. Instead, the program writes data as soon as possible after passing the desired time. For example, **TDiff** will write dump 010 when $t \geq$ 10**DTime*. The default value of *DTime* is infinity.

DIAG SETTIME 3.45

Sometimes you may want to inspect data at certain critical times or at the end of a run. For this situation, you can set up to 100 time markers using multiple *DIAG SETTIME* commands. **TDiff** will write data as soon as possible after passing each marked time. Note that multiple *SETTIME* commands must be in order of increasing time.

DIAG NSTEP 100

The *NSTEP* option sets the number of time steps per data dump.

Note that the three commands for data dumps can work concurrently. Be careful in setting the parameters. You may generate enough data to fill a hard disk.

DIAG HISTORY 5.00 9.85

TDiff creates probe files with names of the form `FPREFIX.P01`, `FPREFIX.P02`,... when it detects a *DIAG HISTORY* command. The parameters in the command give the spatial location of a probe where the time and temperature should be recorded. The location is given as (*x*,*y*) or (*z*,*r*). Enter the coordinates in the units set by DUnit. **TDiff** divides the values by *DUnit* to convert them to meters. The program locates the closest mesh node and records the temperature at that point. You can define up to 20 probes with multiple *DIAG HISTORY* commands. The history file contains information on the actual probe locations.

12. Avoiding instabilities in dynamic solutions

TDiff uses the time-centered Dufort-Frankiel method to advance the diffusion equation in time. The method is stable for all choices of time step on a simple rectangular finite-difference mesh. A numerical instability may occur if the method is applied on conformal meshes that have triangular elements that are short in *y* or *r* directions. For a discussion of the theory and conditions of the instability, see S. Humphries, **Field Solutions on Computers** (CRC Press, Boca Raton, 1997), Sect. 12.4. We have added an automatic correction feature in Version 5.0 of **Mesh** to eliminate the instability in diffusive programs like **TDiff** and **Pulse**. To employ the correction, you must include the command

DCORRECT

(for *diffusive correction*) in the **GLOBAL** section of the **Mesh** input (`MIN`) file. Figure 5 shows the effect for a mesh with elements where $\Delta y = 0.4\Delta x$. In the top plot with no correction most of the triangles have an internal angle greater than 90°, the condition for an instability. With the

