VF-02-04-B2

OPERA-3D REFERENCE MANUAL

Vector Fields Limited 24 Bankside Kidlington Oxford OX5 1JE England

Copyright © 1999-2003 by Vector Fields Limited, England

This document was prepared using Adobe[®] FrameMaker[®].

UNIX is a trademark of X/Open Company Ltd. HPGL is a trademark of Hewlett-Packard Incorporated PostScript is a trademark of Adobe Systems Incorporated Windows 98/NT/2000/XP are trademarks of Microsoft Corporation X Window System is a trademark of Massachusetts Institute of Technology I-DEAS is a trademark of Structural Dynamics Reseach Corporation OpenGL is a trademark of Silicon Graphics Corporation

All other brand or product names are trademarks or registered trademarks of their respective companies or organisations.

CONTENTS

Chapter 1 System Overview

Introduction	1-1
Supported Analysis Programs	1-2
Program Limits	1-4

Chapter 2 User Interface

Introduction	2-1
Keyboard Input	2-2
Output Files	2-3
Commands and Parameters	2-4
The Help Character (!)	2-5
Parameter Assignment	2-6
Parameter Values	2-8
Command Interpreter Errors	2-15
Confidence Level	2-16
Prompted free format Input	2-18
Built-in Commands	2-19
Command Separator and Comments	2-33
Euler Angles	2-34

Chapter 3

OPERA-3d Geometric Modeller

Introduction	 3-	1

Geometric Modelling	. 3-3
Summary of Menus and Commands	3-15
The ANALYSISDATA Command	3-23
The ARC Command	3-27
The BEDSTEAD Command	3-29
The BEND Command	3-31
The BHDATA Command	3-32
The BLEND Command	3-35
The BLOCK Command	3-36
The BOUNDARY Command	3-37
The BRICK8 Command	3-41
The BRICK20 Command	3-44
The CELLDATA Command	3-47
The CHECK Command	3-49
The CIRCUIT Command	3-51
The CLEAR Command	3-55
The COLOUR Command	3-56
The COMBINE Command	3-58
The COMMENT Command	3-60
The CONDUCTOR Command	3-61
The CONTOUR Command	3-63
The CYLINDER Command	3-65
The DBCASEDATA Command	3-66
The DELETE Command	3-67
The DRIVE Command	3-68
The EDGEDATA Command	3-71
The END Command	3-72
The EXPORT Command	3-73
The FACEDATA Command	3-74
The FILL Command	3-77
The FILTER Command	3-80
The FITTEDCPE Command	3-82
The GUIOPTIONS Command	3-84
The HELICALEND Command	3-85
The HIDE Command	3-88
The HISTORY Command	3-89
The IMPORT Command	3-90
The INSERTOP2FILE Command	3-91

The ABEL Command	3-93
The LCS Command	3-94
The LIST Command	3-95
The LOAD Command	3-96
The LOFT Command	3-98
The MATERIALS Command	-100
The MESH Command	-104
The MODEL Command	-107
The MORPH Command	-109
The MOUSE Command	-110
The PERIODICITY Command	-111
The PICK Command	-114
The PICTURE Command	-117
The PRECISIONDATA Command	-118
The PRINT Command	-119
The PRISM Command	-120
The RACETRACK Command	-121
The REDO Command	-123
The RENAME Command	-124
The SAVE Command	-125
The SELECT Command	-127
The SOLENOID Command	-132
The SOLVERS Command	-135
The SPHERE Command	-137
The STRAIGHT Command	-138
The STRETCH Command	-140
The SWEEP Command	-141
The TANGENTIALCPE Command	-144
The THREED Command	-147
The TITLE Command	-149
The TORUS Command	-151
The TRANSFORM Command	-152
The TWIST Command	-155
The UNDO Command	-156
The VARIABLE Command	-157
The VECTOR Command	-159
The VERTEXDATA Command	-161
The VOLUME Command	-162

The WCS Command		3-165
The WINDOW Command	1	3-166

Chapter 4 OPERA-3d Pre Processor

Introduction	-1
The Graphical User Interface 4-	-3
Pre Processor Quick Reference Guide 4-1	10
The BHDATA Command 4-1	15
The CHECK Command	18
The CLEAR Command	21
The COLOUR Command	22
The CONDUCTOR Command	24
The DEFINE Command	19
The DEVICE Command	€
The DISPLAY Command) 6
The DUMP Command)3
The EDIT Command)8
The END Command 4-11	11
The EXTEND Command 4-11	12
The FILL Command 4-11	14
The HELP Command	16
The IDEAS Command	17
The LABEL Command	23
The MATERIALS Command	27
The MESH Command	31
The MODIFY Command	33
The READ Command	46
The REDEFINE Command	18
The SLIP Command	19
The SOLVERS Command 4-15	50
The TABLE Command	58
The THREED Command	59
The TITLE Command	74
The TRANSFORM Command	76
The WRITE Command 4-17	77

Chapter 5 OPERA-3d Post Processor

Introduction	. 5-1
Post Processor Quick Reference Guide	. 5-3
Field Component Evaluation	5-10
Post Processor Data Files	5-18
The ACTIVATE Command	5-22
The ARC Command	5-24
The ARITHMETIC Command	5-26
The BHDATA Command	5-27
The BODY Command	5-28
The CARTESIAN Command	5-31
The CIRCLE Command	5-33
The CLEAR Command	5-35
The COLOUR Command	5-36
The COMBINE Command	5-38
The CONDUCTOR Command	5-40
The END Command	5-46
The ENERGY Command	5-47
The FIT Command	5-51
The GRAPH Command	5-53
The GRID Command	5-56
The GUIOPTIONS Command	5-58
The IDEAS Command	5-60
Integration Commands	5-63
The INTEGRATE Command	5-65
The LINE Command	5-68
The LOAD Command	5-70
The MAP Command	5-72
The MOUSE Command	5-75
The PICTURE Command	5-76
The PLOT Command	5-77
The POINT Command	5-80
The POLAR Command	5-81
The PRINT Command	5-83
The SELECT Command	5-84
The SET Command	5-88
The SHOW Command	5-92

The SURFACE Command	5-94
The SYSVARIABLE Command	5-96
The TABLE Command	5-101
The THREED Command	5-104
The TITLE Command	5-107
The TRACK Command	5-109
The UNITS Command	5-112
The VIEW Command	5-115
The VOLUME Command	5-121

Chapter 1 System Overview

Introduction

OPERA-3d (an **OP**erating environment for **E**lectromagnetic **R**esearch and **A**nalysis) is the pre and post processing system for well known electromagnetics analysis programs including TOSCA, ELEKTRA, SCALA, CAR-MEN, SOPRANO and TEMPO.

Finite element discretisation forms the basis of the methods used in these analysis programs. This widely applicable technique for the solution of partial differential equations requires special enhancements to make it applicable to electromagnetic field calculations. Access to these features is supported by the OPERA-3d Geometric Modeller and pre processor. These programs provide facilities for the creation of finite element models, specification of complicated conductor geometry, definition of material characteristics including for example, non-linear and anisotropic descriptions and graphical displays for examination of the data.

Similarly, the OPERA-3d post processor provides facilities necessary for calculating electromagnetic fields. As well as displaying field quantities as graphs and contour maps, the OPERA-3d post processor can calculate and display many derived quantities and can plot particle trajectories through the calculated fields.

This Reference Manual describes in detail the OPERA-3d Modeller and the pre and post processors: Chapter 2 describes the common User Interface; Chapter 3 is a Reference Manual for the Modeller, Chapter 4 for the pre processor and Chapter 5 for the post processor.

The rest of this chapter gives summaries of the analysis programs which are supported. Full details of the analysis programs can be found in the User Guide.

Supported Analysis Programs

There are currently 6 supported analysis packages: TOSCA, ELEKTRA, SCALA, CARMEN, SOPRANO and TEMPO.

TOSCA

TOSCA solves non-linear magnetostatic or electrostatic field and current flow problems in three dimensions. It has been in use for many years, but is being continually improved to increase its accuracy and efficiency. TOSCA uses a formulation based on total and reduced scalar potentials, solved using finite elements.

ELEKTRA

ELEKTRA analyses time dependent electromagnetic fields, including the effects of eddy currents, in three dimensions. There are 3 analysis options: the time variation can be transient (TR), steady state ac (SS) or eddy currents can be induced in moving conductors with a specified linear or rotational velocity in the presence of a static field (VL).

SCALA

SCALA analyses electrostatic fields taking into account the effects of space charge created by beams of charged particles.

CARMEN

CARMEN analyses transient electromagnetic fields in rotating machines, with constant rotor speeds.

SOPRANO

SOPRANO analyses high frequency electromagnetic fields in three dimensions. There are two analysis options: steady state ac or eigenvalue extraction.

TEMPO

TEMPO analyses transient and steady state thermal fields arising as a result of electromagnetic heating and external heat sources. Heat generated from electromagnetic heating can be included from the electromagnetics analysis programs.

Program Limits

The OPERA-3d pre processor has a limit on the maximum numbers of entities which can be created and the Standard versions of the analysis programs have a limit on the size of finite element mesh which can be analysed. The Professional versions of the analysis programs and post processor will "grow" to accommodate the data up to the limit of available virtual memory (swap space).

OPERA-3d Program Limits				
		Standard	Profes	ssional
		Size 1	Size 2	Size 3
Pre processor internal database	entities ^a	200000	400000	2 000 000
Pre processor	conductors available	no limit	no limit	no limit
Pre processor	conductors displayed	2000	2000	20 000
TOSCA, ELEKTRA, SCALA, CARMEN, SOPRANO, TEMPO and post processor	nodes	50 000	no limit	no limit
Modeller	nodes	no limit	no limit	no limit

The current limits of the three sizes are given in the following table:

a. Entities = total number of points+lines+facets+volumes.

• UNIX workstations are supplied with Sizes 2 and 3 of the Pre-processor. Other programs are self-sizing.

Chapter 2 User Interface

Introduction

The interactive Geometric Modeller, pre and post processors of OPERA-3d have user interface which comprises both a command line and a Graphical User Interface (GUI).

The GUI generates text commands which have the same syntax as the commands which can be typed in directly at the keyboard. There are some features which can be used only from the keyboard. This chapter gives full details of the command line interface. In subsequent chapters, the keyboard commands are described with indications, where appropriate, of the corresponding GUI interaction. The use of the GUI in the Modeller and post processor is described in the User Guides; the use of the GUI for the pre processor is given in "The Graphical User Interface" on page 4-3.

Within this manual, different fonts are used to differentiate between input and output of various types. The program's commands, parameters and keywords are shown in **RED UPPERCASE SANS-SERIF FONT**; **input** and output from the program in **bold** and normal weight teletype font. File names are shown in a *slanted font*. GUI items are shown in a **san-serif font**.

Keyboard Input

A typical keyboard input consists of a command to perform some action, together with parameters that determine how the action is performed. At other times the input is a list of 'free-format' keywords or numbers which provide additional input to an earlier command. Keyboard input is requested by a prompt of the form

name >

where name is the name of the program or program section being used or sometimes is a question to be answered. Alphabetic input can be in either upper or lower case. The maximum length of a command line is 255 characters.

Even in keyboard mode, some commands require graphical input. This is provided by positioning the cross-shaped cursor on the graphics window and typing a key on the keyboard or a mouse button.

Some commands are 'built-into' the command interpreter. Whenever a prompt of the form given above is issued, built-in commands can be typed, by starting the input line with \$.

Output Files

All user input and the responses from the program are stored in dialogue files:

- *Opera3d_Modeller_***n**.*lp* for the modeller
- *Opera3d_Pre_n.lp* for the pre processor
- *Opera3d_Post_n.lp* for the post processor.

User input is written to files called

- *Opera3d_Modeller_n.log* for the modeller
- Opera3d_Pre_n.log for the pre processor
- Opera3d_Post_n.log. for the post processor

log files are in a format which can be used as input to the program with the **\$ COMINPUT** command. Graphical input or cursor commands are included in the log files; they can be read back into the program when the **\$ COMINPUT** command is accessed from the GUI. (**\$ COMINPUT** is described on page 2-25.) The pre processor also written to a file called *Opera3d_Pre_n.backup*. This file can be used in place of a pre processor data file (*.oppre*). A unique set of files is created for each run of the programs. The lowest available value of **n** is used for all files.

lp, *log* and *backup* files are stored in a sub-folder (sub-directory) of the project folder (current directory) named *opera_logs*.

Additional output files can be created by the user to contain the program's usual output or user-defined output or both. The commands to do this are described on page 2-27.

Commands and Parameters

Commands and parameters control the programs. All commands and their parameters may be shortened to their minimum unambiguous form. In any case, a maximum of 16 characters is used, except for file name parameter values. For example, in order to input the **DISPLAY** command, any of the following character strings could be typed: **DISPLAY** or **DISPLA** or **DISPL** or **DISP** or **DIS** or **DI**. The single character **D** will not be sufficient, because other commands also begin with this character. The programs' command interpreter will return an intelligible message when an error is detected in the input. If **D** had been input as a command the interpreter would reply:

DCOD Message 2: Command 'D' ambiguous (CMND)

The Help Character (!)

Short help messages on the commands and their parameters can be obtained at any time by entering the help escape character; this is the exclamation mark (1). Entering a single exclamation mark on a new line will cause a list of all the commands to be displayed, together with a one line description of each command's function. Entering a command name followed by a single exclamation mark will produce a one line description of the command, followed by a list of all the command's parameters with their current value and a description of their function.

• Examples: (Not all the commands and parameters are shown here.)

OPERA >	!
Valid o	commands are:
BHDATA	Define and modify BH data
SOLVE	Create or update an analysis database
READ	Read a file of OPERA-3d pre processor data
END	End OPERA-3d pre processor
\$	Built-in commands. Type `\$!' for a list.
OPERA >	read !
Read a f	ile of OPERA-3d pre processor data
Paramet	er Value Meaning
FILE	File name

Parameter Assignment

Parameter values are specified either by entering an assignment instruction

parameter=value

or positionally by entering the values for the parameters in sequence. Both forms of specification may be mixed, in which case specifying

xeye=value1 value2 value3

implies that **value2** is assigned to the next parameter after **xeye** and **value3** to the one after that. The parameter sequence for a command is fixed in the order listed by the help escape character !. When assignment instructions are used to specify the value of parameters the order is not important, except when expressions which reference other parameters are used (see examples on page 2-10).

Parameter assignments may be separated either by a comma or spaces; any number of spaces may be used, but if two commas are used in positional input mode this implies that the parameter value is not supplied. Whichever input mode is being used, a comma at the end of an input line implies that the command will be continued on a subsequent line. In this case the parameters entered on the first line are assigned in the program, but the action is not initiated. The first parameter on a continuation line must be assigned explicitly, i.e. using **parameter=value** syntax.

Parameters are unique to the command with which they are associated. The only exceptions to this are the parameters **COMPONENT**, VX, VY and VZ. These take expressions to define the output field quantities. In this case the expression(s) given in one command become the default values for other commands which use the parameters.

The value of the parameters associated with a command are in general initialized to sensible defaults when the programs start, although there are cases where it is not sensible to provide a default. For example, there is no default for the file name with the **READ** command. The last value used for a parameter (in a command) becomes the default value for that parameter the next time the command is used, except in cases where this could be disastrous. The exceptions are obvious, for example, with commands that delete objects the object names or numbers will not be defaulted to the last value. • Example: Using the following **DISPLAY** command as an illustration (only some of the parameters are shown):

```
OPERA > display !Display mesh and conductorsParameter Value MeaningSIZE100.0 The size of the viewing windowXEYE0.0 X coordinate of eye positionYEYE0.0 Y coordinate of eye positionZEYE0.0 Z coordinate of eye position
```

• Example: Assignment instruction mode:

OPERA > disp size=50 xeye=10 yeye=5 zeye=2

• Example: Positional input mode

OPERA > disp 50 10 5 2

• Example: Mixed positional and assignment

OPERA > disp xeye=10 5 2 size=50

• Example: Missing positional input, YEYE takes last value by default

OPERA > disp size=50 xeye=10,,2

Parameter Values

There are 4 types of value which may be assigned to a parameter: **Numeric, Expression, Character** and **Boolean**. Some parameters can take several value types but some combinations, such as character and expression are not allowed. Error messages indicate if an inappropriate value type has been used, e.g.

DCOD Message 19: Parameter 'TYPE' cannot take
numeric values (DECODE)

Numeric Parameter Values

Numeric values are used in many commands for specifying position, size, number of objects etc. Numeric values can be integer, fixed or floating point REAL or DOUBLE PRECISION numbers.

- Examples:
- 23 1.2 3E5 -5.789E+04 2.305983743795d5 -0.04

Expressions in Parameter Values

Most parameters which can take numeric values can also take algebraic expressions to specify the values. Expressions used in this way for data input are a replacement for a calculator. Variables within such expressions can be other parameters, system variables or user variables (see the **\$ PARAMETER** and **\$ CONSTANT** commands on page 2-21). These input expressions are not remembered; they are evaluated and the result is stored.

Parameters which cannot take expressions as values are those which also take character values. The text functions, %INT() and %REAL() (page 2-12) provide a way of getting around this restriction.

Expressions are also used to specify user defined parameters for output field quantities in post processing. These expressions are remembered and used for evaluation when referenced. Variables in output expressions can

also include the position and the field components. Full details are given in Chapter 5.

Within expressions, variable (parameter) names cannot be abbreviated. If a command parameter is used in an expression, its name must be typed in full.

The following characters can be used in expressions, with their usual FOR-TRAN meanings: + - / * (). No spaces can be included within expressions.

The following functions are supported, again using their usual FORTRAN definitions:

FUNCTIONS			
Arithmetic			
ABS(a)	the modulus of a		
INT(a)	the largest integer whose magnitude does not exceed		
	the magnitude of a times the sign of a		
NINT(a)	the integer closest to a		
MAX(a;b)	the maximum of a and b		
MIN(a;b)	the minimum of a and b		
MOD(a;b)	the remainder when a is divided by b		
SIGN(a;b)	the modulus of a times the sign of b		
Trigonometry (a	ngles in radians)		
ACOS(a)	the angle whose cosine is a		
ASIN(a)	the angle whose sine is a		
ATAN(a)	the angle whose tangent is a , angle in the range		
	$\left(-\frac{\pi}{2},\frac{\pi}{2}\right)$		
ATAN2(a;b)	the angle whose tangent is a/b taking into account		
	the signs of a and b and allowing b to be zero,		
	angle in the range $(-\pi, \pi)$		
COS(a)	the cosine of a		
COSH(a)	the hyperbolic cosine of a		
COTAN(a)	the cotangent of a		
SIN(a)	the sine of a		
SINH(a)	the hyperbolic sine of a		
TAN(a)	the tangent of a		
Trigonometry (angles in degrees)			
ACOSD(a)	the angle whose cosine is a		
ASIND(a)	the angle whose sine is a		
ATAND(a)	the angle whose tangent is a , angle in the range		
	(-90, 90)		

FUNCTIONS (continued)		
ATAN2D(a;b)	the angle whose tangent is a/b taking into account	
	the signs of a and b and allowing b to be zero,	
	angle in the range (-180, 180)	
COSD(a)	the cosine of a	
SIND(a)	the sine of a	
TAND(a)	the tangent of a	
Exponentials and logarithms		
EXP(a)	the value of $e^{\mathbf{a}}$	
LOG(a)	the natural logarithm of a	
LOG10(a)	the common logarithm of a	
SQRT(a)	the square root of a	

N.B. Functions with 2 arguments, ATAN2 and MOD, use ';' to separate the arguments, since ',' is the separator between parameter assignments.

• Example: Parameter assignments:

```
OPERA > line x1=x1+10 x2=x2+10
```

• Example: order is important since the expressions are decoded in the order given; the following commands are not equivalent:

OPERA > line x1=y1+4 y1=y1+3 OPERA > line y1=y1+3 x1=y1+4

• Example: Output components in post processing:

```
OPERA > map comp=sqrt(x**2+y**2)*bx
```

• Example: Functions can be concatenated:

OPERA > \$const #prob max(abs(#xmin);abs(#xmax))

• Example: The MIN and MAX functions can only have two arguments. In order to find the maximum of 4 numbers, the following expressions can be used:

OPERA > \$const #prob max(max(#a;#b);max(#c;#d))

Character Values for Parameters

Character values are character strings, starting with an alphabetic character. In most cases the value is compared against a list of valid options. In such cases the value can be abbreviated to its minimum non-ambiguous length. Specifying the help character, '!', will cause the program to give a list of the valid options. In other cases character values are used to give file names. In such cases longer strings are permitted. For operating systems which allow file name extensions or file types, the types are added automatically, the precise type being determined from the context. On systems where file names are case sensitive, file names which are entirely upper-case are given upper-case extensions; other file names are given lower-case extensions.

File names given as tree-names can include environment variables within the directory part of the name. Variables **\$VFDIR** (UNIX) and **\$VFDIR**% (Windows) are defined by the software as the parent directory or folder holding the software.

Some character strings are used for titles or text messages. For these strings, the rule about the first character being alphabetic can be relaxed. However any string which contains spaces or commas must be enclosed in quotation marks ('). Quotation marks embedded within character strings must be paired. The GUI automatically supplies quotation marks when necessary.

In the pre processor, superscripts can be used in title strings. This is done by including the characters $\sim E$ before and $\sim A$ after the superscripts. It is not necessary to use $\sim A$ if the string ends with superscripts.

• Examples:

```
OPERA > disp label=pote
OPERA > acti file=quadrupole
OPERA > title string='Field after 10~E-3~As'
```

The last example would give the title:

Field after 10^{-3} s

Boolean Parameter Values

Boolean parameters take the values YES or NO and are in general used for switching features on or off. Boolean values can also be specified by **+PARAMETER** or **-PARAMETER**, being equivalent to **PARAMETER=YES** and **PARAMETER=NO**. The state of a boolean parameter can also be changed from NO to YES (or from YES to NO) with **!PARAMETER**.

Some parameters can take boolean or character values

• Examples:

OPERA > disp +perspective OPERA > disp pers=yes OPERA > disp elem=no

Text functions

It is sometimes necessary to insert the value of an expression into a parameter value as a character string, for example, to include an index number in a file name or to supply a value by expression to a parameter which cannot take expressions. This can be done using text functions.

%INT and%INT(expression) or %INT(expression, width)%REAL%REAL(expression) or %REAL(expression, width)

These functions evaluate the expressions given and replace %*func-tion(expression)* on the command line with characters representing the value (%REAL) or the value rounded to an integer (%INT).

For example, to specify the maximum and minimum contour values using user variables:

disp min=%real(#min) max=%real(#max)

The forms of %REAL and %INT with the second argument, *width*, allow the user to specify the number of characters used in the characters string representation of the value. For example, if **#a** has the value **1234**, then

- %int(#a) will be replaced by 1234
- %int(#a,0) will be replaced by 1234 (0 has the same effect as omitting the width)
- %int(#a,2) will be replaced by 1234 (if the width is smaller than that required to represent the number, the actual width is used)
- %int(#a,6) will be replaced by 001234 (if the width is too large, leading zeroes are included)
- %int(#a,10) will be replaced by 0000001234
- %int(#a,15) will be replaced by 0000001234 (10 is the maximum allowable width)

For example, if **#b** has the value **pi*1.e6**, then

- %real(#b) will be replaced by 3.14159265358979E+06 (15 significant figures)
- %real(#b,0) will be replaced by 3.14159265358979E+06
 (0 has the same effect as omitting the width)
- %real(#b,2) will be replaced by 3.E+06 (if the width is smaller than that required to represent the number, at least 1 significant figure will be shown)

- %real(#b,10) will be replaced by 3141592.65 (10 characters)

%EXPR The text function %EXPR(variable)

replaces itself with the expression which variable represents. The *variable* can be the parameter of a command, e.g. COMP or a user variable. It can be used to modify the expression. In the following example, the first component expression is modified by dividing by a constant:

```
disp comp=bmod/hmod
disp comp=%expr(comp)/mu0
```

This second command is equivalent to

disp comp=bmod/hmod/mu0

%COMPARE %COMPARE(**a**,**b**) compares two character strings, which can be specified using character variables. It replaces itself with the numerical values

- -1 if **a** is less than **b**
- 0 if **a** is the same as **b**
- 1 if **a** is greater than **b**

where *less than, same as* and *greater than* refer to the positions of **a** and **b** in alphabetical order ignoring any difference between upper and lower case. For example command input files can be used to ask questions. The replies can now be given using character strings rather than numbers. The command input file below processes results at different times for a steady state ac or transient eddy current analysis:

\$prompt type 'Steady-state or Transient (AC or TR)'
\$if %compare(&type&,ac) eq 0
commands to display ac results, varying phase angle
\$elif %compare(&type&,tr) eq 0
commands to display transient results, loading different cases
\$end if

%ENV

%ENV(*a*) returns the value of the environment variable, *a* allowing command scripts to be programmed to behave differently for different users, processors, operating systems, etc. If the given variable is not defined, an empty string is returned. The following environment variables are defined for all operating systems:

Variable	Meaning
COMPUTERNAME	Network name of computer
CPU	Processor type
HOME	Home directory or folder
OS	Operating system name
USERNAME	User's login name

For example, to restrict some commands to only run on a computer running the operating system HP-UX:

```
$cons #ishpux 1-abs(%compare(%env(OS),HP-UX))
$if #ishpux eq 1
...
$end if
```

Command Interpreter Errors

The command interpreter provides input error recovery facilities. If a parameter name is mistyped, the other assignments on the input line will be performed, unless they are positional assignments whose position can not be determined, but command action will not continue. The incorrect parameter(s) can then be re-specified without having to retype the whole input line. The same applies to errors detected in the value of a parameter. The command interpreter will display any portion of the input string which it cannot recognize or which it thinks is in error so that the user can see which parameters need to be re-specified.

Confidence Level

Experienced users rely on the last used defaults and the mixed assignment and positional input modes to make efficient use of the programs. There are other useful features in the interpreter which can be used to reduce the amount which has to be typed.

Repeated commands

If the same command is being repeated many times the command name need not be supplied, providing that an assignment instruction starts the input line.

• Examples:

```
OPERA > disp size=100
OPERA > size=10
```

Prompted input of parameter values

The final feature of the command processor is its prompted input mode. Issuing a command followed by two help escape characters (!!) puts the command interpreter into prompt mode. Each parameter is displayed together with its default value and description. The default is accepted by pressing the **<Enter>**, **<Return>** or \dashv key, or a new value may be entered. When all the parameters have been offered the program waits for either **<Enter>**, **<Return>** or \dashv to be pressed, which then executes the command, or if '\$ABORT' is entered the command is aborted. '\$ABORT' can be used instead of any parameter value to abort the prompting at that point and not execute the command. '\$SKIP' can be used to skip over the remaining parameters and execute the command.

Note that Boolean parameters cannot be specified using **+PARAMETER**, **-PARAMETER** or **!PARAMETER** when in prompt mode. The character values **YES** and **NO** should be used instead.

• Example:

```
OPERA > disp !!
There are 24 parameters
For each parameter:
hit return to accept default
OR enter new value
OR type $HELP for help
OR type $SKIP to skip remaining parameters and exe-
```

cute command OR type \$ABORT to skip remaining parameters and abort command NO. Name Value Meaning 1 SIZE 100 Size of half-axis in each direction !! > 10 2 XEYE O X coordinate of eye position !! > 3 3 YEYE 0 Y coordinate of eye position !! > 4 4 ZEYE 1000 Z coordinate of eye position !! > 5 Type return to obey command, or \$ABORT to abort !! >

Prompted free format Input

Once a specific option has been selected by command or graphical input the programs may prompt for extra input to define further parameters. In such cases the user is shown the parameters required and asked to provide values. The parameters are input in free format using **<space>** or comma as the parameter separator. The order of the parameters in this type of input is shown by the prompt, however parameters defined in the manuals as optional keywords may be specified in any order. Free format input lines cannot be continued on subsequent lines by means of a comma.

In some contexts, for example coordinate input in the Point Definition Mode of the pre processor, values can be omitted in free-format input so that default, or previously specified values apply. The first value is omitted by using a comma at the start of the input line; subsequent values are omitted by using repeated commas within the line or by truncating the line after one or more values.

• Example: Boundary condition input in the pre processor **DEFINE** command.

Define the boundary condition for this face e.g. POT 0.0 OP-B/C > **pot 2 all**

The following line would be rejected since the keyword potential is not followed by a value.

```
OP-B/C > potential all 2
```

So that expressions can be used in free format input, each item is given a name according to its position on the line. #1 is the first; #2 the second etc. These names can be used to access defaults values in the input of coordinate positions or to use the values of earlier items on the same line.

• Example: Construction line input, the value of *u2* is set in terms of *u1*:

OP-C/LINES > line 1.34*sin(pi/6) 10 #2+20 10 0

Built-in Commands

Built-in' commands provide control-structures (loops and conditions), user variables, command input from files and access to the operating system. 'Built-in' commands can be used at (almost) any prompt.

\$ at the start of an input line introduces a 'built-in' command or \$-command. There is a built-in dictionary of commands and parameters and the normal

\$ command parameter=value ...

syntax can be used. The parameters have been ordered so that it is natural to use positional assignments. The parameter names are useful to provide on-line documentation using the ! character. Except where noted, there are no default values. \$-commands must be specified in full and cannot be continued on subsequent lines.

Limitations

The code which implements these loops and control-structures has the following limitations. Control structures can be nested to a depth of 20 levels. The maximum number of commands from the first control command to the last \$ END (inclusive) is 10000.

Loops

Three types of loop are available: **\$ DO**, **\$ FOR** and **\$ WHILE**. In each case the commands between the loop command and the corresponding **\$ END** command are executed a number of times.

\$ DO-*loops* The \$ DO-loop is similar to the FORTRAN do-loop. At the start of each execution of the loop, and index-variable is set to a value specified by a starting value, a final value and an increment. The syntax of the command is

prompt > \$ DO index start final increment
... commands to be executed ...
prompt > \$ END DO

The *index* should be the name of a user-variable, (up to 16 characters, beginning with #). Its value can be changed within the loop, but is always

set to the correct value (c.f. **\$ CONSTANT**, see page 2-21) at the start of the loop.

start, final and *increment* can be specified as numerical values or expressions. Expressions are evaluated before the first pass through the loop. If *increment* is omitted it has a default value of 1.

\$ FOR-each loops
The \$ FOR-each loop executes a set of commands with a user-variable set in turn to each of the expressions given on the \$ FOR command. The syntax is

prompt > \$ FOR index ex1 ex2 ex3 ... ex9
... commands to be executed ...
prompt > \$ END FOR

At least one, and at most 9, expressions (ex_n) can be given. *index* is assigned in turn to each expression (c.f. **\$ PARAMETER**, see page 2-21) at the start of the loop.

\$ WHILE-loops The **\$ WHILE**-loop executes a set of commands while a logical expression remains true. The syntax is

prompt > \$ WHILE ex1 logical_operator ex2
... commands to be executed while logical expression is true ...
prompt > \$ END WHILE

The logical operators are EQ, NE, LE, LT, GE and GT.

Conditional commands

Three conditional commands are available: **\$ IF**, **\$ ELIF** and **\$ ELSE**. The commands **\$ IF** and **\$ ELIF** should be followed by a logical expression. The **\$ ELSE** command has no parameters.

A \$ IF block (the commands executed if the logical expression is true) is terminated by a \$ ELIF, \$ ELSE or \$ END IF command.

A \$ ELIF (else-if) block is terminated by a \$ ELIF, \$ ELSE or \$ END IF command; it is only executed if the logical expression is true and none of the preceding \$ IF or \$ ELIF blocks at the same level have executed.

A **\$ ELSE** block is terminated by a **\$ END IF** command; it is only executed if none of the preceding **\$ IF** or **\$ ELIF** blocks at the same level have executed.

The syntax is

```
prompt > $ IF value1 logical_operator value2
... commands to be executed if logical expression is true ...
prompt > $ ELIF value1 logical_operator value2
... commands to be executed if previous blocks have not been executed
and logical expression is true ...
prompt > $ ELSE
... commands to be executed if previous blocks have not been executed
...
prompt > $ END IF
```

The logical operators are EQ, NE, LE, LT, GE and GT.

To enable the user to create command scripts which can be run in the Modeller, pre processor and post processor, the following variables are set by the software and can be tested using the \$ IF command: PREPROCES-SOR, MODELLER, POSTPROCESSOR and PROGRAM. For example

```
$ if program eq modeller
... commands for modeller ...
```

```
$ elif program eq postprocessor
```

```
... commands for postprocessor
```

\$ end if

The **\$ END** command

The **\$** END command ends the current block (DO, FOR, IF or WHILE). Although the block type is not logically necessary, it must be specified to ensure that the user knows which block is being ENDed and to help in supplying the correct number of **\$** END commands. The syntax is

prompt > \$ END block_type

User Variable Commands

The **\$** CONSTANT, **\$** PARAMETER and **\$** STRING commands define user variables. Two further commands, **\$** ASK and **\$** PROMPT, request the user to supply values for user variables and are described on page 2-26.

Each of the three commands has the same two parameters. The first defines the NAME of the user variable and the second the VALUE. If the name is used again then the value for that variable is overwritten. If no VALUE is given, the current value for the NAMEd user variable is displayed. If NAME=! is used then all the user variables currently defined are listed. The second parameter gives the VALUE for the user variable.

NumericalNumerical user variable names, defined with \$ CONSTANT orVariables\$ PARAMETER, start with # and have up to 15 additional characters.
They are not case-sensitive.

The VALUE can be a simple numeric value or can be an expression referencing other user variables or system variables. The **\$ CONSTANT** command evaluates the VALUE at the time the command is used and any expression is lost. The **\$ PARAMETER** command stores the expression given by the VALUE parameter so that it can be re-evaluated each time the variable is referenced.

• Example - to define degrees to radians conversion factor:

```
OPERA > $ constant #fac pi/180
Assign a value to a user variable
Name Value Expression or Value
#FAC 0.0174533 0.0174533
OK
```

It is possible to write simple programs using the **\$ CONSTANT** and **\$ PARAMETER** commands. The user parameters are evaluated at the time they are defined and again whenever they are referenced. Thus changing a user variable definition implies a change in all user parameters which reference that variable. This is shown by the following example. Note how changing the value for #A implies a change in value for #B and #C.

• Example

```
OPERA > $ constant #a 3
Assign a value to a user variable
Name Value Expression or Value
      3.0
#A
             3.0
OK
OPERA > $ parameter #b #a**2
Assign an expression to a user variable
Name Value Expression or Value
#B
      9.0
             #A**2
OK
OPERA > $ parameter #c #b-4
Assign an expression to a user variable
Name Value Expression or Value
      5.0
#C
             #B-4
OK
OPERA > $ constant name=! value=0
```

Assign an expression to a user variable Name Value Expression or Value 3.0 #A 3.0 #A**2 9.0 #Β #C 5.0 #B−4 OK OPERA > \$ constant #a 2 Assign a value to a user variable Name Value Expression or Value 2.0 #A 2.0 OK OPERA > \$ constant n=! Assign a value to a user variable Name Value Expression or Value 2.0 2.0 #A 4.0 #A**2 #Β #C 0.0 #B−4 OK

Menu Routes to \$PARAMETER and \$CONSTANT • Pre-processor: OPTIONS↓ Parameters

OPTIONS↓ Constants

The **\$ PARAMETER** and **\$ CONSTANT** commands are also available at many places in the pre processor GUI using the menu option Calculator.

- Post processor: icon 🔲 and menu route:
- Options \downarrow
 - User variable calculator
- Modeller: access to parameters and constants is provided by the VARI-ABLE command (page 3-157). On the Modeller menu this is accessed

through icon and menu route:

```
Create \downarrow
```

```
Variables
```

Options \downarrow User variable calculator

Character Variables Character variable names, defined with the **\$ STRING** command, have up to 16 characters, starting with a letter.

The VALUE can be any character string. The character string can be recovered on (almost) any input line by use of the NAME surrounded by ampersands ($\boldsymbol{\varepsilon}$). Any quotation marks used to define the string are lost. This allows several strings to be concatenated.

There are three predefined character variables and one predefined name which can be used if necessary:

- NOW and TODAY, always hold the current time and date.
- VERSION holds the version number of the software.
- YESORNO can be defined as YES or NO to pre-answer (and therefore avoid) questions the software might ask, e.g. before over-writing a file or ending a program.

• Example - storing a title for later use (note the use of **%real** to obtain a character representation of the value of a system variable (see page 2-12):

OPERA > \$ string t1 'Septum Magnet'
Assign a string to a user variable
Name String
T1 Septum Magnet
OK
OPERA > \$ string t2 '(RMS error %real(#err)%)'
Assign a string to a user variable
Name String
T2 (RMS error 5.23146%)
OK
OPERA > title '&tl& &t2&' tr
Set a title for the graphics window

The title displayed is 'Septum Magnet (RMS error 5.23146%)'.

Menu Routes to
\$STRING
• Post processor
Options ↓
User defined string variable
• Modeller
Create ↓
User defined string variables
Options ↓
User defined string variable
I

Command Input Files

The **\$ COMINPUT** command allows commands to be read from a file and additionally sets the message output mode. If a file with no file name extension is given, the extension *comi* is assumed. The syntax is:

prompt > \$ COMINPUT filename mode commentstyle

If the **\$ COMINPUT** command appears in a loop, the file of commands is read each time the loop is executed. Almost any command can be included in a command input file.

	Menu Route to \$ COMINPUT	File \downarrow Commands in
_	Text Output Modes	The parameter MODE applies whether or not a command file is requested. In PAGED and CONTINUOUS modes, the text output continues until the next input is requested, and with MODE=OFF most of the normal text out- put does not appear at all. MODE=PICTURES is useful for running 'dem- onstration' command files, since the program pauses for an <enter></enter> , <return></return> or \dashv before each time the graphics window is cleared, but does not stop when the text window is full. In each MODE , text output is written to the dialogue file.
		When menus are being used the text output modes are slightly different. While a command file is being read with MODE=CONTINUOUS , text output appears on the text window and does not appear in MessageBoxes. An additional option, MODE=MESSAGE causes the MessageBoxes to be used.
	Comment Styles	 Two different styles of comment are supported: commentstyle=line means that if a line starts with a comment character, /, the whole line is commented; commentstyle=command means that the line is split into separate commands by the command delimiter, , before looking for comments. See "Command Separator and Comments" on page 2-33 for more information on comments and multiple commands.
	\$ PAUSE Command	Execution of command files can be interrupted using the settings of the MODE parameter. It can also be interrupted by inclusion of \$ PAUSE commands. The syntax is: prompt > \$ PAUSE seconds

\$ PAU	SE	wa	its fo	or a i	number of	secon	ds b	efore	e cont	inu	ing. It	f seconds	is
omitted	or	is	≤0,	the	program	waits	for	the	user	to	type	<enter< td=""><td>>,</td></enter<>	>,
<retur< td=""><td>n></td><td>• or</td><td>0 1</td><td>r dis</td><td>miss a Me</td><td>essageI</td><td>Box</td><td>befo</td><td>re cor</td><td>ntin</td><td>uing.</td><td></td><td></td></retur<>	n>	• or	0 1	r dis	miss a Me	essageI	Box	befo	re cor	ntin	uing.		

Prompting Commands Command input files can contain user variables, which must be assigned values before the commands are executed. The \$ASK command can be used to request the user to supply a value for a numerical variable (c.f. the \$CONSTANT command). The \$PROMPT command can be used to request the user to supply a value for a character variable (c.f. the \$STRING command). The syntax is:

prompt > \$ ASK #name prompt_string
prompt > \$ PROMPT name prompt_string

The optional *prompt_string* is displayed to show what input is required. The value must be supplied at the keyboard before the program will continue.

N.B. **\$ ASK** and **\$ PROMPT** cannot be used in OPERA-3d pre processor data files; they can only be used in command input files.

File Existence The **\$ EXIST** command tests the existence of a file. The syntax is

\$ exist filename

If the file exists, the system variable FILEEXISTS is given the value 1; if the file does not exist, FILEEXISTS is set to 0.

Error Handling Commands The \$ ERRORHANDLER command selects the behaviour of the command processor after an error in a \$-command has been detected. The default behaviour, which can be selected using \$ errorhandler yes, is that all commands already stored for execution are ignored.

If **\$ errorhandler no** has been specified, the programs continue to execute stored commands. (Commands are stored during execution of **\$ COMINPUT**, **\$ DO**, **\$ FOR** and **\$ WHILE** commands.)

While **\$ ERRORHANDLER** is set to NO, the **\$ BREAKERROR** command causes the command processor to exit the current loop if an error has been detected. This enables command loops which read (see page 2-27) to the end of a file without knowing in advance the number of lines in the file. A typical sequence of commands could be:

\$ constant #i 1
\$ while #i eq 1

Command

(note that in this example the **\$ BREAKERROR** command directly follows the **\$ READ** command).

User Input/Output Commands

There are seven commands for user input and output of files.

OPEN	Before a file can be read or written it must be opened.
	<pre>prompt > \$ OPEN stream filenm authority ± REDIRECT</pre>
	opens a file on a logical stream number which can be in the range 1 to 10. The file can be used in 4 ways, depending on the authority . These are READ an old file, WRITE a new file, OVERWRITE an old file and APPEND to an old file.
	If +REDIRECT is selected for an output file, the output which is written to the dialogue file will be written to the output file as well.
CLOSE	When all input or output has been completed a file can be closed to release its logical stream number or to make it available for opening with different authority.
	prompt > \$ CLOSE stream
	closes a logical stream number in the range 1 to 10.
READ	The \$ READ command takes one line from the file opened on the given logical stream number and assigns any data on the line to user or string variables. Up to 20 variable names can be given. If no character variable names are given, any character strings on the line are ignored. The variable assignments are listed unless the -PRINT option is given. The syntax is
	prompt > \$ READ stream var1 var2 var3 ±PRINT
WRITE	The \$ WRITE command is similar to the \$ READ command. The syntax is
	prompt > \$ WRITE stream data1 data2 ±OUTPUT

Up to 20 data items can be supplied.