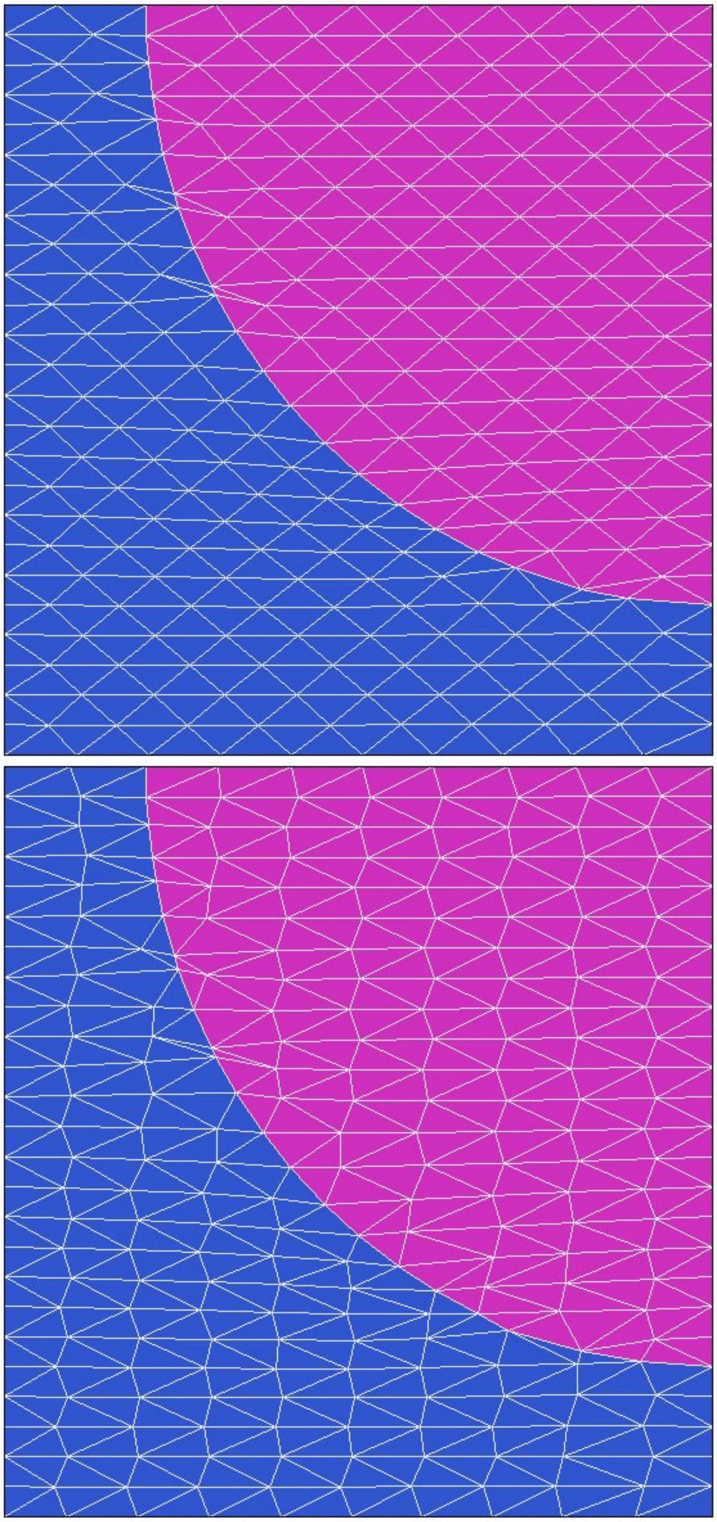


Figure 5. Effect of the *DCORRECT* command.

diffusive correction (bottom plot), **Mesh** automatically adjust the shapes of mesh elements so that there are no contiguous blocks of elements with obtuse internal angles. The following global section was used for the bottom plot:

```
GLOBAL
  XMESH
       0.0  5.0  0.5
    END
  YMESH
       0.0  5.0  0.2
    END
    DCORRECT
END
```

As a rule, always include *DCORRECT* in **Mesh** input files that will be used for dynamic **TDiff** solutions. Mesh correction is not necessary for steady-state solutions.

13. Boundary conditions in thermal solutions

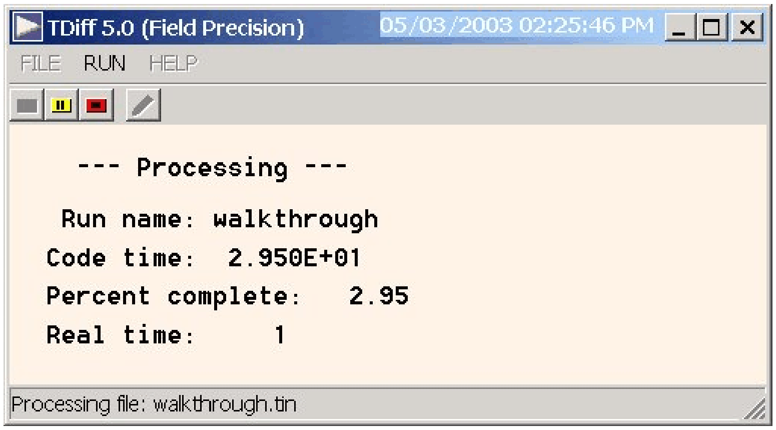
The *boundary* is the outer edge of the solution volume. It may or may not correspond to the edges of the solution rectangle. For thermal problems, vertices on the boundary can assume one of two conditions.

Dirichlet

Dirichlet boundary points have a fixed temperature (either constant or specified time variation) that does not depend on the temperature at neighboring nodes. Thermal flux lines are normal to such a surface.

Neumann

A Neumann boundary is one where the normal derivative of the temperature is specified. The boundaries in **TDiff** are limited to the special case $\partial T/\partial n = 0$. The special Neumann condition implies that the thermal flux is parallel to the boundary (*i.e.*, ideal thermal insulator). 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.



```
   --- Processing ---

  Run name: walkthrough
 Code time:  2.950E+01
 Percent complete:   2.95
 Real time:      1
```

Processing file: walkthrough.tin

Figure 6. **TDiff** in the interactive mode.