A line of output can be built up using several \$ WRITE commands. If this is necessary the first \$ WRITE commands should have -OUTPUT. The last \$ WRITE should have +OUTPUT. The data from the second and subsequent \$ WRITE commands will be positioned after the data of the previous write commands in an internal buffer which is written and re-initialized when +OUTPUT is used.

Data items on a \$ WRITE command can be numerical, characters or user variables. Before the \$ WRITE command is used, the \$ FORMAT and \$ ASSIGN commands should be used to define the type of data and the style of output to be used for each item on the \$ WRITE command.

FORMAT

The **\$ FORMAT** command can be used to define up to 20 different formats for output items. In each form the *width* can be specified as zero which implies that the program should calculate a width to fit the data being written. The syntax has one of the following forms.

• To define a format for character data, truncated or padded with spaces to a particular width:

```
prompt > $ FORMAT number CHARACTER width
```

• To define a fixed point format for numerical data, with **decs** as the number of decimal places.

```
prompt > $ FORMAT number FIXED width decs
```

- To define a floating point format for numerical data: prompt > **\$ FORMAT number EXPONENTIAL width**
- To define a format for integer data:
- prompt > \$ FORMAT number INTEGER width
- To define a format for a user variable to display the expression defining the variable truncated or padded to a particular width:

prompt > \$ FORMAT number USER width

• To define a character string to be output irrespective of the data on the \$ WRITE command (N.B. It is necessary to define STRING formats containing spaces to appear between other data items if those other data items are written with a width of zero.):

prompt > \$ FORMAT number STRING width STRING=chars

To list the defined formats:
 prompt > \$ FORMAT +LIST

ASSIGN

The **\$** ASSIGN command assigns format numbers to the data items of subsequent **\$** WRITE commands. The first data item will be written with the first non-STRING format, the second item with the next non-STRING format, and so on. The syntax of the \$ASSIGN command is:

```
prompt > $ ASSIGN form1 form2 form3 ...
```

Up to 20 formats can be assigned, with no default format assumed.

BACKSPACE The final input/output command is **\$ BACKSPACE** which allows a file opened with **\$** OPEN to be backspaced or positioned at its start. The syntax is:

```
prompt > $ BACKSPACE stream records
```

where *stream* is the logical stream number and *records* is the number of records the file is to be backspaced. If *records* is given as -1, the file is positioned at its start.

Example 1 A file *points.dat* contains:

 $\begin{array}{cccc} X & 0 & Y & 0 \\ X & 0 & Y & 1 \\ X & 0 & Y & 3 \\ X & 0 & Y & 5 \\ X & 0 & Y & 7 \\ X & 0 & Y & 7 \\ X & 0 & Y & 1 \\ X & 1 & Y & 1 \\ X & 1 & Y & 5 \\ X & 1 & Y & 9 \end{array}$

The following commands will read the data and use the coordinates in **POINT** commands to evaluate the field and to write a corresponding output file. Note the use of comments.

```
Open input file for reading
$ open 1 points.dat read
        Read the first line into user variable #np
/
$ read 1 #np
        Open output file for writing
/
$ open 2 fields.dat write
        Define a string format to space the output
$ form 1 string string='
        Define a floating point format
$ form 2 expo 0
        Assign format numbers for the output
$ assi 2 1 2 1 2 1 2
        Start a loop from 1 to #np
$ do #i 1 #np
        Read #x and #y from input file
$ read 1 #x #y
        Evaluate fields at #x #y
/
poin x1=#x y1=#y
        Write coordinates and flux density to output file
/
```

\$ write 2 x y bx by / End of loop \$ end do / Close files \$ close 1 \$ close 2

The output file, *fields.dat* contains:

0.0 0.0 1.179564E-07 -0.000120992 0.0 1.0 6.322637E-08 -0.000121184 0.0 3.0 3.739035E-06 -0.000133139 0.0 5.0 -4.42901E-06 -0.00013248 0.0 7.0 -7.59639E-06 -8.48776E-05 0.0 9.0 -7.74227E-06 -2.32268E-05 0.0 10.0 0.0 0.0 1.0 1.0 4.347521E-07 -0.000121116 1.0 5.0 -1.34893E-05 -0.000141764 1.0 9.0 -3.10067E-05 -3.09315E-05

Example 2

An example showing both the features mentioned above is as follows. A file *blocks.dat* is created that contains the definitions of some cubes to be defined in the OPERA-3d Modeller. The data file contains

 one
 0
 0
 0
 5

 two
 10
 0
 0
 5

 three
 20
 0
 0
 5

 four
 5
 5
 0
 5

 five
 15
 5
 0
 5

 six
 10
 0
 5

The following command file could be used to read *blocks.dat* and create the data:

```
$open 1 blocks.dat read
$cons #a 1
$errorhandler no
$while #a eq 1
   $read 1 name #x #y #z #l -print
   $breakerror
   block &name& #x #y #z #x+#l #y+#l #z+#l
$end while
$close 1
```

(note that in this example the **\$ BREAKERROR** command directly follows the **\$ READ** command).

Operating System Commands

There are two commands to execute operating system commands:

prompt > \$ OS str1 str2 str3 str4 str5 str6
prompt > \$ CD directory

\$ OS passes up to 6 strings (str_n) which together form a command to the operating system to be executed. This can be used to issue single commands to list names of files in directories (folders), delete or rename files, etc., using the normal syntax of the operating system. The output from the commands is listed, with the usual page breaks.

On UNIX systems, in order to redirect the output from the command to a file, the program adds to the command the appropriate notation:

- in c-shell: user_command >& TeMpOsCmNdFile
- in other shells: user_command > TeMpOsCmNdFile 2>&1

The contents of the file, *TeMpOsCmNdFiLe*, are then listed. For this reason, shell meta-characters within the user command should be used with care and it might be necessary to enclose the user command in parentheses. For example, to run a background command use the following syntax:

\$OS (xterm &)

The \$ OSOn Windows systems a limited set of Command Prompt commands have
been implemented, although without full functionality. The commands
available are:

- dir or ls to list files in the current folder
- **del** or **rm** to delete a file
- **mkdir** or **md** to create a new folder in the current folder
- **ren** to rename a file
- copy to copy a file
- pwd or cd to report the current folder

N.B. File names including spaces should be enclosed in double-quotes (").

Additional **\$ OS** Commands to Launch OPERA On all operating systems additional commands are available for starting interactive and analysis programs from the interactive programs. The commands for the interactive programs are:

- \$ OS operapre mode
- \$ OS operapost mode
- \$ OS operamodeller mode

and for the analysis programs:

The **\$ OS** command on UNIX systems

	\$ OS operaanl program datafile mode
	 program is one of CARMEN, ELEKTRASS, ELEKTRATR, ELE- KTRAVL, SCALA, SOPRANOEV, SOPRANOSS, TOSCA or TEMPO
	• <i>datafile</i> is the name of the OPERA3d database to be analysed
	• <i>mode</i> is FORE or BACK. The interactive program waits while FORE- ground analysis jobs are run, but can be continued or ended while BACKground jobs are run.
The \$ CD command	Because the \$ OS command spawns a new sub-process, the command: prompt > \$ OS cd directory
	has no lasting effect. The \$ CD <i>directory</i> command should be used instead to change the current directory or folder. Directory names can be given using environment variables. Environment variables \$VFDIR (on UNIX systems) and \$VFDIR % (on Windows systems) are defined by the software as the parent directory of the software.
Menu Route	File \downarrow Change directory

Command Separator and Comments

The command separator allows several keyboard commands to be given on one input line. The command separator is the vertical bar, |.

Example using the LINE and PLOT commands:
 OPERA > line 0 0 0 10 0 0 | plot comp=by

If the first non-space character on an input line is /, the line is treated as a comment. Comments are output to the dialogue file and comments in command files are displayed as the file is being read. In menu mode, comments which start /* are displayed in GUI MessageBoxes.

In some contexts it is not possible to give comments, since for some operating systems, file names can begin with /.

Note that if multiple commands exist on a single line (separated with |), then starting the line with a comment character (/) will prevent all the commands on that line from executing (the comment character applies to the whole line, not just to the first command on the line). This behaviour can be changed using the **\$ COMINPUT** command with **comment-style=command** (see page 2-25).

Euler Angles

The programs occasionally require that the orientation of a component or a physical property be defined. Euler angles are always used to define the orientations. The following table shows the Euler angle convention used:

ANGLE	DESCRIPTION
PHI or P	Rotation about the original (global) Z axis positive rota- tion by right-hand screw convention i.e. from X towards Y.
THETA or T	Rotation about the local Y axis created by the PHI rota- tion, positive right-hand screw convention i.e. from Z' to X'.
PSI or S	Rotation about the local Z axis created by the PHI and THETA rotations, positive right-hand screw convention i.e. from X' to Y'.

Unless there are parameter names, the convention is always to define the rotation by the ordered triple **THETA PHI PSI** or **T P S**.

N.B. This order is different from the order used in the definition above.

Examples:

Coordinate Transformation	ϑ	Φ	Ψ
$XYZ_{local} = XYZ_{global}$	0	0	0
$XYZ_{local} = YZX_{global}$	90	0	90
$XYZ_{local} = ZXY_{global}$	90	90	180

Whenever the ordered triple for Euler angles is to be entered into the pre processor, the escape function **\$EULER** can be typed instead. The program prompts the user to define the Euler angles by specifying **THREE** points or by **SWAPPING** the coordinate axes. The 3 points required are a local origin, and points on the local Z axis and either the local X or the local Y axis. The coordinate axes can be swapped by specifying the global axes which correspond to the local X and Y axes. The global axes, in either positive or negative directions, are referred to by keywords, **POSX**, **NEGX**, **POSY**, **NEGY**, **POSZ**, **NEGZ**. The Euler angles can be stored within OPERA-3d under user defined names and recalled later by name.

• Example: Setting material VECTOR property in the pre processor.

Set the material name, potential type and constants (or HELP or QUIT) OP-MATERIALS > iron total vector \$euler Give name of Euler angle set. Type LIST for a list or NEW to define a new Euler angle set. OP-EULER > **new** Give name and definition method. Methods are: THREe - coordinates of origin, and points on z and one other axis SWAP - swap global axes OP-EULER > **iron swap** Give a combination of POSX, NEGX, ..., POSZ, NEGZ to define local X, Y and Z axes. (Only X and Y need be given.) OP-EULER > **posy posz** Input line edited to replace \$EULER: iron total vector 90 0 90

2-36

Chapter 3 OPERA-3d Geometric Modeller

Introduction

The Geometric Modeller provides facilities for creating models for use with the OPERA-3d analysis modules and post processor.

The Modeller manipulates any defined objects through operations such as transformations and combinations. Basic objects (blocks, cylinders, spheres, cones, pyramids and toroids) can be created at any position in space. Once created they can be manipulated to reposition them. They can also be merged, intersected or subtracted from other objects in space to create more complex geometries.



Figure 3.1 Complex geometry created with the Modeller

This modelling technique allows many models to be created from the basic building blocks. Other more advanced techniques allow the geometry to be enhanced by e.g. sweeping an existing face.

Reference Manual Overview

Geometric Modelling	This section describes the terminology used throughout the program and guides, and introduces the basic techniques that are available when using the Modeller.
Reference	This section introduces each of the commands available within the Model- ler and gives details of the specific usage for each. Commands available through the menu system generate a command line equivalent. The details of how the commands operate is described in this section and gives more in depth detail of the functionality the command supplies

Geometric Modelling

Topology Reference

This section describes the different terms used to describe the topology of a model.

- *Vertex* This is a simple point in space, defined by a position within a cartesian coordinate system. Normally a vertex will mark a sharp change in geometry, e.g. the intersection of 2 or more edges.
- *Edge* An edge is a line in space. Typically an edge will have 2 vertices defining its ends. The edge can also have an underlying geometry associated with it, e.g. it could be straight, an arc of a circle, or a more complex underlying spline curve. For certain edges, notably circles and ellipses, the edge may have a single vertex or no end vertices at all.
- *Face* A set of edges, connected together at their ends, forms a loop. For most cases, such a loop forms the boundary of a face. A face also has an underlying geometry, e.g. the edges could bound part of a plane, part of a sphere or part of a more complex underlying surface definition.

There are also some cases where the face is bounded by more or less than a single loop. For some underlying surface geometries, e.g. a sphere or torus, there may be no bounding loop required. For structures formed from complex operations, there may be one or more internal loops that bound an interior section of the surface, excluding it from the face.

- *Cell* A closed set of faces connected at the bounding edges forms a shell. A cell represents an enclosed volume of space bounded by one or more shells. Any point within the cell can be reached from another point within the cell by travelling a path that does not pass through a face. For some complex geometries there may be more than a single shell of faces bounding a cell, e.g. where a section has been removed from the interior of a volume, leaving an internal set of faces restricting the volume occupied by the cell.
- **Body** A body is a collection of the above topological entities. All objects within the body are closely linked geometrically, so that parts of a body can not be moved without considering the impact on other parts of the body. For

example a body may contain 2 cells. These cells each occupy their own space. Movement of one cell within the body may make the cells overlap which would cause the cells to become invalid.

However, if there are 2 bodies, each with one cell, it is permissible to have the bodies overlapping as there is no connection between the bodies. Before forming a final model, the bodies will be merged into a single body to ensure that the topology forms a single valid structure.

Modeller Topology and Geometry Functionality

Creating New Objects Basic primitive commands exist for creating objects. The type of objects that can be created are:

Туре	Command	Description
Block	BLOCK	Creates a cuboid block. Specify one corner and its opposite to define a cuboid
Cylinder, cone, ellipsoidal cylinder	CYLINDER	Specify the positions at the centre of the base and centre of the top as well as the radius. Ellipsoidal cylinders and cones can be created by giving major / minor axes or a top radius of zero.
Sphere	SPHERE	Give the centre of the sphere and the radius
Torus	TORUS	Give the centre, the major and minor radii
Regular n-sided prisms and pyramids	PRISM	Creates a regular <i>n</i> -sided polygon with corner points on an ellipse (of given major and minor radius), and extends this in the third dimen- sion a given height with a taper given by a major radius at the top

These objects are created in the current Working Coordinate System. Local Coordinate Systems can be defined, and one of them selected to be the Working Coordinate System. All primitives are created within the Working Coordinate System, as are transformations.

A standard set of conductors can also be created. This is done through separate commands, e.g. the RACETRACK, BEDSTEAD commands. These conductors do not form a physical part of the model and are not included within the finite element mesh. They can be selected for transformation, repositioning and copying. For more details see the CONDUCTOR section and the individual CONDUCTOR commands.

Modifying Objects	Once created, objects within the model can be picked for modification. There are 2 ways to pick objects:
	• by graphical interaction, or
	• by picking objects that have certain attributes associated with them.
	Once an object is picked, operations can be performed upon that object. In many cases it is possible to pick more than one object and to operate upon all of the picked objects.
	The operation may perform differently depending upon the type of object picked, or may have no effect if the picked objects are not of suitable type.
	For example: the copy operation (see the TRANSFORM command) can be applied to all picked bodies to create transformed copies of these bodies. If a cell has been picked, this can only be copied individually, and the process creates a new body containing the single cell that has been copied. A new body is created to ensure that the body's topology is not corrupted which may happen if the cell were copied into a space already occupied by another part of the body. A face will behave in the same way as the cell, but creates a 2D body that has no volume. Edges and vertices are not operated upon.
	The topological entities that are affected by each command are listed in detail in the reference manual.
Graphical Interaction	Graphical interaction is controlled by the FILTER command. This determines what type of topological object can be picked by positioning the cursor over it and double-clicking with the left mouse button. The options for the filter command include: vertex, edge, face, cell, body.
	As well as the topological entities described, conductors and Local Coor- dinate Systems can be picked and modified.
	When double clicking, the system generates a command to pick a num- bered entity. If using the log file to generate command input files for auto- mated modelling, commands generated through the graphical interaction should be avoided. Although these commands are valid, there is no guaran- tee that variations between platforms or versions of the software will pro- duce the same numbering sequence. This may lead to incorrect models being produced. Where possible, picking by attribute should be the pre- ferred method when automating the Modeller using scripts.
Picking By Attribute	Each entity can have attribute data associated with it. These data will vary according to the entity type. For example, bodies can have a name associ-

ated with them, and a set of labels that can be applied. Cells have a material label, a volume label, a potential type, an element type, a mesh control size and a set of labels. Faces have a boundary condition label, a mesh control size and a set of labels. Edges and vertices have only a mesh control size and a set of labels.

The PICK command can use these attributes and labels to identify parts of a model for modification, viewing, etc.

Modifying Objects Geometry

The main command for modifying the geometry is the **TRANSFORM** command. This command can be applied to bodies. A limited form can also be applied to conductors and Local Coordinate Systems. All transformations occur within the Working Coordinate System. Transformations available include:

Transformation	Description
TRANSLATION	Re-position by translation
ROTATION	Rotation about an axis vector (through the origin) by an angle, or rotation through a set of Euler angles
REFLECTION (body only)	Reflect in a plane whose normal is specified (through the origin)
SCALE (body only)	Scale the body by anisotropic scale factors

More complex geometric changes can be performed using the morphing commands on bodies. These commands change the underlying geometric shape of the object. Morphing commands available include

Transformation	Description
BEND	Bend a body by through a fixed angle
MORPH	Apply any functional transformation
STRETCH	Stretch part of a body between 2 points
TWIST	Twist part of a body between 2 points

Creating Complex Bodies

More complex structures can be formed by performing Boolean operations upon 2 or more bodies using the **COMBINE** command. The Boolean operations are:

Operation	Description
UNION	Merge the 2 bodies so that the resulting body has the components that existed in both original bodies
INTERSECTION	Intersection will leave a body filling the volume of the model that was originally common to both bodies

Operation	Description (continued)
SUBTRACTION	Removes volume from the first body that also lies within the second body
TRIM	As subtraction, but removes a copy of the second and subsequent bodies from the first picked body
CUTAWAY	Leaves the first body unchanged and cuts it from all other picked bodies

For SUBTRACTION, TRIM, and CUTAWAY, the order of picking bodies is important. In this case the second body picked is subtracted from the first body picked. If more than two bodies have been picked, the third body picked is subtracted from the result of the first subtraction, and so on. For other operations, the order is unimportant.

For all operations, there is an option to regularise the result. Regularising removes internal faces and edges that are formed from the operation. This is most easily seen during union, but can also be useful for subtraction.

Extending Geometry

An existing planar face within the model may be picked and swept through space to extend the volume occupied by a body. When sweeping there are 3 options available:

Sweep operation	Description
DISTANCE	Sweep the face by a fixed distance normal to the surface
VECTOR	Sweep the face through a specified vector
ROTATION	Sweep the face around an axis, passing through a point, by a given angle
PATH	Sweep the face along a path formed from edges of the model

An option exists to keep the original face, in which case the sweep operation will form a new cell within the body.

The operation can also apply a taper so that the face expands or contracts at a constant angular rate.

When sweeping, care should be taken to avoid self intersections and formation of incorrect topologies. Where possible the sweep operation will try to fix these inconsistencies, but it cannot be guaranteed to work in all cases.

Blending And Chamfering The edges between adjoining faces can be blended to form a smooth interface between the faces, or chamfered to trim the edge. Both of these are achieved using the **BLEND** command. Such blending and chamfering may only be applied for non-manifold geometry.

Removing Geometry Features Sections of the model can be deleted. Any picked body, conductor or Local Coordinate System can be deleted from the model.

Other topological entities can also be deleted in certain cases.

- A vertex can be removed if the edges that link to it all have the same underlying geometric curve, e.g. represent adjoining parts of an arc of a circle.
- An edge can be removed if all faces touched by that edge have the same underlying surface geometry.
- Internal or external sheet faces can be deleted from a body. A sheet face is one that exists but supports no volume e.g. the product of copying a single face from a body creates such a face. An option to regularise allows any edges that are not needed to maintain the topology of the body to be removed from the body.
- External faces can be removed if the switch to allow this is activated. The removal will only occur if the resulting hole formed in the body can be patched. This cannot occur with faces that form part of a non-manifold geometry.
- Deleting a cell will remove all external faces that exist to support that cell. Any internal faces will now be external. An option to regularise allows removal of any edges and vertices that are no longer needed to maintain the topology of the body.

Checking And Correcting Topology

In some instances, particularly after some more complex operations, or with data from other applications, the geometry and topology of the structure will be incorrect. A CHECK command exists to test for potential problems. Any entity that fails the check command will be flagged with a SYSTEM label, and the body to which it belongs will also be flagged. These bodies can be displayed and a warning message will give an indication of why the problem occurred. The CHECK command contains additional facilities to help try to repair some of these problems. In some instances it may be necessary to remove and recreate any entities that exhibit problems, as these may cause future operations to fail.

Modeller Data

Within the Modeller it is necessary to set properties defining material properties, mesh control information, volume properties and boundary conditions. This information is used when creating the database for analysis by one of the analysis modules. It is also used to assist in visualisation of the model, as different data can be chosen as the basis for the display.

This data is stored with each entity with which it is associated, and is different for different types of topological entity.

- *Body Data* A body can have:
 - Name

This is used for specifically naming bodies.

• Additional labels

Used for grouping sections of the model for visualisation and modification.

When merging bodies the data from the body that is picked first is always maintained.

Cell Data A cell can have the following data attached to it:

• Material label

A label that is used to reference the material properties associated with the cell. By default cells are labelled AIR, but other labels can be applied, and the properties e.g. permeability or BH curve label, conductivity, permittivity, can be defined for each material label.

- Potential type (TOTAL, REDUCED, VECTOR)
- Element type (LINEAR, QUADRATIC)
- Volume properties label

A volume property label can be attached to cells and stores other data associated with the cell. This can include velocity, local orientation, packing factor etc.

• Mesh control size, surface normal and normal distance tolerance

Used to control the maximum size of mesh that is generated within this cell.

• Additional labels

Used for grouping sections of the model for visualisation and modification.

• A data storage level

When merging bodies with boolean operations, the resulting cells will normally be formed from a combination of the initial cells. In such a case it is not clear which of the initial data sets should be kept with each

Face Data A face can have the following data attached to it:

• Boundary condition label

This is a label that is used to reference the boundary condition data for this face.

- Element type (LINEAR, QUADRATIC)
- Mesh control size, surface normal and normal distance tolerance Used to control the maximum size of element in the finite element mesh that is generated on this face.
- Additional labels

Used for grouping sections of the model for visualisation and modification.

• Layering

Used to create thin layers on either or both sides of the surface with consistent surface meshing for problems where thin geometries are important.

• A data storage level

As with cells, merging may produce ambiguity in face properties. The data set with the greater storage level will be kept.

Edge And Vertex An edge or a vertex can have the following data attached to it: *Data* . Much control size

• Mesh control size

Used to control the maximum size of element in the finite element mesh that is generated on this face.

• Additional labels

Used for grouping sections of the model for visualisation and modification.

• A data storage level

As with cells, merging may produce ambiguity in edge properties. The data set with the greater storage level will be kept.

Setting Data Properties

The different commands for setting properties are

Command	Description
LABEL	Adds and removes labels for all picked objects
RENAME	Gives names to and renames bodies and Local Coordi- nate Systems
CELLDATA	Sets the properties of picked cells
FACEDATA	Sets the properties of picked faces
EDGEDATA	Sets properties of picked edges
VERTEXDATA	Sets properties of picked vertices
MATERIALS	Defines the data associated with a material label
VOLUME	Defines the data for each volume label
BOUNDARY	Defines the boundary condition data for each boundary
	label
BHDATA	Defines the BH curve file to be associated with a BH
	curve label (used when defining the material properties)

Properties of one or more entities can be set together. For cells, faces, edges and vertices: the entities to be modified are picked. Any common data is displayed. This can be changed and the data set. If a data item is left blank, the original data is left unchanged.

Listing Data A LIST command can be used to list data associated with the picked entity. The FILTER command can also be used to adjust the effect of double clicking over a part of the model, so that the LIST command is automatically called instead of picking the item for modification.

Display

The display uses an OpenGL interface to allow visualisation of the model. The model can be rotated, translated or zoomed using the mouse controls and the buttons in the display window.

View Selection The display of the model is controlled by the SELECT command. This allows individual items to be selected for display or to be explicitly hidden from view. There are many data items attached to objects, and these can be used to control the display of the model. The SELECT command allows the user to display or hide any object that has data of a certain type, e.g. any face with any boundary label, or more explicitly e.g. any face with a particular boundary label.

Contour Display	Some data attributes can be contoured on surfaces of the model. The con- tours will only be shown on parts of the model that are visible through the normal display selection. The CONTOUR command gives greater detail.
Vector Display	Some data attributes can be displayed as vectors within the model. The vectors will only be shown on parts of the model that are visible through the normal display selection. The VECTOR command gives greater detail.

Command History

The Modeller maintains a history of model changes. Most commands will generate an entry within the history stream. The UNDO command is available to return to any previously noted state of the Modeller. After an UNDO command, a REDO command can undo the changes of the UNDO command, until a new command has been issued. At this point the REDO command becomes invalid, and the history between the current position and the position at which the UNDO command was issued is removed.

Certain commands such as THREED, FILTER and SAVE do not affect the model and do not create an entry in the history stream.

Other commands, such as the **SELECT** command, generate changes. However, when the command is issued several times in succession, the effects are merged into a single state within the history stream.

Model Storage

The data within the Modeller can be saved at any time. Once saved a file can be opened using the LOAD command. This will clear any existing data from Modeller and will overwrite it with data from the new file. The file can then be saved after modifications. A complete model, including the finite element mesh can be saved in a binary file.

If no file was opened a name must be given the first time the model is to be saved.

Data from a file can also be inserted into a currently open model. This will add the topology and data from the file into the open model.

Individual component bodies can be picked and exported to data file.

The standard data formats are the *.opc* file containing the OPERA-3d Modeller components and *.opcb* file which include the finite element mesh data. ACIS *.sat* data files containing geometry and data from other applications

may be used within the OPERA-3d Modeller. It should be noted that in some cases, such data are not suitable for use with finite element analysis and may cause problems.

Building a Model for Mesh Generation

To generate a continuous mesh through the volume of space modelled it is necessary to have a single body. This body is made up of multiple cells containing different material data representing the different components of the structure being modelled.

With such a potentially complex single body, it is difficult to modify the geometry and topology of the structure being modelled.

To assist in making modelling easier, it is preferable to have many simple bodies, possibly overlapping if necessary, that can be easily moved, scaled etc. as required.

To make the transition from multiple bodies to a single body suitable for meshing, there is a MODEL command. This command makes a copy of each body within the model and merges them using a boolean union operation, without regularisation to ensure that internal material boundaries are maintained. The single body formed will have the properties defined on the components, adjusted where necessary to take account of conflicts by using the data storage level. The model created should be suitable for meshing and database creation. If the model is not correct, it is possible to adjust the properties and topology, although these changes will be lost when returning to component view.

When creating the model, a **BACKGROUND** region can be used to limit the model space. This is useful for cases where the whole model is easy to define, e.g. cylinders, but where symmetry is available and so the whole model is not needed for analysis. Any body may represent the background region, and it is defined simply by having the name 'background'. The effect of the background region is that it is intersected with the final model to limit the extent of the model's size and shape. If boundary conditions have been attached, these will be maintained on the final model if the data storage level is set correctly.

Additionally layering information attached to some faces can be used to create thin layers. These layers are forced to have the same topology as the master face, so that the mesh can be transferred directly between them, improving the quality of the tetrahedral mesh in these regions.

Once created, if changes are again required, the MODEL can be deleted, and the view will return to the component view. If adjustments have been made to the model, these would be lost when returning to component view.

Mesh Generation	Mesh generation is a two stage process. The MESH command generates a triangular surface mesh over all faces of the model. The mesh size is controlled by the mesh size, the surface, and the normal distance tolerances. The smallest of each of these is used to control the local mesh size. This command can only be used when the model has been created. The mesh can be displayed as outlines on the model display.
	The FILL command creates the volume tetrahedral elements throughout the model space, and can only be used once a valid surface mesh exists.
	Changes to the model topology will destroy the surface mesh, as will returning to component view. It is possible to UNDO back to the last meshes formed, but meshes prior to this will have been overwritten.
Database Generation	General settings for analysis control are entered using the ANALYSIS- DATA command. The options used are dependent on the analysis type.
	Control of the drive data for ELEKTRA-TR is set using the DRIVE command, along with the phase angle for ELEKTRA-SS and SOPRANO-SS.
	Output times for ELEKTRA-TR and frequencies for ELEKTRA-SS and SOPRANO-SS are set using the DBCASEDATA command.
	When the model is ready with a volume mesh created, the database can be prepared using the SOLVERS command.

Summary of Menus and Commands

Menus

The following is an overview of the first level menu structure. A brief description of the functionality of each item is also given.

<u>F</u> i]	.e <u>E</u> dit	<u>V</u> iew	<u>C</u> reate	<u>P</u> icł	king	<u>O</u> perations	P <u>r</u> operties	Model	<u>W</u> indow	IS	Help
			File								
È	<u>O</u> pen	•	C	Ctrl+O	load	a data file				page	3-96
	<u>S</u> ave		C	Ctrl+S	save	data to same fi	le			page	3-125
	S <u>a</u> ve a data	s new 1	model		creat	e a new data fil	e			page	3-125
	Save m <u>m</u> esh	odel w: •	ith		creat	e a new binary	file including	the mesh	ı data	page	3-125
	<u>R</u> evert	to sa	ved		undo	all changes ma	ade since last	file open		page	3-55
	<u>C</u> lose				remo	ove all data and	re-initialise c	ommands	5	page	3-55
8	<u>P</u> rint.	••	C	Ctrl+P	send	the picture to a	printer			page	3-119
	Comman	ds <u>i</u> n.	••		read	command inpu	t file			page	2-25
	Change	<u>d</u> irect	tory		chan	ge the current d	lirectory			page	2-30
	Recent	files			load	recently used d	ata files			page	3-96
	E <u>x</u> it				end I	Modeller sessio	n			page	3-72

Edit

ŝ	Undo	Ctrl+Z	remove the last change to the model	page 3-156
C	<u>R</u> edo	Ctrl+Y	re-apply a change which has been undone	page 3-123
	<u>H</u> istory		return to a previous state of the model	page 3-156 and page 3-89
	<u>C</u> opy to clipboard	Ctrl+C	copies the picture to the clipboard or a bitmap file	page 3-117
	Copy to <u>f</u> ile			
	<u>T</u> oler- ances		adjust tolerances	page 3-118

I

I

View

E Selection	select items for display	page 3-127
S <u>e</u> t view	set view parameters: centre, rotation angles, size and perspec- tive	page 3-147
<u>R</u> efresh	refresh the picture	page 3-147
<u>P</u> arts of the display	toggle visibility of lines, solids, mesh, etc.	page 3-166
Views	predefined views	page 3-147
Contours	display contours of boundary condition values and volume properties, etc.	page 3-63
Vectors	display vectors or material and volume properties	page 3-159
Change col-	change colours used for the display	page 3-56
ours		
Title	add a title or time and date to the display	page 3-149

Create

Insert from file	insert components from another 3D file (page 3-96) o model geometry from an OPERA-2d data file (page 3	r insert -91)
<u>O</u> bject	create a new primitive volume: block (page 3-36), cyl (page 3-65), prism (page 3-120), sphere (page 3-137) (page 3-151)	inder or torus
<u>C</u> onductor	create a new conductor: solenoid (page 3-132), racetra 121), bedstead (page 3-29), helical end (page 3-85), c perimeter end (fitted page 3-82 or tangential page 3-14 (page 3-138), arc (page 3-27) or brick (8-noded page 3 noded page 3-44) or import (page 3-90) or export (pag ductor files.	ack (page 3- onstant- 44), straight 3-41 or 20- e 3-73) con-
Local coordinate system	create a new local coordinate system	page 3-94
<u>V</u> ariables	create, edit and list numerical user variables	page 3-157
User defined <u>s</u> tring variables	create, edit and list string variables	page 2-23

Picking

🛃 Pick bodies	set picking filter to pick bodies	page 3-80
🏨 Pick cells	set picking filter to pick cells	page 3-80
🙀 Pick faces	set picking filter to pick faces	page 3-80
🛃 Pick edges	set picking filter to pick edges	page 3-80
🛃 Pick vertices	set picking filter to pick vertices	page 3-80
🗯 Pick LCS	set picking filter to pick local coordinate systems	page 3-80

Ç	Pick conductors	set picking filter to pick conductors	page 3-80
ĸ	Switch off picking	switch off picking	page 3-80
1	Pick entity	when an entity is picked, add it to the list of picked entities	page 3-80
1	Hide entity	when an entity is picked, hide it	page 3-80
	Show entity proper- ties	when an entity is picked, list its properties	page 3-80
	Pick any visible entity	anything visible can be picked OR only entities selected for display can be picked	page 3-80
	Pick selected enti- ties only		
80	Pick all filter type entities	pick all entities of the filter type	page 3-80
	Hide picked entities	hide all the picked entities	page 3-127
2	Apply filter to picked entities	remove from the list of picked entities any which do not match the current filter setting	page 3-114
<mark>19</mark> ,	Change type of picked entities	replace list of picked entities with those of current fil- ter type	page 3-114
	Reset picked entities	empty list of picked entities	page 3-114
	Unhide entities	unhide all hidden entities	page 3-127
`	Pick entities by property	pick all entities with a particular property	page 3-114
	Pick entities by position	pick entities of a particular type at specific coordi- nates	page 3-114

Operations

Set working coordinate system (WCS)	choose which coordinate system should be the working coordinate system	page 3-165
Transform WCS	transform the working coordinate system by displacement or rotation	page 3-152
Combine bodies	perform boolean operations on two or more picked bodies	page 3-58
Transform	transform one or more picked bodies by dis- placement, rotation, reflection of scaling	page 3-152
Сору	copy one or more picked bodies, cells or edges with displacement, rotation, reflection or scal- ing	page 3-152
Morphing	transform one or more picked bodies by twisting 155), stretching (page 3-140), bending (page 3-16) functional transformation (page 3-109)	g (page 3- 31) or by a

Sweep face	sweep one or more picked faces along a path to extend or create new bodies	page 3-141
Loft	create a new body between 2 picked faces	page 3-98
Blend or chamfer edges	blend of chamfer faces touching one or more picked edges	page 3-35
Delete	delete picked entities	page 3-67
Check	check the topology of one or more picked bod-	page 3-49
	ies	
Modify conductors	modify one or more picked conductors	page 3-61
Save picked components	save picked entities in an opc file	page 3-125
Export picked bodies	save picked entities in a sat file	page 3-125

Properties

Cell proper-	set properties of one or more picked cells	page 3-47
ties		
Face proper-	set properties of one or more picked faces	page 3-74
ties		
Edge proper-	set properties of one or more picked edges	page 3-71
ties		
Vertex proper-	set properties of one or more picked vertices	page 3-161
ties		
Set model compo-	specify whether picked bodies should be included in the	page 3-107
nents	model body	
Rename	rename one or more picked body or one picked local coor-	page 3-124
	dinate system	
Additional	add or remove labels on one or more picked entities	page 3-91
labels		
List properties	list the properties of the picked entities	page 3-95

Model

Symmetry condi-	connect faces which have symmetry boundary condi-	page 3-111
tions	tions	
Create model body	create a single model body for meshing and analysis	page 3-107
Generate surface	generate triangular mesh on all faces in the model body	page 3-104
mesh		
Generate volume	generate tetrahedral mesh in all cells in the model body	page 3-77
mesh		
Set material prop-	set properties for each material property label	page 3-100
erties		
Set BH curve prop-	set BH curve properties for each BH label	page 3-32
erties		

I

I

I

I

I

I

I

I

I

Set volume proper- ties	set properties for each volume property label	page 3-162
Set boundary con- ditions	set properties for each boundary condition label	page 3-37
Set circuit prop- erties	set properties of circuits	page 3-51
Set drive proper- ties	set drive functions for time varying analyses	page 3-68
Analysis settings	set options for analysis programs	page 3-23
Create analysis database	create an analysis database	page 3-135
Start analysis	start an analysis program	page 2-31
Launch post-proc-	start the post-processor	page 2-31
essor		
Delete model body	delete the model body to continue modelling	page 3-107

Windows

Tool <u>b</u> ars	choose which toolbars are visible	
<u>P</u> refer-	set console visibility and other window options	page 3-84
ences		

Help

Reference F1 Manual	Reference Manual	
About	show version number and support information	

Commands

Primitive Construction Commands

Command	Description
BLOCK	Creates a new cuboid body
CYLINDER	Creates a new cylinder / cone body
PRISM	Creates an n-sided prism or pyramid body
SPHERE	Creates a new spherical body
TORUS	Creates a new toroid

File And General Commands

Command	Description
CLEAR	Clears and resets all data from the model
END	Ends this session of the Modeller
HISTORY	Controls the size of the history stream
LOAD	Loads model data from a file
REDO	Returns changes that have been undone
SAVE	Saves model data to a file
UNDO	Returns the model to a previous state

Display And Selection Commands

Command	Description
COLOUR	Sets the colour for display of items
CONTOUR	Displays contours of components on displayed enti-
	ties
FILTER	Sets a filter and command for graphical selection
GUIOPTIONS	Set window preferences
HIDE	Hides entities from display
MOUSE	Changes the operation of the right mouse button
PICK	Picks specific model entities for modification
SELECT	Selects the criteria which determine what is
	included in the display
THREED	Creates a display window for model visualisation
TITLE	Adds date, time and title to the display
VECTOR	Display vectors: material properties, boundary con-
	ditions or current directions
WINDOW	Controls display of parts of the 3D view

Conductor Command

Command	Description
ARC	Creates or modifies an arc conductor
BEDSTEAD	Creates or modifies a bedstead conductor
BRICK8	Creates or modifies an 8-node brick conductor
BRICK20	Creates or modifies a 20-node brick conductor
FITTEDCPE	Creates or modifies a fitted constant perimeter end conductor
HELICALEND	Creates or modifies a helical end conductor
RACETRACK	Creates or modifies a racetrack conductor

Description
Creates or modifies a solenoid conductor
Creates or modifies a straight bar conductor
Creates or modifies a tangential constant perimeter end conductor

Local Coordinate System Commands

Command	Description
LCS	Creates a new Local Coordinate System (LCS)
WCS	Sets an Local Coordinate System as the Working Coordinate System (WCS)

Topology / Geometry Modification

Command	Description		
BEND	Bend a body by through a fixed angle		
BLEND	Adds blends or chamfers at edges		
CHECK	Checks the topology of parts of the model		
COMBINE	Combine (union, subtract, etc.) two or more bodies		
DELETE	Deletes a body or parts of a body		
LOFT			
MORPH	Apply any functional transformation		
PRECISIONDATA	Set geometric tolerances		
SWEEP	Sweeps a face to form a volume		
STRETCH	Stretch part of a body between 2 points		
TRANSFORM	Transforms or copies existing bodies, conductors		
	and Local Coordinate Systems		
TWIST	Twist part of a body between 2 points		

Data Commands

Command	Description		
BHDATA	Sets the BH data for a BHDATA label		
BOUNDARY	Sets the boundary conditions for a boundary label		
CELLDATA	Sets the properties associated with cells		
CIRCUIT	Define external circuits		
EDGEDATA	Sets the properties associated with edges		
FACEDATA	Sets the properties associated with faces		

Command	Description (continued)		
LABEL	Attaches a label to any item		
LIST	Lists data associated with any item		
MATERIALS	Sets the material properties for a material label		
RENAME	Renames a body or Local Coordinate System		
VERTEXDATA	Sets the properties associated with vertices		
VOLUME	Sets the properties for a volume label		

Analysis Database Preparation Commands

	-
Command	Description
ANALYSISDATA	Sets analysis options used by the analysis modules
DBCASEDATA	Sets simulation output frequencies and times
DRIVE	Sets drive information for sources for ELEKTRA and SOPRANO
FILL	Creates a volume mesh
MESH	Creates a surface mesh
MODEL	Creates (and deletes) a model suitable for analysis, from the components
PERIODICITY	Allows model periodicity to be used
SOLVERS	Creates a database for analysis
VARIABLE	Sets and stores user defined variables

The ANALYSISDATA Command

- *Summary* Sets the options for the analysis modules.
- Menu Route
 Model ↓

 Analysis settings

Command Line Parameters

Command	ANALYSISDATA			
Parameter	Default	Function		
OPTION	none	LOAD		
		SET		
PROGRAM	none	Analysis program type		
		TOSCA-	TOSCA: magnetic,	
		MAGN	electrostatic or current	
		TOSCAELEC	flow	
		TOSCACURR		
		SCALA	SCALA space charge	
			beam analysis	
		ELEKTRASS	ELEKTRA: Steady	
		ELEKTRATR	state harmonic, tran-	
		ELEKTRAVL	sient or velocity	
		SOPRANOSS	SOPRANO: Steady	
		SOPRANOEV	state harmonic or	
			eigenvalue	
		TEMPOST	TEMPO: steady state	
			thermal analysis	
		TEMPOTR	TEMPO: transient	
			thermal analysis	
LINEAR	YES	YES	Linear permeability	
			for analysis	
		NO	Non-linear permeabil-	
			ity (TOSCA magnetic	
			and ELEKTRA)	
NLITERTYPE	NEWTON	NEWTON	Use Newton-Raphson	
			non-linear updates	
		SIMPLE	Use simple updates	

Command	ANALYSISI	DATA (continued)		
Parameter	Default	Function		
NITERATIONS	21	Maximum number of non-linear itera- tions		
TOLERANCE	0.001	Non-linear iteration convergence toler-		
	1.0	ance		
		SCALA lelaxation factor		
RHS	ADAPTIVE	SIMPLE Simple coll integration		
			A lastice as it is to as	
		ADAPTIVE	tion	
HXEXT	0	Externally applied H field for TOSCA		
HYEXT	0	magnetic, SCALA and ELEKTRA		
HZEXT	0			
DRIVELABEL		Drive label for external fields in ELE- KTRA-SS and ELEKTRA-TR		
UPWINDING	YES	NO	No upwinding for ELEKTRA-VL analy- sis	
		YES	Use upwinding	
ITPTSTEP	21	Non-linear iterations per time step (ELEKTRA-TR)		
UPDATE	SIMPLE	ELEKTRA-TR time update method		
		SIMPLE	Simple theta method fixed step updates	
		AD2RK	Adaptive 2 nd order Runge-Kutta update	
		AD4RK	Adaptive 4 th order Runge-Kutta update	
		F2RK	Fixed 2 nd order Runge- Kutta update	
		F4RK	Fixed 4 th order Runge- Kutta update	
MAXADERR	1.0	Maximum error for adaptive updates		
DELTAT	0.01	Initial or fixed time step		
RPM	3000	Unused		
FREQ1	1.0e9	Lower range of SOPRANO-EV fre- quencies		
FREQ2	2.0e9	Upper range of SOPRANO-EV fre- quencies		
NEV	1	Number of eigenvalues to be found		
Command	ANALYSISDATA (continued)			
--------------	--------------------------	--	-------------------------------------	--
Parameter	Default	Function		
CONVTOL	1.0e-8	Convergence tolerance for linear equa- tion solutions		
PERIODICITY	YES	YES	Use periodicity	
		NO	Do not use periodicity	
POTENTIALCUT	YES	YES	Use automatic poten- tial cuts	
		NO	Do not use automatic potential cuts	
THERMALTEMP		Initial temperature for TEMPO-TR. Specify an expression or RT to use values already stored in the database.		