14. Running TDiff interactively

The program `TDIFF.EXE` can run interactively in a Window (Figure 6). In this mode you can carry out several solutions in a session or leave the program while you work in other Windows applications.

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

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

Create command 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 **TDiff** 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 **TDiff**. 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.TIN`. The *Edit listing file* command displays files with names `FPREFIX.TLS`.

The *Run* menu has four commands.

Start run

Pick an input file (such as FPREFIX.TIN) 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.

15. Running EStat from the command prompt

To run **TDiff** in the background, go to the Command Prompt from Windows and log to the data directory that contains the required MOU and EIN files.. For example, suppose the input data files SWITCH.MOU and SWITCH.TIN are stored in \TRICOMP\BUFFER and that the program TDIFF.EXE is in the directory \TRICOMP. From \TRICOMP\BUFFER type

```
..\TDIFF SWITCH <Enter>
```

The program runs silently, writing detailed information in the listing file SWITCH.TLS. If the solution is successful, the program writes output files 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.TIN,...,SWT08.TIN` 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 ..\TDIFF.EXE SWT01
START ..\MESH.EXE SWT02
START ..\TDIFF.EXE SWT02
 ...
START ..\MESH.EXE SWT08
START ..\TDIFF.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.

16. TView file menu

TView is the interactive graphical analysis program for output data dumps from **TDiff**. **TView** 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.

Program units

Length: meters
Time: seconds
Temperature: ºK or ºC

Material density: kg/m^3
Specific heat: J/(ºK-kg)
Thermal conductivity: J/(m-s-ºK)
Thermal sources: W/m^3

Load first solution file

The *Load first solution file* command displays a dialog with a list of all data dump files with the suffix `001`. Changing the directory in the dialog changes the program working directory. Pick an available file and click *OK*. The program loads the first file in the series and updates the status bar. If data retrieval is successful, the *Spatial plots* and *Analysis* menus become active.

Load next solution file

Load the next solution in the current series. For example, if you are currently working with `RUNNAME.003`, TView loads `RUNNAME004`.

Load solution file number

Load a specific file in the current series. For example, if you are currently working with `RUNNAME.002` and specify the number 5, TView loads `RUNNAME.005` if it is available.

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 Sect. 20. Note that the script file must 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.

17. TView spatial plot menu

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

Plot type

TView 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.

Element. Elements of the solution space color-coded according to a computed quantity (such as the thermal flux electric field magnitude). You can add 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 vector quantity.

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 temperature 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.

TView supports the following plot quantities.

Contour plot quantities

Temperature, T
Thermal flux magnitude, $|\mathbf{F}|$
Thermal conductivity, k
Specific heat, C_p

Element plot quantities

Temperature, T
Thermal flux magnitude, $|\mathbf{F}|$
Thermal conductivity, k
Specific heat, C_p

Vector plot quantities

Thermal flux $\mathbf{F}$

Surface plot quantities

Temperature, T
Thermal flux magnitude, $|\mathbf{F}|$
Thermal flux, F_x or F_z
Thermal flux, F_y or F_r
Thermal conductivity, k
Specific heat, C_p

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 **TView**. 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 could 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, the post-processor must map the current quantity to a rectangular grid, performing 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* × *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.

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. 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

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

18. TView 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

TView employs a sophisticated interpolation technique that preserve discontinuities at material boundaries. The program gives correct values for thermal flux on both sides of 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 **TView**. 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 thermal 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 19). 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 choices are temperature (T), magnitude of the thermal flux (|**F**|), components of thermal flux (F_x-F_y or F_z-F_r), thermal conductivity (k) or specific heat (C_p).

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

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

Region properties

To see the physical properties associated with a region of the solution space, click the mouse close to any arc or line vector of the region. Partial results are shown on the screen and a complete analysis is included in the data file. **TView** shows the region volume, average and peak temperatures as well as the integral of thermal flux around the boundary.