Notes

This command sets the options for the different analysis modules. Each module has its own independent set of data which is configured by this command.

Using **ANALYSISDATA OPTION=LOAD PROGRAM=TOSCAMAGN** will load the currently set options for the TOSCA magnetic analyses into the parameters. These can be set using **OPTION=SET** with new values for the parameters.

The ANALYSISDATA command sets various analysis options. From the menus, only those options relevant to the current simulation are available.

- Linear or Non-linear: Linear analyses use constant material properties; non-linear analyses update the material properties, depending on the solution and re-solve. Non-linear analyses can include some linear materials and linear analyses can include non-linear materials. In this latter case, the material property characteristic (e.g. BH curve) is used to supply the value of the material property (permeability). The slope of the curve at zero flux-density is used.
- **RHS Integrals**: The line and surface integrals of coil fields which are part of TOSCA magnetostatics and ELEKTRA analyses can be done using simple integrals, which only use the values of coil fields at the nodes or using adaptive integrals which use additional field points in order to reduce the errors.
- **External Fields** can be added to TOSCA magnetostatics, SCALA and ELEKTRA analyses. For ELEKTRA, a DRIVELABEL is needed to link to the drive function to be associated with the external fields.
- Automatic Potential cuts can be used in TOSCA magnetostatic analyses to automatically insert potential cuts to avoid having multiply con-

nected volumes, where a loop of total potential volume encloses a non-zero net current.

- **Periodicity** can be used in TOSCA and SCALA to avoid the need to build the full model where it is known that both field and geometry have rotational or translational symmetry.
- SCALA Iterations converge to a consistent set of particle trajectories and fields. The number of iterations and under-relaxation factor can be set.
- **Upwinding** is a technique to improve analysis of moving systems (ELEKTRA-VL). The analysis program reports whether upwinding is required or not.
- **Time stepping** in ELEKTRA-TR and TEMPO-TR can use fixed time steps or can adjust the time step to achieve a given accuracy (specified as a percentage).

ELEKTRA-TR only: If several similar models are to be analysed, the most efficient approach might be to use adaptive 4th order Runge-Kutta for one model to ascertain the appropriate time step to use. Subsequent models can then use a fixed time step method.

• **Initial temperature** in TEMPO-TR can be specified as a constant value or as RT to pick up values of temperature already stored in the database from an earlier analysis or added by the Post-processor TABLE command (page 5-101).

Each analysis program uses only a subset of the analysis parameters.

The **ARC** Command

Create or modify arc conductors.
Create \downarrow Conductor \rightarrow Arc Operations \downarrow Modify conductors \rightarrow Arc

Command Line Parameters

Command	ARC		
Parameter	Default	Function	
OPTION	NEW	NEW	Creates a new arc conductor
		MODIFY	Modifies properties of the
			picked arc conductors
		LOAD	Loads defaults from picked
			conductors
DRIVELABEL		Name for t	the arc drive label
LCNAME		Name for 1	Local Coordinate System for
		coordinate system 1	
SYMMETRY		Rotational symmetry about global Z axis	
XCEN2		Origin of coordinate system 2	
YCEN2			
ZCEN2			
THETA2		Euler angle	es defining orientation of coor-
PHI2		dinate system 2	
PSI2		-	
RXY		Reflection	symmetries in XY, YZ and ZX
RYZ		planes.	
RZX			
A		Cross-sect	ional width
В		Cross-sectional height	
R1		Radius of the arc	
ANGLE		Angle subtended by the arc	

Command	ARC (c	ontinued)		
Parameter	Default	Function		
CURD		Current de	nsity in the conductor. This can	
		be defined	in terms of the current as	
		current	/AREA	
TOLERANCE		Field calcu	lation tolerance	
INCIRCUIT		Is the cond	luctor part of an external cir-	
		cuit:		
		NO	The conductor has defined	
			current density.	
		YES	The current in the conductor	
			is determined by an external	
			circuit.	
REVERSE		Reverse the connections to this conductor		
		in its circuit: YES or NO.		
CIRCUITELEMENT		The name of circuit element this conduc-		
		tor is part of.		
KEEP	NO	NO	Clear the list of picked items	
		YES	Keep the list of picked items	
			for further modification	

Notes

This command creates a new arc conductor when using OPTION=NEW.

OPTION=MODIFY it is used to operate on the list of picked arcs, changing geometry, coordinate systems, symmetries, current density, tolerance etc. The current common values of the picked arc conductors are available by calling **CONDUCTOR OPTION=LOAD**.

The default values of the command parameters are updated to match common values shared by all of the picked arcs. If the picked arcs do not have a common value for a parameter, that parameter is left unset. Using OPTION=MODIFY will change the conductor data of all of the picked arcs to the new values given in the parameters, but will not affect the conductor data of any unset parameter.

For more information on the parameters, see "Circular Arcs" on page 4-36. For more information on connecting conductors to external circuits, see "The CIRCUIT Command" on page 3-51.

The **BEDSTEAD** Command

Summary	Create or modify bedstead conductors.
Icon	∽

Menu Route	Create \downarrow
	$\texttt{Conductor} \ \rightarrow \ \texttt{Bedstead}$
	Operations \downarrow
	Modify conductors $ ightarrow$ Bedstead

Command Line Parameters

Command	BEDST	EAD		
Parameter	Default	Function		
OPTION	NEW	NEW	Creates a new bedstead con-	
			ductor	
		MODIFY	Modifies properties of the picked bedstead conductors	
		LOAD	Loads defaults from picked conductors	
DRIVELABEL		Name for t	he bedstead drive label	
LCNAME		Name for Local Coordinate System for		
		coordinate	system 1	
SYMMETRY		Rotational symmetry about global Z axis		
XCEN2		Origin of coordinate system 2		
YCEN2				
ZCEN2				
THETA2		Euler angles defining orientation of coor		
PHI2		dinate syst	em 2	
PSI2				
RXY		Reflection symmetries in XY, YZ and ZX		
RYZ		planes		
RZX				
XP1		Local coor	dinates of lower, inside corner	
YP1		1		
А		Cross-sect	ional width	

Command	BEDST	EAD (cont	inued)
Parameter	Default	Function	
В		Cross-secti	onal height
H1		Half-length	n of the straight
H2		Half length	n of the upright
R1		Radius of t	he arc
R2		Radius of t	he upper bend
CURD		Current der	nsity in the conductor. This can
		be defined	in terms of the current as
		current	/AREA
TOLERANCE		Field calcu	lation tolerance
INCIRCUIT		Is the cond	uctor part of an external cir-
		cuit:	
		NO	The conductor has defined
			current density.
		YES	The current in the conductor
			is determined by an external
			circuit.
REVERSE		Reverse the connections to this conductor	
		in its circuit: YES or NO.	
CIRCUITELEMENT		The name of circuit element this conduc-	
		tor is part of.	
KEEP	NO	NO	Clear the list of picked items
		YES	Keep the list of picked items
			for further modification

Notes

This command creates a new bedstead conductor when using OPTION=NEW.

OPTION=MODIFY it is used to operate on the list of picked bedsteads, changing geometry, coordinate systems, symmetries, current density, tolerance etc. The current common values of the picked bedstead conductors are available by calling **CONDUCTOR OPTION=LOAD**.

The default values of the command parameters are updated to match common values shared by all of the picked bedsteads. If the picked bedsteads do not have a common value for a parameter, that parameter is left unset. Using OPTION=MODIFY will change the conductor data of all of the picked bedsteads to the new values given in the parameters, but will not affect the conductor data of any unset parameter.

For more information on the parameters, see "Bedsteads" on page 4-30. For more information on connecting conductors to external circuits, see "The CIRCUIT Command" on page 3-51.

The **BEND** Command

Summarv	Bends a body	through a	specified	angle
Sterrery			~r	

Menu Route	Operations \downarrow
	Morphing \rightarrow Bend

Command Line Parameters

Command	BEND			
Parameter	Default	Function		
LCSNAME		Name of local coordinate system defining the bend orientation		
RADIUS		Radius of the bend		
ANGLE		Angle of the bend		
CENTRE	NO	NO YES	Centre the bend	
KEEP	NO	NO YES	Keep the body for further operations	

Notes

This command operates on a set of picked bodies and bends them around a cylinder. The local coordinate system given by LCSNAME specifies the axes around which the bend occurs. The portion of the body along the positive W axis of the LCS is bent around an axis parallel with the U axis, but shifted by a distance RADIUS along the V axis.

The bodies are bent through the ANGLE given. If the bodies to not extend far enough in the W direction, the bend may not complete the full angle specified.

If CENTRE=YES, the bend angle will be centred on the origin of the LCS. If CENTRE=NO, the bend will start at the origin of the LCS

The **BHDATA** Command

Summary	Assigns BH curve	properties to a BH data label
---------	------------------	-------------------------------

Menu Route	Model \downarrow			
	Set	BH	Curve	properties

Command Line Parameters

Notes

Command	BHDAT	A			
Parameter	Default	Function			
OPTION		ADD	Adds a point to the end of the BH		
			curve		
		DELETE	Deletes a BH curve label		
		EDIT	Starts the BH editor window		
		INIT	Clears the data from the BH curve		
		LIST	Lists the BH data		
		LOAD Loads BH data from file			
		NEW Creates a new BH data label			
		VIEW	Starts the BH viewer window		
LABEL		BH data label to be used by the command.			
Н		New value of H to be added			
В		New value of B to be added			
CGS	YES	NO	The BH curve will be defined in CGS		
		YES	units		
FILE		File name	of BH data curve to be loaded		

The **BHDATA** command allows editing of BH data associated with BH curve labels.

For all commands, the label being edited or created should be specified in the LABEL parameter.

OPTION=NEW will create a new BH data label.

OPTION=EDIT will start the interactive BH editor window (note that this editor does not directly operate the **BHDATA** command, but will generate a consistent set of commands upon closure).

OPTION=INIT will clear the BH data associated with the label. The units being used by the curve should be specified on the command line with this option.

OPTION=ADD adds a point to the end of the BH data associated with the label.

OPTION=DELETE deletes a BH curve associated with a label.

OPTION=LIST will list the data associated with the label.

OPTION=LOAD will load data from a BH data file (*.bh*).

OPTION=VIEW will start the interactive BH viewer.

BH Viewer

The BH viewer is an interactive tool for viewing properties of the BH data associated with a label. The viewer allows views of different graphs generated from the data. These graphs include:

- **B** vs **H** data values
- Interpolated values of **B** vs **H**, over any range of **H** values.
- Interpolated values of M vs H
- Interpolated values of μ vs **H**

• Interpolated values of $\frac{\partial \mu}{\partial \mathbf{H}}$ vs **H**

The interpolated graphs are displayed for both positive and negative values of \mathbf{H} - \mathbf{H}_{c} , but will only be available if there are no errors found in the data. The graphs are interactive and the display can be zoomed using

- the left mouse button to drag a box for zooming a local area,
- moving the mouse up or down, with the middle mouse held down, zooms in or out on the centre of the screen.
- Moving the mouse with the right button held down, pans the picture. This can also be achieved using the scroll bars.

Zoom bounding box and previous views can be obtained from the view menu. Additionally a set view menu allows an explicit size to be defined, and allows the aspect ratio to be locked for subsequent zoom operations.

The units in which the data is displayed (CGS or SI) can be changed from the options menu.

The data being displayed can be exported to a new BH data file (.*bh*).

BH Editor

The BH editor incorporates all the facilities of the BH viewer.

In addition it provides a spreadsheet style interface in which the underlying data can be edited.

- Individual values can be changed by editing the entry in the table.
- Data can be imported from a BH data file (.*bh*)
- Changes can be undone or re-applied.
- Data from other sources may be copied in from the Windows clipboard if it is in the correct format (tab delimited data). Data may be exported in similar format.

All changes automatically update the graphs in the viewer.

The **BLEND** Command

Summary	Creates a blend or cha	amfer at edges betwee	en adjoining faces.
---------	------------------------	-----------------------	---------------------

Menu Route	Operations \downarrow	
	Blend or chamfer edg	ges

Command Line Parameters

Command	BLEND			
Parameter	Default	Function		
OPTION	BLEND	BLEND Create a blend		
		CHAMFER	Create a chamfer	
RADIUS		Radius of the	blend	
LEFTCHAMFER		Distance to chamfer from the edge on the first face		
RIGHTCHAMFER		Distance to chamfer from the edge on the adjoining face		

Notes

This command operates on a set of picked edges. The edges must be nonmanifold (i.e. there must not be more than two faces meeting at the edge).

If OPTION=BLEND, then for each picked edge a tangential join is formed. The join is cylindrical in nature, with radius given by the parameter RADIUS.

If OPTION=CHAMFER, the edge is planed off to give a planar face join between the two faces, with each of the original faces cut by the LEFTCH-AMFER or RIGHTCHAMFER distances. At present there is no way to distinguish which value is applied to which face, so the operation must be UNDONE and re-applied with the values switched if they are incorrect at the first attempt.

When applying the command to multiple edges, the result may differ from the result of applying the command to each edge individually. The **BLEND** command may fail if no suitable blend or chamfer surface can be found.

The **BLOCK** Command

Summary	Creates	a cuboid	block f	rom the	data sup	plied.
Summury	Creates	a cubbia	DIOUK I	ioni uic	uata sup	pheu.

Icon



Menu Route

 $\begin{array}{c} \texttt{Create} \ \downarrow \\ \texttt{Object} \ \rightarrow \ \texttt{Block} \end{array}$

Command Line Parameters

Command	BLOCK	
Parameter	Default	Function
NAME		Attaches this name to the cuboid body formed
X0		Coordinates of corner of the cuboid
Y0		
Z0		
X1		Coordinates of opposite point in the cuboid
Y1		
Z1		

Notes The body formed must have non-zero area in the XY plane, i.e. X0 is not equal to X1, Y0 is not equal to Y1. If Z0 equals Z1 a planar body of zero volume will be created.

The coordinates specified are in the Working Coordinate System.

The **BOUNDARY** Command

 Menu Route
 Model ↓

 Set boundary conditions

Command Line Parameters

Command	BOUNDARY			
Parameter	Default	Function		
OPTION		PICK	Adds a boundary label to a list to be set	
		UNPICK	Clears the list of picked boundary labels	
		RESET	Clears the data from the picked bound- ary labels	
		MODIFY	Sets the data for the picked boundary labels	
		LIST	Lists the boundary conditions of picked labels	
		DELETE	Deletes the picked labels	
BOUNDARYLABEL		Boundary labe	l to be picked	

Command	BOUNDARY (continued)				
Parameter	Default	Function			
CONDITION		NONE	None		
		TANGMAGN	Tangential mag-		
			netic		
		NORMMAGN	Normal magnetic		
		NORMMAGP	Normal magnetic		
			with assigned con-		
			stant potential		
		TANGELEC	Tangential electric		
		NORMELEC	Normal electric		
		NORMELEV	Normal electric		
			with assigned con-		
			stant voltage		
		POTENTIAL	Functional mag-		
			netic scalar poten-		
			tiai Eurotional voltago		
			Mined netential		
		MIXED	condition		
		DPDN	Assigned derivative		
			of scalar potential		
		DVDN	Assigned derivative		
			of voltage		
		VECTOR	Assigned vector		
			potential		
		IVECTOR	Assigned incident		
		DADIATION	vector potential		
		RADIATION	Radiation		
		PEC	Perfect conductor		
		SYMMETRY	Symmetry bound- ary		
VOLTAGE		Functional electric	scalar potential		
DVOLTAGE		Derivative of electr	ic scalar potential		
MPOTENTIAL		Functional magneti	c scalar potential		
DMPOTENTIAL		Derivative of magnetic scalar potential			
AX		Components of ma	gnetic vector poten-		
AY		tial			
AZ					
INAX		Components of inc	ident magnetic vec-		
INAY		tor potential			
INAZ					

Command	BOUNE	DARY	(continued)		
Parameter	Default	Functio	on		
PMIXA		Mixed	derivative an	nd potential condi-	
PMIXB		tion co	efficients		
CMPOTENTIAL		Value f potenti	for constant 1 al	magnetic scalar	
CVOLTAGE		Value	for constant	voltage	
DRIVELABEL		Drivela of assig	Drivelabel for functional time variation of assigned values		
THERMALCONDITION		NONE		No condition applied	
		INSUL	ATOR	Perfect insulator	
		TEMPERATURE Specified temperature ture FLUX		Specified tempera- ture	
				Specified heat flux	
		TRAN	SFER	Specified heat transfer	
TEMPERATURE		Function specifying temperature for use with TEMPERATURE condition			
HEATFLUX		Function specifying heat flux for use with FLUX condition			
HEATTRANSFER		Function specifying heat transfer coefficient for use with TRANSFER condition			
AMBIENTTEMP		Function specifying ambient tempera- ture for use with TRANSFER condition			

Notes

This command defines the boundary conditions for use by the analysis programs.

A set of boundary labels is picked using the command repeatedly, with OPTION=PICK and a BOUNDARYLABEL specified. A boundary label can be removed from the set using OPTION=UNPICK. If no BOUNDA-RYLABEL is given, the set is emptied.

Issuing the command with OPTION=MODIFY will modify the properties of the set of picked boundary labels to the new values given in the parameters. The value of a property associated with the boundary labels is unchanged if the parameter is unset.

The default value of each of the parameters is updated to be the common value of all the picked boundary labels. If the data of one of these parameters is unset, or the picked boundary labels do not share the same value, then the parameter value is left clear.

OPTION=RESET will clear the properties associated with all of the picked boundary labels.

'The properties of all boundary labels can be listed using OPTION=LIST.

Boundary labels that are not used, i.e. have no face referencing them can be deleted using OPTION=DELETE. Deleting a boundary label that is in use will clear its properties.

All parameters can be specified. The value of **CONDITION** and **THER-MALCONDITION** determines which will be used.

The DRIVELABEL allows value and functional based boundary conditions to be driven in ELEKTRA-TR or assigned a phase lag in ELEKTRA-SS and SOPRANO-SS.

The **BRICK8** Command

Icon



Menu Route	Create \downarrow
	Conductor $ ightarrow$ 8-node brick
	Operations \downarrow
	Modify conductors $ ightarrow$ 20-node brick

Command Line Parameters

Command	BRICK	BRICK8		
Parameter	Default	Function		
OPTION	NEW	NEW	Creates a new 8-node brick conductor	
		MODIFY	Modifies properties of the picked 8-node brick conduc- tors	
		LOAD	Loads defaults from picked conductors	
DRIVELABEL		Name for the drive label		
LCNAME		Name for Local Coordinate System for coordinate system 1		
SYMMETRY		Rotational symmetry about global Z axis		
XCEN2		Origin of coordinate system 2		
YCEN2				
ZCEN2				
THETA2		Euler angles defining orientation of coor-		
PHI2		dinate sys	tem 2	
PSI2				
RXY		Reflection	n symmetries in XY, YZ and ZX	
RYZ		planes		
RZX				

Command	BRICK	(continued)		
Parameter	Default	Function		
XP1		First point	t defining the conductor corners	
YP1				
ZP1				
	I			
XP8		Last point	defining the conductor corners	
YP8				
ZP8				
CURD		Current density in the conductor. This can		
		be defined	l in terms of the current as cur-	
		rent/AREA		
TOLERANCE		Field calculation tolerance		
INCIRCUIT		Is the conductor part of an external circuit:		
		NO	The conductor has defined cur-	
			rent density.	
		YES	The current in the conductor is	
			determined by an external cir-	
			cuit.	
REVERSE		Reverse the connections to this conductor		
		in its circuit: YES or NO.		
CIRCUITELEMENT		The name of circuit element this conduc		
		tor is part of.		
KEEP	NO	NO	Clear the list of picked items	
		YES	Keep the list of picked items	
			for further modification	

This command creates a new 8-node brick conductor when using **OPTION=NEW**.

OPTION=MODIFY it is used to operate on the list of picked 8-node bricks, changing geometry, coordinate systems, symmetries, current density, tolerance etc. The current common values of the picked 8-node brick conductors are available by calling **CONDUCTOR OPTION=LOAD**.

The default values of the command parameters are updated to match common values shared by all of the picked 8-node brick conductors. If the picked 8-node bricks do not have a common value for a parameter, that parameter is left unset. Using OPTION=MODIFY will change the conductor data of all of the picked 8-node bricks to the new values given in the parameters, but will not affect the conductor data of any unset parameter.

Notes

For more information on the parameters, see "Bricks" on page 4-37. For more information on connecting conductors to external circuits, see "The CIRCUIT Command" on page 3-51.

The BRICK20 Command

icon

20

Menu Route	Create \downarrow
	Conductor $ ightarrow$ 8-node brick
	Operations \downarrow
	Modify conductors $ ightarrow$ 20-node brick

Command Line Parameters

Command	BRICK20			
Parameter	Default	Function		
OPTION	NEW	NEW	Creates a new 8-node brick conductor	
		MODIFY	Modifies properties of the picked 8-node brick conduc- tors	
		LOAD	Loads defaults from picked conductors	
DRIVELABEL		Name for the drive label		
LCNAME		Name for coordinate	Local Coordinate System for e system 1	
SYMMETRY		Rotational symmetry about global Z axis		
XCEN2		Origin of coordinate system 2		
YCEN2				
ZCEN2				
THETA2		Euler angles defining orientation of coor-		
PHI2		dinate system 2		
PSI2				
RXY		Reflection symmetries in XY, YZ and		
RYZ		ZX planes	3	
RZX				

³⁻⁴⁴

Command	BRICK2	20 (continued)		
Parameter	Default	Function		
XP1		First point defining the conductor corners		
YP1				
ZP1		-		
•••	1	1		
XP20		Last point	defining the conductor corners	
YP20		-		
ZP20		-		
CURD		Current density in the conductor. This		
		can be def	Fined in terms of the current as	
		current/AREA		
TOLERANCE		Field calculation tolerance		
INCIRCUIT		Is the conductor part of an external cir-		
		cuit:		
		NO	The conductor has defined current density.	
		YES	The current in the conductor is	
			determined by an external cir-	
			cuit.	
REVERSE		Reverse the connections to this conduc-		
		tor in its circuit: YES or NO.		
CIRCUITELEMENT		The name of circuit element this conduc-		
		tor is part of.		
KEEP	NO	NO	Clear the list of picked items	
		YES	Keep the list of picked items	
			for further modification	

Notes

This command creates a new 20-node brick conductor when using OPTION=NEW.

OPTION=MODIFY it is used to operate on the list of picked 20-node bricks, changing geometry, coordinate systems, symmetries, current density, tolerance etc. The current common values of the picked 20-node brick conductors are available by calling **CONDUCTOR OPTION=LOAD**.

The default values of the command parameters are updated to match common values shared by all of the picked 20-node brick conductors. If the picked 20-node bricks do not have a common value for a parameter, that parameter is left unset. Using OPTION=MODIFY will change the conductor data of all of the picked 20-node bricks to the new values given in the parameters, but will not affect the conductor data of any unset parameter. For more information on the parameters, see "Bricks" on page 4-37. For more information on connecting conductors to external circuits, see "The CIRCUIT Command" on page 3-51.

The **CELLDATA** Command

- *Summary* Sets properties of picked cells.
- Menu Route
 Properties ↓

 Cell properties

Command Line Parameters

Command	CELLDA	ТА		
Parameter	Default	Function		
OPTION	MODIFY	MODIFY	Applies new values to the	
			picked cells	
		RESET	Clears all data and sets to	
			the default values	
MATERIAL	See notes	Material label		
POTENTIAL	See notes	TOTAL	Potential type of the volume	
		REDUCED		
		VECTOR		
ELEMENTTYPE	See notes	LINEAR	Type of finite element in the	
		QUADRATIC	volume	
VOLUME	See notes	Volume property label		
SIZE	See notes	Mesh control size		
NORMALTOL	See notes	Maximum normal angle between mesh		
		nodes		
SURFACETOL	See notes	Maximum deviation of the mesh from the		
		surface		
LEVEL	See notes	Data storage level for the cell data		

Notes This command is used to set or clear the properties of all picked cells.

When a cell is created, it has default properties of

- MATERIAL=AIR,
- POTENTIAL=REDUCED
- ELEMTYPE=LINEAR.

These properties are also set when the cell's data is cleared using OPTION=RESET.

OPTION=MODIFY can be used to change the value of all of the picked cells. Any unset parameter values are not modified. The new value of parameters that have been set replace the existing values in the data attached to the cells.

Each time a cell is picked the value of each of the parameters is updated to be the common value of all the picked cells. If the data of one of these parameters is unset, or the picked cells do not share the same value, then the parameter value is left clear.

After issuing the command with OPTION=MODIFY, the list of picked items is cleared. Issuing the command with OPTION=RESET keeps the same set of picked objects.

The MATERIAL parameter controls the material label attached to a cell. The properties associated with such a label are set using the MATERIALS command.

The ELEMENTTYPE parameter allows the user to force the use of quadratic elements when creating the database. All elements in the cell will be quadratic if the database is created with mixed elements. See the SOLV-ERS command for greater detail.

The VOLUME parameter holds the volume property label that may be attached to a cell. The properties associated with such a label are set using the VOLUME command.

The SIZE, NORMALTOL and SURFACETOL parameters control the mesh size of elements in the cell and near the faces of the cell, when generating the surface and volume mesh.

The LEVEL parameter controls the storage of data when there is a conflict during the merging of multiple cells. The data set with the greater level will be maintained. The result of merging 2 cells with the same level is indeterminate.

The CHECK Command

Summary Checks a picked body for topological problems.

Menu Route Operations ↓ Check

Command Line				
Parameters	Command	COMM	ENT	
	Parameter	Default	Function	
	LEVEL	HIGH	LOW	Level of detail used in the checking
			MEDIUM	
			HIGH	
			FULL	
	REPAIR	NO	NO	Attempt to repair errors found during
				checking
			YES	
	FILENAME			Output the results of the checking to
				the specified file

Notes

I

I

This command checks for topological problems of a single picked body. If problems exist, the objects are given a SYSTEM label and information regarding the problem is also attached to the body, or constituent parts of the body. This can be seen by listing the data on the object through the FIL-TER COMMAND=LIST command.

Generally the parameter LEVEL should be HIGH or FULL, as this will report most possible errors within the model. If errors exist, REPAIR=YES can be used to try to fix them. The repair might improve the condition of the model by improving the model's geometry and by creating tolerant entities. However, tolerant entities might themselves fail during some operations, so might not be a cure for all models. Very short edges and sliver faces will also be removed by the repair.

The **REPAIR** option can only be used on component bodies, not on the model body.

If problems exist with a body, it may be necessary to undo the operation that caused the problem and try an alternative method to generate the model. Failure to do this could cause the body to fail in future operations.

There are many reasons why a body fails the check command. The output of the check command can be output to a FILENAME if more details are required.

The **CIRCUIT** Command

Menu Route Model ↓

el \downarrow Set circuit properties

Command Line

Parameters

Command	CIRCUIT			
Parameter	Default	Function		
OPTION		INIT	Initialise the circuit dialog	
		CHOOSETYPE	Set the active ele- ment type in the circuit dialog	
		CHOOSEELEMENT	Set the active ele- ment in the circuit dialog	
		SETELEMENT	Create or modify a new circuit ele- ment	
		DELETEELEMENT	Delete a circuit ele- ment	
		NEWLOOP	Create a new loop	
		LOOPADD	Add an element to a current loop	
		LOOPDELETE	Remove an ele- ment from a loop	
		REVERSE	Reverse the direc- tion of an element in a loop	
ELEMENTNAME		Name of element to deleted etc.	be added, changed,	
TYPE		CAPACITOR	Capacitor	
		CURRENT	Current source	
		INDUCTOR	Inductor	
		RESISTOR	Resistor	
		VOLTAGE	Voltage source	
		WINDING	Circuit winding	

Command	CIRCU	(continued)
Parameter	Default	Function
RESISTANCE		Resistance of the resistive element
INDUCTANCE		Inductance of the inductive element
CAPACITANCE		Capacitance of the capacitive element
VOLTAGE		Voltage from the voltage source element
CURRENT		Current from the current source element
RPUL		Resistance Per Unit Length of the winding
		element
TURNS		Number of turns in the winding element
FILAMENTS		Number of representative filaments in the
		winding element
SYMMETRY		Symmetry adjustment factor needed to cre-
		ate the full winding
DRIVELABEL		Name of the drive function controlling the
		source strength
LOOPNAME		Name of the current loop being controlled
LOOPELEMENT		Name of active element in the current loop
NEWLOOPNAME		Name for a new loop being created
INITIALCAPVOLT		Initial voltage on capacitor

Notes

External circuits are defined using the **CIRCUIT** command. This command has two main roles.

- To define the external circuit elements. Resistors, capacitors, inductors, voltage or current sources and the properties of the circuit conductors defined in the model.
- To specify the connectivity of the elements in the external circuit by defining a set of current loops.

The creation of components and the specification of the circuit loops can be performed at any time before the analysis database is created. There are however two exceptions. Before a current loop can be created all the components forming the loop must be defined. The external circuit conductors must be created and their properties defined before the model body is generated. This second rule is a consequence of the method used to represent the external circuits within the finite element analysis.

A circuit element is created by issuing the command with OPTION=SETELEMENT. The TYPE parameter is used to define whether the element is a resistor, inductor etc. and the ELEMENTNAME parameter is used to specify a unique name for the component. Any properties of the component can be set by assigning a value to the appropriate

parameters for the component being created. For example to create a new 100 ohm resistor named R1 the following command can be issued,

CIRCUIT OPTION=SETELEMENT TYPE=RESISTOR, ELEMENTNAME='R1' RESISTANCE=100

Five different external circuit element types can be defined,

- Resistors. The resistance of the component is defined using the **RESISTANCE** parameter.
- Inductors. The inductance of the component is defined using the **INDUCTANCE** parameter.
- Capacitors.

The capacitance of the component is defined using the CAPACI-TANCE parameter. In transient problems, capacitors can have an initial voltage applied using INITIALCAPVOLT.

• Voltage sources.

The VOLTAGE parameter can be used to set the supply voltage. An optional drive label can also be defined, DRIVELABEL.

• Current sources.

The **CURRENT** parameter can be used to set the current. An optional drive label can also be defined, **DRIVELABEL**.

• Windings.

Windings form the link between the finite element model and the external circuit. Windings are represented in the model by a conductor or a set of conductors that have been created as an external circuit component with a circuit element name.

Four parameters need to be defined for each winding. The resistance per unit length of the winding, defined through the RPUL parameter and the number of turns in the winding, defined using the TURNS parameter. The FILAMENTS parameter control the representation of the winding in the finite element mesh. For example FILAMENTS=1 means that the winding will be represented in the mesh by a single filament through the centre of the conductor. To improve the accuracy of models a higher number of filaments can be used. FILAMENTS=2 will give a set of 4 filaments etc. The final parameter, SYMMETRY, can be used to adjust the model when symmetry boundary conditions are used and the external circuit conductor does not lie entirely within the finite element mesh. If half the winding is included in the mesh then SYMMETRY=2, if a quarter then SYMMETRY=4 etc.

Calling the CIRCUIT command with OPTION=SETELEMENT can be used to modify the properties of components that have already been defined. A component can be deleted by issuing the command with **OPTION=DELETEELEMENT** and naming the element to be deleted using the **ELEMENTNAME** parameter.

The external circuit is defined in terms of circuit loops. These loops are created using OPTION=NEWLOOP and giving a unique name for the loop through the NEWLOOPNAME parameter. For example, the following line will create a new loop named Loop1.

CIRCUIT OPTION=NEWLOOP NEWLOOPNAME='LOOP1'

Components are added to a loop using the command with OPTION=LOO-PADD. The loop to which the component is to be added is specified by the LOOPNAME parameter and the component to be added is given by the ELEMENTNAME parameter. The component is positioned in the loop immediately after the element given by LOOPELEMENT. For example, to add the resistor R1 at the start of the loop LOOP1,

CIRCUIT OPTION=LOOPADD ELEMENTNAME='R1', LOOPNAME='Loop1' LOOPELEMENT=

A component can be removed from the loop using OPTION=LOOPDE-LETE, as is shown in the following example,

CIRCUIT OPTION=LOOPDELETE LOOPNAME='Loop1', LOOPELEMENT='R1'

If a loop is no longer required it can be removed by leaving the LOOPE-LEMENT parameter empty in the line above.

The **CLEAR** Command

Menu Route	File \downarrow	
	Close	
	File \downarrow	
	Rever	t to saved

Command Line Parameters

Command	CLEAR		
Parameter	Default	Function	
REVERT	NO	NO	Clears and re-initialises all data within the Modeller
		YES	Clears all data and reloads the previ- ously open file

Notes This command is used to reset the Modeller to its initial state. If REVERT=YES, the open file is re-loaded to remove changes from the model that are not required.

All history states are cleared, so this operation cannot be undone.

The **COLOUR** Command

Summary	Sets the colour attac	hed to a display label
		1 2

Icon



Menu Route

View \downarrow Change colours

Command Line Parameters

Command	COLOU	IR		
Parameter	Default	Function		
OPTION	SET	Option:		
		LOAD	Load current values as defaults	
		SET	Set new values	
CODE		Code number for the colour to be altered		
PROPERTY		Property type for item to be set		
LABEL		Individual label of property		
RED		Value of red component (0 - 255)		
GREEN		Value of green component (0 - 255)		
BLUE		Value of blue component (0 - 255)		
TRANSLUCENT	NO	NO	Switch off colour translucency	
		YES	Set colour property to be translu-	
			cent	

Notes

This command sets the COLOURs used for the display of the model.

Properties displayed within the Modeller are assigned an integer colour CODE. If this code number has been specified as a parameter this will be used. If not, a property and label can be specified, and the code for this property will be found and used, e.g. PROPERTY=MATERIAL and LABEL=AIR will change the colour of the display of items with material label air.

With OPTION=SET, the colour is changed to the values of RED, GREEN, BLUE and TRANSLUCENT supplied. If any value is not given, this component remains unchanged.

Making a material colour translucent (TRANSLUCENT=YES) makes other materials inside it visible. Only one material (or all the contour colours) can be translucent at a time. All other materials will be opaque (TRANSLUCENT=NO).

The **COMBINE** Command

Carron and and	Combinad	michad	hadiaa	maina	a haalaan	onaration
Summary	Combines	DICKEU	Doules	using	a boolean	operation
~						

Menu Route Operations ↓ Combine picked bodies

Command Line Parameters

Command	COMBINE			
Parameter	Default	Function		
OPTION	none	UNION	Forms the union of the picked bodies	
		INTERSECT	Returns the intersection of the bodies	
		SUBTRACT	Subtracts all other picked bodies from the first body picked	
		TRIM	Subtracts images of other picked bodies from the first body, leav- ing these intact	
		CUTAWAY	Leaves the first picked body unaf- fected, but subtracts it from all other picked bodies	
REGULAR YE	YES	YES	Regularise the result of the boolean operation	
		NO	Do not regularise the result	

Notes

This command combines all picked bodies using the specified operation. At least 2 bodies must be picked for these operations.

UNION will form the sum of the picked bodies in a single body. After the operation the original bodies no longer exist within the model.

INTERSECT keeps the common volume that exists within all picked bodies. After the operation the original bodies no longer exist within the model.

SUBTRACT returns a body containing cells occupying the volume that exists in the first body picked, and which does not exist in any of the subsequently picked bodies. After the operation the original bodies no longer exist within the model.

TRIM leaves all picked bodies except the first untouched, and trims the first body so that it has no overlap with the other picked bodies.

CUTAWAY leaves only the first picked body untouched, and cuts away the first body from each of the others, so that it has no overlap with the other picked bodies.

If **REGULAR=YES**, faces, edges and vertices that are affected by the boolean operation are checked to see if they are needed to support the geometry. If not, they are removed and the body simplified. Internal faces formed by the operation are not needed to support the geometry, and so are removed. Faces bounding cells that have no volume, i.e. sheets, are also removed. Edges that bound two faces with the same underlying geometry are removed.

If the bodies do not overlap, the **INTERSECT** operation will have the effect of removing all the picked bodies.

If the first picked body is completely surrounded by the other picked bodies, the **SUBTRACT** operation will also remove all picked bodies.

It should be noted that the non-regular operations may leave sheet faces. Such sheet faces can be picked and deleted using the DELETE command (page 3-67).

The **COMMENT** Command

Summary Adds comments into the database when creating a new simulation for analysis.

Command Line Parameters

Command	COMMEN	IT		
Parameter	Default	Function		
TEXT	none	Line of text to be added to the stored comment text		
CLEAR	NO	NO	Add additional text to the stored com- ments	
		YES	Clears existing stored text	
TYPE	DBTITLE	DBTITLE	Type of comment	

Notes

This command adds text as a comment for storing with a database simulation. These comments can be used to add information about the simulation for later reference.

Lines of text are added into a stored string. This stored string can be cleared using CLEAR=YES.
The **CONDUCTOR** Command

Summary	Modifies data common to all picked conductors of any type.
Menu Route	Operations \downarrow

Modify conductors \rightarrow Any conductor type

Command Line Parameters

Command	CONDUC	CTOR	
Parameter	Default	Function	
OPTION	none	MODIFY	Modifies properties of the picked conductors
		LOAD	Loads defaults from picked conductors
DRIVELABEL		Name for	the conductor drive label
LCNAME		Name for l coordinate	Local Coordinate System for system 1
SYMMETRY		Rotational axis	symmetry about global Z
XCEN2		Origin of a	coordinate system 2
YCEN2			
ZCEN2			
THETA2		Euler angl	es defining orientation of
PHI2		coordinate	e system 2
PSI2			
RXY		Reflection	symmetries in XY, YZ and
RYZ		ZX planes	
RZX		-	
CURD		Current de	ensity in the conductor. This
		can be defi	ined in terms of the current as
		current	AREA
TOLERANCE		Field calcu	ulation tolerance

Command	CONDUC	CTOR (co	ntinued)
Parameter	Default	Function	
INCIRCUIT		Is the cond cuit:	luctor part of an external cir-
		NO	The conductor has defined current density.
		YES	The current in the conductor is determined by an external circuit.
REVERSE		Reverse th tor in its ci	e connections to this conduc- rcuit: YES or NO.
CIRCUITELEMENT		The name ductor is p	of circuit element this con- art of.
KEEP	NO	NO	Clear the list of picked items
		YES	Keep the list of picked items for further modification

Notes

This command operates on the list of picked conductors of any type, and can be used to change common properties such as coordinate systems, symmetries, current density, tolerance etc.

This command cannot be used to change the parameters defining the geometry of the conductors. To modify the geometry of e.g. a solenoid, the SOLENOID command should be used.

The default values of the command parameters are updated to match common values shared by all of the picked conductors. If the picked conductors do not have a common value for a parameter, that parameter is left unset. **CONDUCTOR MODIFY** will change the conductor data of all of the picked conductors to the new values given in the parameters, but will not affect the conductor data of any unset parameter. The current common values of the picked conductors are available by calling **CONDUCTOR OPTION=LOAD**.

For more information on the parameters, see "The CONDUCTOR Subcommand MODIFY" on page 4-44. For more information on connecting conductors to external circuits, see "The CIRCUIT Command" on page 3-51.