Line integrals

Input for this command is similar to that for the *Line scan* command. Enter two points to define a line.**TView** calculates line integrals of the normal component of thermal flux. The sign of the quantity depends on the direction of the flux and the direction of the line integral. The quantity is positive if the cross product of the flux vector and a vector parallel to a line integral points in the positive z or θ directions.

Volume integrals

No input is needed for this command. TView automatically computes integrals of quantities over the full solution volume and over individual regions. Information is recorded on the screen or in a history file. **TView** calculates the following quantities.

1) Global and region volumes.

2) Volume averaged temperature in the full volume and in each region.

3) The location and value of the peak temperature in the visible solution volume and in each region.

Matrix file

TView can make matrix files of values to help you create your own analysis routines. Although information is available in the output file of the solution program, it is often 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.

19. TView 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 17. 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 **TView**. This command closes the scan plot and returns the program to the previous spatial plot.

20. TView script file commands

Script files to control **TView** 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.

TView 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 `TView.EXE` is in the directory `\TRICOMP.` From `\TRICOMP\BUFFER`, type

```
..\TView 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 Switch1.003

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 `SWITCH1.003`. 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 points 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, [x1,y1,x2,y2] or [z1,r1,z2,r2] (real).

Endfile

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

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

```
NSCAN 200
OPEN OUTPUT COMP
OPEN INPUT SWITCH01.001
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH02.002
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH03.003
SCAN 0.00 -50.00 0.00 50.00
OPEN INPUT SWITCH04.004
SCAN 0.00 -50.00 0.00 50.00
ENDFILE
```

21. Format of the TDiff output file

The **TDiff** output file `FPREFIX.001` 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 10 data lines:

```
--- Run parameters ---
XMin: -6.000000E-02
XMax:  6.000000E-02
KMax:  141
YMin:  0.000000E+00
YMax:  6.000000E-02
LMax:   76
DUnit:  1.000000E+02
NReg:    5
ICylin:  1
Time:  2.005000E+02
```

Lines 2 and 3 list x_{min} and x_{max}, the limits along 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). Finally, Line 11 gives the simulation time for the data dump.

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

```
--- Vertices ---
  k    l  RgNo RgUp RgDn       x             y             T
===================================================================
   1    1    4    4    0 -6.000000E-02  0.000000E+00  9.500000E+01
   2    1    4    4    0 -5.920272E-02  0.000000E+00  9.500000E+01
   3    1    4    4    0 -5.820767E-02  0.000000E+00  9.500000E+01
   4    1    4    4    0 -5.718518E-02  0.000000E+00  9.500000E+01
   5    1    4    4    0 -5.621330E-02  0.000000E+00  9.500000E+01
   6    1    4    4    0 -5.519550E-02  0.000000E+00  9.500000E+01
...

            CondUp        CondDn        CpUp          CpDn
      =========================================================
        0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
   ...
```

Each data line contains the following quantities:

- The indices of the node (*K*, *L*)

- The region number of the node (*RgNo*) and region numbers for two associated elements (*RgUp* and *RgDn*). The upper element lies above 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 temperature at the node (*T*)

- The thermal conductivity of the upper and lower elements (*CondUp*, *CondDn*)

- The specific heat of the upper and lower elements (*CpUp*, *CpDn*)

The region section consists of four title lines following by *NReg* data lines, one for each region:

```
--- Regions ---
RegNo Fix                          Conductivity    Density      Specific heat
=========================================================================
   1    0    0    0    0    0  5.000000E-02  1.100000E+03  5.000000E+03
   2    0    0    0    0    0  0.000000E+00  1.100000E+03  5.000000E+03
   3    0    0    0    0    0  1.000000E-02  8.000000E+02  2.500000E+03
   4    1    0    0    0    0  0.000000E+00  0.000000E+00  0.000000E+00
   5    1    0    0    0    0  0.000000E+00  0.000000E+00  0.000000E+00
...
```

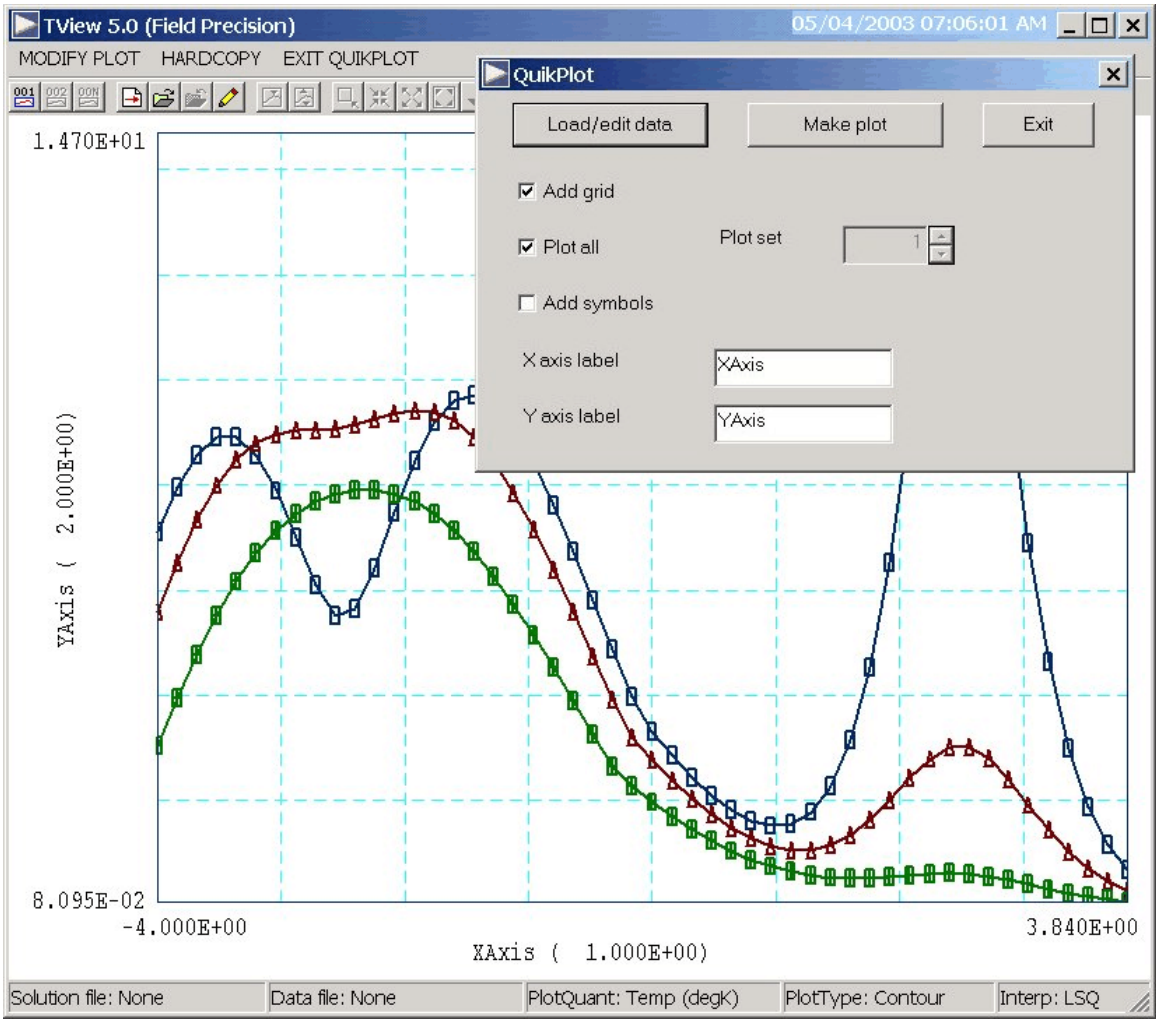
A one in the second column designates a fixed-temperature region (electrode). The other integer quantities and real quantities following *Specific heat* are not used in **TDiff**.

21. TView 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 thermal conductivity. Click on *QUIKPLOT* in the *TOOLS* menu to bring up a dialog (Fig. 7). 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.



☑ Add grid

☑ Plot all

Plot set 1

☐ Add symbols

X axis label XAxis

Y axis label YAxis

Solution file: None | Data file: None | PlotQuant: Temp (degK) | PlotType: Contour | Interp: LSQ

Figure 7. Control dialog superimposed on **QuikPlot** screen.

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 **TView**.