The **CONTOUR** Command

Summary	Displays contour of attached data on the model.
---------	---

Menu Route View ↓ Contours

Command Line Parameters

Command	CONTC	UR	
Parameter	Default	Function	
COMPONENT	NONE	NONE	No contours will be dis- played
		AX	Magnetic vector potential
		AY	
		AZ	
		AMOD	
		CHARGE	Charge density
		CMPOTENTIAL	Constant magnetic poten- tial
		CVOLTAGE	Constant voltage
		DMPOTENTIAL	Normal derivative of mag- netic potential
		DVOLTAGE	Normal derivative of volt- age
		HEATVALUE	Applied volume heat source
		JX	Applied source current density
		JY	
		JZ	
		JMOD	
		MPOTENTIAL	Magnetic potential
		PACKING	Packing factor
		ROTATION	Rotational velocity
		TEMPERATURE	Temperature

I

I

Command	CONTC	UR (continued)	
Parameter	Default	Function	
COMPONENT cont		VOLTAGE	Voltage
		VX	Applied linear velocity
		VY	
		VZ	
		VMOD	

Notes

This command displays contours of the **COMPONENT** on surfaces of objects in the model. If an object has data containing the component, then the value of the data will be contoured over it. Other displayed objects will be displayed normally. The contour command options are based on data that can be set on cells and faces only.

If the **CONTOUR** command is used to display functional components, the picture on the screen depends on the existence of a surface mesh.

With the surface mesh generated, the display will be based on the mesh that has been formed, and it will show the distribution of the component which is passed on to the analysis module.

If the surface mesh is not formed yet, the display is based on the surface display facets, which may not represent the final mesh very well. In such a case, a large non-linearity in functional definitions of these values may not be seen over the surface.

The **CYLINDER** Command

Summary	Creates a cylinder or cone, with either circular or elliptic base.
Icon	
Menu Route	Create \downarrow Object $ ightarrow$ Cylinder / cone

Command Line
Parameters

Command	CYLINE	DER
Parameter	Default	Function
NAME		Attaches this name to the body formed
X0		X coordinate of central point on the base
Y0		Y coordinate of central point on the base
Z0		Z coordinate of central point on the base
X1		X coordinate of central point on the top
Y1		Y coordinate of central point at the top
Z1		Z coordinate of central point at the top
MAJORRADIUS		Major radius of the base of the cylinder
MINORRADIUS		Minor radius of the base of the cylinder
TOPRADIUS		Radius at the top of the cylinder

Notes

All values must be given. The axis of the body is formed between points (X0,Y0,Z0) and (X1,Y1,Z1) and the normals of the base and top planes are parallel to this. The base point and top point must be different, and the two base radii given must be greater than zero.

If the **TOPRADIUS** is given as zero, the body formed is a cone.

If the MAJORRADIUS is not the same as the MINORRADIUS, the body formed has an elliptic base.

The coordinates specified are in the Working Coordinate System.

The **DBCASEDATA** Command

Summary Sets the list of simulations frequencies for ELEKTRA-SS and SOPRANO-SS and the list of output times for ELEKTRA-TR analyses.

Menu Route	Model \downarrow
	Analysis settings

Command Line	Command	DBCAS	EDATA	
Parameters	Parameter	Default	Function	
	PROGRAM	none	ELEKTRASS	Set frequencies for ELEKTRA-SS
			ELEKTRATR	Set output times for ELEKTRA- TR
			SOPRANOSS	Set frequencies for SOPRANO-SS
			TEMPOTR	Set output times for TEMPO-TR
	OPTION	ADD	ADD	Add a new frequency / output time to the list
			INSERT	Insert a frequency at a position in the list
			DELETE	Delete an item from the list
			REPLACE	Replace an item in the list
			LIST	Lists the set of frequencies / out- put times.
	INDEX	1	The position at v	which to insert, delete or replace
	VALUE		The new value f	or add, insert or replace

Notes

I

I

This command creates a list of frequencies or output times for the different analysis programs. Each program has a separate list of values.

- ELEKTRA-SS or SOPRANO-SS data bases will be analysed for each of the frequencies given in the list.
- An ELEKTRA-TR or TEMPO-TR solution will be output at each of the times in the list. The output time list is automatically chronologically ordered, so that insert and add have the same effect. An output time at t=0 is also added.

All lists automatically remove duplicates.

The **DELETE** Command

Summary Deletes the picked entities

Menu Route Operations ↓ Delete

Command Line Parameters

Command	DELETI		
Parameter	Default	Function	
REGULARISE	YES	YES	Regularise the result
		NO	Do not regularise the result
EXTERNAL	NO	YES	Allow external faces to be deleted
		NO	Do not delete external faces

Notes

This command tries to **DELETE** all picked objects, including Local Coordinate Systems and conductors.

If bodies have been picked, these will always be deleted. If cells have been picked, the delete command will remove the cells from the body and delete them. If a body only has a single cell, deleting the cell has the same effect as deleting the body.

If a face to be deleted is internal to a body, or if the face does not form part of a cell, i.e. both sides are external, the face can be deleted. If the face is internal, the two cells to either side of the face will be merged. If regularise is on, the edges and vertices that formed the boundary of the face are tested. If they are not required to support the geometry, they are deleted as well.

Faces forming the external boundaries of a body can be deleted only if the gap formed can be patched by the neighbouring faces. Therefore, one side of a cube could not be deleted because the neighbouring faces would be unable to patch it. The parameter **EXTERNAL=YES** must be used to allow these faces to be deleted.

Edges and vertices may be deleted if they are not required to support the topology of the body, e.g. if the face to both sides of an edge has the same underlying geometric surface.

The **DRIVE** Command

Summary	Sets the properties	associated w	ith a drive label.
---------	---------------------	--------------	--------------------

 Menu Route
 Model ↓

 Set drive properties

Command Line

Parameters

Command	DRIVE		
Parameter	Default	Function	
OPTION		PICK	Adds a drive label to a list to be set
		UNPICK	Clears the list of picked drive labels
		RESET	Clears the data from the picked drive labels
		MODIFY	Sets the data for the picked drive labels
		LIST	Lists the drive data of picked labels
		DELETE	Deletes the picked labels
DRIVELABEL		Drive label t	to be picked
TYPE		DC	Drive function associated with the label
		STEP	Step at time zero
		RAMP	Ramp drive
		SINE	Sine drive
		COSINE	Cos drive
		PEAK	Rise to peak value before decaying
		RISE	Exponential rise function
		TABLE	Switch on table
		TOFF	Switch off table
SSPHASE		Phase lag us and SOPRA	ed during ELEKTRA-SS NO-SS analysis
SINFREQUENCY		Frequency for	or Sine drives
SINPHASE		Phase for Si	ne drives
COSFREQUENCY		Frequency for	or Cosine drives

Command	DRIVE	(continued)
Parameter	Default	Function
COSPHASE		Phase for Cosine drives
STEPTIME		Unused
RAMPTIME		Time taken to ramp
PEAKTIME		Time at which peak occurs
RISETIME		Time constant for the rise
TABLEFILE		File for switch on timetable information
TABLEOFFFILE		File for switch off timetable information
ROTATING		Not used at present
SCALE		Drive scaling factor

Notes

This command defines the drive functions for use by the analysis programs:

- ELEKTRA-TR: transient drives
- ELEKTRA-SS and SOPRANO-SS: scaling factors and phase angles
- ELEKTRA-VL and TOSCA: scaling factors

A set of drive labels is picked using the command repeatedly, with OPTION=PICK and a DRIVELABEL specified. A drive label can be removed from the set using OPTION=UNPICK. If no DRIVELABEL is given, the set is emptied.

Issuing the command with OPTION=MODIFY will modify the properties of the set of picked drive labels to the new values given in the parameters. The value of a property associated with the drive labels is unchanged if the parameter is unset.

The default value of each of the parameters is updated to be the common value of all the picked volume labels. If the data of one of these parameters is unset, or the picked drive labels do not share the same value, then the parameter value is left clear.

OPTION=RESET will clear the properties associated with all of the picked drive labels.

The properties of all drive labels can be listed using OPTION=LIST.

Drive labels that are not used, i.e. have no boundary condition or conductor referencing them, can be deleted using OPTION=DELETE. Deleting a drive label that is in use will reset its properties.

ELEKTRA-SS and SOPRANO-SS use the SSPHASE parameter information. If this is unset, it is assumed to be 0. ELEKTRA-SS, ELEKTRA-VL, SOPRANO-SS and TOSCA use the **SCALE** parameter. If this is unset, it is assumed to be 1.

ELEKTRA-TR drives are mainly controlled by the TYPE parameter. Which of the other parameters are used is determined from the option assigned to type. If unset, a TYPE=DC is assumed.

The **EDGEDATA** Command

Summar	v Sets	properties	of r	bicked	edges.
Summu	y	properties	or p	nencu	cuges.

Menu RouteProperties ↓Edge properties

Command Line Parameters

Command	EDGEDA	TA			
Parameter	Default	Function			
OPTION	MODIFY	MODIFY	Applies new values to the picked edges		
		RESET	Clears all data from the picked edges		
SIZE	See notes	Mesh cont	trol size		
LEVEL	See notes	Data stora	Data storage level for the edge data		

Notes

This command is used to set or clear the properties of all picked edges.

Edges initially have no data assigned to them.

If issuing the command with OPTION=MODIFY, new values of parameters that have been set replace the existing values of data on the edges. The data on the edges, associated with any unset parameters, is left unchanged.

The default value of each of the parameters is updated to be the common value of all the picked edges. If the data of one of these parameters is unset, or the picked edges do not share the same value, then the parameter value is left clear.

SIZE is the maximum length of an element side along the edge.

The LEVEL parameter controls the storage of data when there is conflict during the merging of multiple edges. The data set with the greater level will be maintained. The result of merging 2 edges with the same level is indeterminate.

Upon issuing the command with OPTION=MODIFY, all picked items are de-selected. Issuing the command with OPTION=RESET keeps the same set of picked objects.

The END Command

Summary	Ends the current	Modeller	session.
-			

Exit

File \downarrow

Menu Route

Command Line Parameters				
	Command	END		
	Parameter	Default	Function	
	none		·	

This command ends the current session of the Modeller. If there have been changes since the last SAVE OPTION=ALL or SAVE OPTION=NEW command, then confirmation of the end command will be required.

Notes

The **EXPORT** Command

Summary	Exports conductor data to a file for use by the Modeller, pre, or post proc-
	essor.

Menu Route	Create \downarrow
	$\texttt{Conductor} \ \rightarrow \ \texttt{Export}$

Parameters	Command	EXPORT			
	Parameter	Default	Function		
	FILE		Conductor file to be created		

Notes

The EXPORT command writes a file containing all conductors in the model. This file can be read by the pre processor READ command (page 4-146), the post processor CONDUCTOR command (page 5-40), and the Modeller IMPORT command (page 3-90).

The conductor data in the file contains the values of the origin position and Euler angles defining coordinate system 1, not the name of the Local Coordinate System with which the conductor is associated.

The FACEDATA Command

Summary	Sets	properties	of	picked	faces
		1 1			

Menu RouteProperties ↓Face properties

Command Line

Parameters

I

I

I

Command	FACEDAT	ΓA	
Parameter	Default	Function	
OPTION	MODIFY	MODIFY	Applies new values to
			the picked faces
		RESET	Clears all data from the
			picked faces
BOUNDARYLABEL	See notes	Boundary label	on the face
ELEMENTTYPE	See notes	LINEAR	Elements touching the
			face will be linear
		QUADRATIC	Elements touching this
			face will be quadratic
SIZE	See notes	Mesh control si	ize
NORMALTOL	See notes	Maximum norn	nal angle between mesh
		nodes	
SURFACETOL	See notes	Maximum devi	ation of the mesh from
		the surface	
LEVEL	See notes	Sets the data storage level for the face	
		data	
FORLAYERS	0	Number of forward layers from the	
		face	
FOROFFSET		Distance betwe	en each forward layer
BACKLAYERS	0	Number of backwards layers from the	
		face	
BACKOFFSET		Distance betwe	en each backward layer

Notes

This command is used to set or clear the properties of all picked faces.

Faces initially have no data assigned to them, and OPTION=RESET will clear all data of the picked faces.

Issuing the command with OPTION=MODIFY will modify the properties of the picked faces to the values given in the parameters. The value of properties on the picked faces are unchanged if the parameter is unset.

The default value of each of the parameters is updated to be the common value of all the picked faces. If the data of one of these parameters is unset, or the picked faces do not share the same value, then the parameter value is left clear.

Upon issuing the command with OPTION=MODIFY, the list of picked items is cleared. Issuing the command with OPTION=RESET keeps the same set of picked objects.

The BOUNDARY parameter allows a boundary label to be attached to a face. The properties of such a label are set or modified using the BOUND-ARY command.

The ELEMENTTYPE parameter allows the user to force the use of quadratic elements when creating the database. Any elements that touch the face will become quadratic if the database is created with mixed surface elements. See "The SOLVERS Command" on page 3-135 for greater detail.

The SIZE, NORMALTOL and SURFACETOL parameters control the mesh size of elements on and near the face, when generating the surface and volume mesh.

The LEVEL parameter controls the storage of data when there is conflict during the merging of multiple faces. The data set with the greater level will be maintained. The result of merging 2 faces with the same level is indeterminate.

Layering creates copies of the face, using an identical mesh topology on each layer. This should help improve the quality of the mesh generated in very thin layers, and should make the modelling of thin surface regions easier. Any planar, spherical or cylindrical face may be layered. When creating the model body, a number of copies of the faces are introduced both in front and behind the original face (FORLAYERS and BACKLAYERS). The offset between each layer is specified by FOROFFSET and BACKOFF-SET. This value may be a function of the system variable LAYER, i.e. forlayers=3, foroffset=2**layer would produce 3 copies of the face with exponentially increasing gap thickness (2, 4, 8) between each layer.

Where interaction of the layers with other parts of the model occur, the original face will be modified to ensure that the face and its layered copies have a matching topology. Layers that extend outside the model body will be removed. The use of layering should only be used where necessary, as

the additional operations when creating the model body will increase the time and complexity of operations needed to form the model body.

The **FILL** Command

Summary	Fills the model with a mesh of tetrahedral elements.
Menu Route	Model ↓
	Generate volume mesh

Command Line Parameters Command FILL Parameter Default Function TOLERANCE 1.0e-6 Tolerance used for checking for equivalent points **Notes** This command generates the volume mesh and must be called prior to creating the database. The TOLERANCE parameter should not need to be adjusted in most cases, and can control the tolerance used for matching equivalent points. The cells inside the model body are meshed sequentially. On execution of the FILL command, the outline frame of the volume which is being processed is highlighted. A progress bar is displayed at the bottom of the Modeller window. This command may take a long time for complex models or large meshes. Volume mesh There are several types of failure that may be reported by the volume mesh generator. Unfortunately the error number and its associated message can-**Errors** not in general be related to the source of the problem. An inappropriate or poor surface mesh is often the reason why the volume Typical source of mesh generation fails. When the program fails to mesh a cell, the cell will the Problem be coloured and given the label VOLUME MESH ERROR, which can then be selected for viewing. The following checklist should be used to identify likely problems.

• View the surface mesh on the cell that failed to mesh

- Look for badly shaped triangles and consider changing the SIZE control parameters to improve the surface mesh.
- Pay particular attention to geometric details on the surface of the cell that failed to mesh. A common cause of problems is a detail with a tight curvature, attached to an adjoining face by a curved edge, shown in Figure 3.2.





This is a typical problem created by using only the normal tolerance to control mesh size, where the blend surface is being discretised with much smaller elements compared to the curved edge of the plate. The problem is easily avoided by using surface fitting tolerance to control the mesh size on the blend, see Figure 3.3.

- Look for areas with unexpectedly small elements. The surface mesh generator will create a dense mesh around very small area faces. When this happens it is often a result of small errors in the dimensions of the model.
- Thin shells with curved faces can cause problems because the meshes on two faces intersect each other. The surface fitting toler-



VECTOR FIELDS

Figure 3.3 Improved mesh using surface fitting tolerance to control the mesh size on the blend surface

ance can be used to avoid this problem, its value should be of the order of the thin dimension of the cell.

- Check the size of the cell compared to the surface mesh size.
 - Numerical precision becomes a problem if the element edge sizes vary by more than two orders of magnitude over the surface of the cell. Cut the cell into smaller volumes to improve the performance of the mesh generator.

Specific	The following error message:
Problems	Unable to allocate memory required

means that a mesh with a large number of elements was created and no more memory was available for storage. The mesh size controls applied to the model are probably not realistic. The maximum number of elements that can be created with a 2GByte addressing limit is approximately 8 million.

The **FILTER** Command

Summary	Sets a filter for graphical selection.

Icons



Menu Route

Picking

Command Line Parameters

Command	FILTER		
Parameter	Default	Function	
TYPE	NONE	NONE	No graphical selection
		BODY	Sets the type of topological
		CELL	entity that is selected when
		FACE	double clicking over it in the
		EDGE	display.
		VERTEX	
		LCS	
		CONDUCTOR	
COMMAND	PICK	PICK	Determines the effect of select-
		HIDE	ing by cursor
		LIST	
DISPLAY	VISIBLE	VISIBLE	Any visible object can be
			selected
		EXPLICIT	Only objects explicitly selected
			for display can be selected

Notes

The filter command controls the graphical interaction with the Modeller. Setting the **TYPE** parameter controls the type of entity that can be selected using the cursor.

The **COMMAND** parameter controls the effect of double-clicking when an object is highlit. The **DISPLAY** parameter can be used for extra control of selection of items.

COMMAND=PICK allows objects to be added to the list of picked entities by issuing a **PICK** command.

COMMAND=HIDE hides the object from the display by issuing a **SELECT OPTION=HIDE** command.

COMMAND=LIST generates a list showing details of the object and attached data.

This command does not generate an entry in the history stream, and hence produces no UNDO or REDO command state.

The **FITTEDCPE** Command

Summary Create or modify fitted constant perimeter end (CPE) conductors.

icon

0

Command Line	Command	FITTED	CPE	
Parameters	Parameter	Default	Function	
	OPTION	NEW	NEW	Creates a new fitted CPE con-
				ductor
			MODIFY	Modifies properties of the picked fitted CPE conductors
			LOAD	Loads defaults from picked conductors
	DRIVELABEL		Name for	the drive label
	LCNAME		Name for coordinate	Local Coordinate System for e system 1
	SYMMETRY		Rotationa	l symmetry about global Z axis
	XCEN2		Origin of	coordinate system 2
	YCEN2			
	ZCEN2			
	THETA2		Euler ang	les defining orientation of coor-
	PHI2		dinate sys	tem 2
	PSI2			
	RXY		Reflection symmetries in XY, YZ and planes	
	RYZ			
	RZX			
	ALPHA		Azimutha	l angular position of the straight
	BETA		Cutter ang	gle
	Α		Cross-sec	tional width
	В		Cross-sec	tional height

Command	FITTED	CPE (continued)		
Parameter	Default	Function		
H1		Half lengt	h of the straight	
R1		Radius of	forming cylinder	
R2		Radius of	cross-over arc	
CURD		Current density in the conductor. This can		
		be defined	in terms of the current as cur-	
		rent/AR	EA	
TOLERANCE		Field calculation tolerance		
INCIRCUIT		Is the conductor part of an external cir-		
		cuit:		
		NO	The conductor has defined cur-	
			rent density.	
		YES	The current in the conductor is	
			determined by an external cir-	
			cuit.	
REVERSE		Reverse the connections to this conductor		
		in its circu	ut: YES or NO.	
CIRCUITELEMENT		The name of circuit element this conduc-		
		tor is part of.		
KEEP	NO	NO	Clear the list of picked items	
		YES	Keep the list of picked items	
			for further modification	

Notes

This command creates a new fitted CPE conductor when using OPTION=NEW.

OPTION=MODIFY it is used to operate on the list of picked fitted CPE conductors, changing geometry, coordinate systems, symmetries, current density, tolerance etc. The current common values of the picked fitted CPE conductors are available by calling **CONDUCTOR OPTION=LOAD**.

The default values of the command parameters are updated to match common values shared by all of the picked fitted CPE conductors. If the picked fitted CPE conductors do not have a common value for a parameter, that parameter is left unset. Using OPTION=MODIFY will change the conductor data of all of the picked fitted CPE conductors to the new values given in the parameters, but will not affect the conductor data of any unset parameter.

For more information on the parameters, see "The CONDUCTOR Subcommand MODIFY" on page 4-44. For more information on connecting conductors to external circuits, see "The CIRCUIT Command" on page 3-51.

The **GUIOPTIONS** Command

Summary Set console visibility and other window option

Menu Route

Window ↓ Preferences

Command Line Parameters

Command	GUIOPTIONS			
Parameter	Function			
OPTION	Action of command			
	LOAD Used by the GUI.			
	SET Update the values.			
CONSOLEVIEW	Console visible: YES or NO			
CONSOLEDOCK	Console docked: YES or NO			
CONSOLEBUFFER	Number of lines saved in the console.			
PRINT3DCARD	Print from graphics card: YES or NO			
WINDOWPOSSAVE	Save window position and size on exit: YES or NO			

The GUIOPTIONS command can be used to change the options which affect the appearance and behaviour of the GUI window. Default settings are loaded from and changes are saved in the registry (on UNIX systems, file ~/.vectorfields-options). Saving the window size and position is optional - see below.

- Console the command input and text output console can be visible or hidden (CONSOLEVIEW) and when it is visible it can be docked (at the bottom of the main window) or undocked (a separate window) (CONSOLEDOCK). The scrolling buffer can be limited to a number of lines (CONSOLEBUFFER).
- Printing the option to print the image stored in the graphics card is preferable with most graphics card (PRINT3DCARD) but can be changed if necessary. The alternative copies the image from the screen display.
- Window size and position the current size and position can be saved when the program exits (WINDOWPOSSAVE). To preserve the size and position from subsequent changes, WINDOWPOSSAVE must be set to NO, the next time the program is started.

The **HELICALEND** Command

Summary	Create or modify helical end conductors.
Icon	\diamond

Menu Route	Create \downarrow
	Conductor $ ightarrow$ Helical end
	Operations \downarrow
	Modify conductors $ ightarrow$ Helical end

Command Line Parameters

Command	HELICA	LICALEND		
Parameter	Default	Function		
OPTION	NEW	NEW	Creates a new helical end con-	
			ductor	
		MODIFY	Modifies properties of the	
			picked helical end conductors	
		LOAD	Loads defaults from picked conductors	
DRIVELABEL		Name for	the drive label	
LCNAME		Name for	Local Coordinate System for	
		coordinate	e system 1	
SYMMETRY		Rotational	symmetry about global Z axis	
XCEN2		Origin of	coordinate system 2	
YCEN2				
ZCEN2				
THETA2		Euler angl	es defining orientation of coor-	
PHI2		dinate system 2		
PSI2				
RXY		Reflection symmetries in XY, YZ and ZX		
RYZ		planes		
RZX				
ALPHA		Azimutha	angular position of the straight	
BETA		Cutter ang	le	
А		Cross-sect	ional width	

Command	HELICA	LEND (continued)				
Parameter	Default	Function				
В		Cross-sectional height				
H1		Half length of the straight				
H2		Length of	conductor			
R1		Radius of	forming cylinder			
R2		Radius of	cross-over arc			
CURD		Current de	ensity in the conductor. This can			
		be defined	l in terms of the current as			
		current	AREA			
TOLERANCE		Field calculation tolerance				
INCIRCUIT		Is the conductor part of an external cir-				
		cuit:				
		NO The conductor has defined				
			current density.			
		YES	The current in the conductor is			
			determined by an external cir-			
			cuit.			
REVERSE		Reverse th	e connections to this conductor			
		in its circu	iit: YES or NO.			
CIRCUITELEMENT		The name of circuit element this conduc-				
		tor is part of.				
KEEP	NO	NO	Clear the list of picked items			
		YES	Keep the list of picked items			
		for further modification				

Notes

This command creates a new helical end conductor when using OPTION=NEW.

OPTION=MODIFY it is used to operate on the list of picked helical end conductors, changing geometry, coordinate systems, symmetries, current density, tolerance etc. The current common values of the picked helical end conductors are available by calling **CONDUCTOR OPTION=LOAD**.

The default values of the command parameters are updated to match common values shared by all of the picked helical end conductors. If the picked helical end conductors do not have a common value for a parameter, that parameter is left unset. Using OPTION=MODIFY will change the conductor data of all of the picked helical end conductors to the new values given in the parameters, but will not affect the conductor data of any unset parameter. For more information on the parameters, see "Helical Ends" on page 4-32. For more information on connecting conductors to external circuits, see "The CIRCUIT Command" on page 3-51.

The **HIDE** Command

Summary	Hides picked entities.
---------	------------------------

Command Line Parameters		
	Command	HIDE
	No parameters	

Notes The HIDE command hides all picked objects from display, and unpicks them. Objects can be temporarily hidden (without removing them from the list of picked objects) by using **SELECT OPTION=PICKEDREMOVE**.

The **HISTORY** Command

Summary	, C	ontrols	the	size	of the	history	stream.
Summe		01101010		OIL C	01 1110	motory	Sucum

History

Edit \downarrow

Menu Route

Command Line Parameters

Command	HISTOF	RY
Parameter	Default	Function
STATES	0	Number of history states to keep in the history stream

Notes

The **HISTORY** command allows the size of the history stream to be limited. This may help reduce the memory usage of the modeller. The **STATES** parameter determines how many states are kept. If zero, there is no limit. If greater than zero, only this number of operations will be retained, and hence the **UNDO** command can only be called a limited number of times.

The **IMPORT** Command

Summary	Imports conductor data from a conductor file generated by the Modeller, pre or post processor.
Menu Route	$ extsf{Create} \downarrow \\ extsf{Conductor} ightarrow extsf{Import}$

Command Line
Parameters

Command	IMPOR [®]	Т
Parameter	Default	Function
FILE		Conductor file to be imported

Notes

The IMPORT command reads a file containing conductor data. The conductor file must be generated by the WRITE subcommand of the pre processor CONDUCTOR command (page 4-48), the ACTION=EXPORT option in the CONDUCTOR command of the post processor (page 5-40), or by the EXPORT command in the Modeller (page 3-73).

The data in the file may contain more than one conductor.

The conductor data in the file contains the Local Coordinate System values, not the names of Local Coordinate Systems. If an existing Local Coordinate System is found that matches the coordinate system values of a conductor, this will be used, otherwise a new Local Coordinate System will be created and set as coordinate system 1 for the conductor.

For hints on how to write compatible command scripts for importing conductors into the Modeller, pre and post processors see "Conditional commands" on page 2-20.

The INSERTOP2FILE Command

Summary	Import an OPERA-2d data file.

Command Line Parameters

Command	INSER	RTOP2FILE		
Parameter	De- fault	Function		
OPTION	LOAD	D LOAD Import the file specified FILE parameter		
		ANALYSE	Open the data file and deter- mine if any region groups have been defined	
		CLEAR	Initialise the dialog	
FILE		Name of the	e data file to be imported	
IGNOREAIR	YES	YES	Regions with the property of air will not be imported	
		NO	Air regions in the data file will be imported	
GROUP		Name of a particular region group to be imported from the file		
USEGROUP	NO	NO	Do not look for a particular group	
		YES	Import regions with a particu- lar group name specified by GROUP	
GROUPLABEL	YES	YES NO	Add a label to the face if it belongs to a group. The label will be based on the group name	
MATERIALLABEL	YES	YES NO	Add a material label to the cells or sheet face bodies cre- ated when the file is imported	

Command	INSER	TOP2FILE (continued)				
Parameter	De- fault	Function				
REGIONLABEL	NO	NO YES	Add a region label to the bod ies created when the file is imported			
SWEEPDISTANCE	1	Defines the extrusion distance or the angle of rotation for axi-symmetric models				

Notes

This command allows OPERA-2d geometries to be imported into the Modeller.

The imported 2D geometry is represented in the 3D Modeller as a set of bodies. If SWEEPDISTANCE is set to zero the OPERA-2d regions will be imported as a set of sheet face bodies. If SWEEPDISTANCE is non zero the imported regions are extruded by the distance, or for axi-symmetric models rotated by the angle, specified by the value of the parameter.

All bodies created from the import geometry are assigned a label based on the name of the OPERA-2d data file.

Additional labels can be added to the imported geometry using the parameters, GROUPLABEL and REGIONLABEL. These labels are always attached to the bodies created from the imported file. The parameter MATERIALLABEL adds a material label to the cells created during the extrusion process. If no extrusion is performed, SWEEPDISTANCE=0, then the material label is added to the sheet face bodies.

The LABEL Command

Summarv	Controls setting	additional	labels on	entities.
~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~				

 Menu Route
 Properties ↓

 Additional labels

Command Line Parameters

Command	LABEL		
Parameter	Default	Function	
OPTION	ADD	ADD	Adds a label to picked items
		REMOVE	Removes the named label from the picked items
		CLEAR	Removes all labels from the picked items
LABEL		Label to be	added or removed

Notes

The LABEL command can be used to add labels to any entity within the model. This label can then be used as the basis for grouping items for selection for visual display, or for picking for modification.

Using OPTION=ADD will add the named label to all picked items. OPTION=REMOVE will remove the named label from all items that are picked, if they have the label.

OPTION=CLEAR will remove all labels from the picked items.

The label defined on a face can also be used as the name in the labelled face scala emitters.

During Boolean operations, the labels attached use the data storage level of any other data attached to the entity when deciding on what labels are kept. If the data is at the same level, the labels from both entities are merged.

The LCS Command

Summary	Creates a new named Local Coordinate System (LCS), or changes the loca-
	tion of an existing Local Coordinate System of this name.

Menu Route	Create \downarrow		
	Local	coordinate	system

Command Line Parameters

Command	LCS	
Parameter	Default	Function
LCNAME		Name of Local Coordinate System to be created or modified
X0		Origin of the Local Coordinate System
Y0		
Z0		
THETA		Euler angles defining the local orientation of the
PHI		Local Coordinate System
PSI		

Notes

All values must be given and are specified in the Working Coordinate System. If a Local Coordinate System of the given name already exists, its position and orientation are reset to the new values.

Each Local Coordinate System is uniquely identified by its LCNAME. A Local Coordinate System can be set to be the current working coordinate system using the WCS command. A Local Coordinate System is also used within conductors to define the Local Coordinate System 1 of the conductor.

A Local Coordinate System can be picked from the display by setting the FILTER command to TYPE=LCS. If a single Local Coordinate System has been picked it can be set to be the Working Coordinate System and can also be renamed. If one or more Local Coordinate System is picked they can be transformed, copied or deleted. Copying a Local Coordinate System will generate a new name from the original to keep the name unique.

The LIST Command

Summary	Shows	informa	tion abo	ut entities	within	the r	nodel.
Summuny	0110110	monna	non aco	at entitles	** 101111		1100001.

Menu Route	Properties \downarrow
	List properties

Command Line
Parameters

Command	LIST		
Parameter	Default	Function	
TYPE	none	BODY	Type of entity whose data is to be
		CELL	shown
		FACE	
		EDGE	
		VERTEX	
		LCS	
		CONDUCTOR	
NUMBER		Entity number	

Notes

The list command shows information about the entity specified by TYPE and NUMBER parameter. If no entity corresponds to this information, data on all picked entities is given.

This command can be activated graphically by using **FILTER COM-MAND=LIST**. Subsequent selections using the cursor will show information on that entity, rather than picking it.

The LOAD Command

a model file.
L

Icon



Menu Route	File \downarrow
	Open model data
	Create \downarrow
	Insert from file

Command Line Parameters

Command	LOAD		
Parameter	Default	Function	
OPTION	none	NEW	Clears all existing model data and loads a new model file
		INSERT	Inserts new data from the file into the existing model
FILE		File to be	opened

Notes

I

If opening a new file, **OPTION=NEW**, the existing model data is cleared. At this point the user will be asked to confirm the operation if data has been changed.

LOAD OPTION=NEW sets the currently open file. This filename will be changed by the **SAVE OPTION=NEW** command and cleared by the **CLEAR REVERT=NO** commands.

OPTION=INSERT is used to append additional parts to the model, by loading them from file, into the working coordinate system. If the model currently has no bodies or conductors, all bodies in the file will be loaded, including a model body and its mesh. If bodies or conductors do exist however, a model body in the file being inserted will not be loaded. All component bodies, conductors and LCS will be added.

With OPTION=INSERT, only undefined data attached to any of these items is loaded. For example, if the current model has a material label
Iron defined, the material properties associated with **Iron** will not be updated from those in the data file. A filename must be given.

The LOAD command will operate on standard .opc files and binary .opcb files, and will also work on ACIS .sat files from the Modeller or other packages. When loading .sat files from other software packages, it should be noted that the model may not be well suited to finite element mesh generation and analysis and may cause problems.

The binary form of the data file can be loaded by explicitly giving the *.opcb* extension. The loading of these files may be slightly faster, but the files may not be portable between different platforms. The *.opcb* files should not be renamed to a different file type, since the file type is used to determine the format.

IGES files (*.igs*) will be recognised from the file extension and loaded. Files in the standard IGES format may not contain a valid solid body geometry. The data within the file will be loaded and manipulated to try to generate a valid solid body geometry, but the data should be checked to ensure that it has no errors.

The LOFT Command

<i>Summary</i> Fills the volume between two fac

Menu Route Operations ↓ Loft

Command Line

Parameters

Command	LOFT				
Parameter	Default	Function	Function		
FACE1TANGENT	1.0	Tangent continuity parameter of face 1, the first face selected			
FACE2TANGENT	1.0	Tangent continuity parameter of face 2, the second face selected			
KEEP	NO	YES	The original lofting faces are kept in the final model. A new cell will be created between the two faces		
		NO	The original lofting faces are removed from the model.		

Notes

The LOFT command can be used to fill the volume between two faces.

It is necessary to select the two lofting faces before the command can be executed.

The way in which the space between the two lofting faces is filled is governed by the face tangent parameters, FACE1TANGENT and FACE2TANGENT, the KEEP parameter and any model edges that already connect the two faces.

The tangent continuity parameters control the smoothness of transition between the faces connected to the lofting face and the new faces created to bound the volume between the two faces. FACE1TANGENT sets the smoothness for the face selected first and FACE2TANGENT for the face selected second. A value of 0 means that the lofting faces will be joined by straight line edges and planar faces. Values greater than 0 results in a smoothing effect and faces with an underlying spline geometry will be created.

If KEEP=YES is specified then the lofting faces are retained and the volume between the two faces is filled with a new cell. The properties of this new cell will be the same as one of the cells adjacent to the initial lofting faces. If KEEP=NO is defined the initial lofting faces are removed. Any cells adjacent to the lofting faces are removed to be replaced by one new cell. The new cell will be defined with the properties of the cell with the greater data storage level.

If any edges connect the two lofting faces these edges will be used as guides to help create the new geometry.

The MATERIALS Command

sel.
)

Menu Route	Model \downarrow	
	Set material pr	operties

Command Line

Parameters

Command	MATEF	TERIALS		
Parameter	Default	Function		
OPTION		PICK	Adds a material label to a list to be set	
		UNPICK	Clears the list of material labels to be set	
		RESET	Sets picked materials to have the properties of air	
		MODIFY	Sets the data for the picked materials	
		METRE	Work in SI units	
		CGS	Work in CGS units	
		INCH	Work in SI units with inches	
		MM	Work in SI units with MM	
		MICRON	Work in SI units with microns	
		LIST	Lists the material proper- ties of the picked materi- als	
		DELETE	Deletes the picked mate- rials	
MATERIALLABEL		Material label use	ed with OPTION=PICK	
ANISOTROPY		Sets the anisotrop mittivity and con-	by of all permeability, per- ductivity	
		ISOTROPIC	Set to isotropic	
		PACKED	Set to packed	
		MULTIPLE	Set to anisotropic	

OPERA-3d Reference Manual

Command	MATERIALS (continued)			
Parameter	Default	t Function		
LINEARITY		Sets the properties to be linear or non-linear		
		LINEAR		Set to linear
		NONLINEAR	2	Set to non-linear
MUANISOTROPY		Sets the anisot	trop	y of permeability
		ISOTROPIC		Set to isotropic
		PACKED		Set to packed
		MULTIPLE		Set to anisotropic
MULINEARITY		LINEAR		Set to linear
		NONLINEAR	2	Set to non-linear
MU		Isotropic linea	ır pe	ermeability
HC		Isotropic coerd	civi	ty
BH		Isotropic non-	line	ar BH curve label
MPHASE		Phase lag for i	isot	ropic permeability
MUXX		Anisotropic co	omp	onents of linear permea-
		bility		
MUYY				
MUZZ				
HCX		Anisotropic components of linear coercivity		
HCY				
HCZ				
BHX		BH data labels defining the anisotropic components of non-linear permeability.		
BHY				
BHZ				
MAPHASE		Complex phase lag for anisotropic permea- bility		
SIGANISOTROPY		Sets the anisot	trop	y of conductivity
		ISOTROPIC		Set to isotropic
		MULTIPLE		Set to anisotropic
SIGMA		Isotropic cond	luct	ivity
SPHASE	1	Phase lag for i	isot	ropic conductivity
SIGXX		Anisotropic components of conductivity		
SIGYY		1	_	
SIGZZ		1		
SAPHASE		Complex phase lag for anisotropic permit- tivity		
EPSANISOTROPY	EPSANISOTROPY Sets the anisotropy of permittivity		y of permittivity	
		ISOTROPIC	•	Set to isotropic
		MULTIPLE		Set to anisotropic

Command	MATERIALS (continued)		
Parameter	Default	Default Function	
EPSILON		Isotropic permitti	vity
EPHASE		Phase lag for isot	ropic permittivity
EPSXX		Anisotropic comp	oonents of permittivity
EPSYY			
EPSZZ			
EAPHASE		Complex phase lag for anisotropic permit-	
		tivity	
KAPANISOTROPY		Sets the anisotropy of thermal conductivity	
		ISOTROPIC	Set to isotropic
		MULTIPLE	Set to anisotropic
KAPPA		Isotropic thermal	conductivity
ΚΑΡΧΧ		Anisotropic comp	ponents of thermal conduc-
		tivity	
ΚΑΡΥΥ			
KAPZZ			
HCAP		Unused at present	
MDEN		Unused at present	

Notes

This command defines the material characteristics for use by the analysis programs.

A set of material labels is picked using the command repeatedly, with OPTION=PICK and a MATERIALLABEL specified. A material label can be removed from the set using OPTION=UNPICK. If no MATERIALLABEL is given, the set is emptied.

Issuing the command with OPTION=MODIFY will modify the properties of the picked material labels to the new values given in the parameters. The value of properties associated with the material labels are unchanged if the parameter is unset.

The default value of each of the parameters is updated to be the common value of all the picked material labels. If the data of one of these parameters is unset, or the picked material labels do not share the same value, then the parameter value is left clear.

New material labels are assumed to be linear and isotropic. OPTION=RESET will clear the properties associated with all of the picked material labels.

The working material unit set can be changed using OPTION=METRE, OPTION=CGS or one of the other sets. If the units are changed, the values

in the parameters are converted to the new unit set, and values entered will be interpreted in this unit set (note however that in **CGS** units the coercive force \mathbf{H}_{c} is in units Oersteds and for all other options it is in units A/m).

The properties of all material labels can be listed using OPTION=LIST.

Material labels that are not used, i.e. have no cell referencing them can be deleted using **OPTION=DELETE**. Deleting a material label that is in use will reset its properties to those of air.

All parameters can be specified. Which of these values will used by the analysis will depend upon the analysis module being used, e.g. the phase lag properties are not used by TOSCA.

The definition of the thermal conductivity, KAPPA or KAPXX, KAPYY and KAPZZ, may be defined as functional properties of position (X, Y, Z) or as a non-linear function of temperature, T.

The **MESH** Command

Summary	Generates the surface mesh for the model body
Menu Route	Model \downarrow Generate surface mesh

Command Line Parameters

Command	MESH	
Parameter	Default	Function
SIZE	1	Maximum element size in the mesh
NORMALTOL	30	Maximum angle between mesh facets normals (degrees)
SURFACETOL	0.0	Absolute tolerance to which the mesh must fit a curved surface
TOLERANCE	1.0e-6	Absolute tolerance used to check point coinci- dence

Notes

The mesh command can only be called once the model body has been created using the MODEL command.

The operation creates the surface mesh from the information provided. If cells, faces, edges or vertices have their own mesh data attached, the smaller value is used.

The mesh element size on a face will be determined from the distribution of elements along the edges. The mesh element size along an edge is determined from the smallest values from:

- the information from the MESH command parameters;
- mesh information on any cells that contain the edge;
- mesh information on any faces that are bounded by the edge;
- mesh information on the edge itself;
- mesh size information on the vertices at the ends of the edge.

The faces of all cells are meshed sequentially. On execution of the MESH command, the outline frame of the face being processed is highlighted. A progress bar is also displayed at the bottom of the Modeller.

The MESH command may take some time for complex models. On completion the status bar at the bottom of the window displays the message "Surface Mesh".

Surface Mesh size control

The initial surface mesh is now formed from elements with edge lengths of the order of the specified size. This stricter interpretation of the size control produces more surface elements than previous releases (version 8.7 and earlier produced elements with area approximately equal to 0.5*size²). The user variable (#MESHSTYLE) can be set to select the earlier interpretation of the size control.

\$CONSTANT #MESHSTYLE 8.7

Surface mesh Simple topology tests are used to check the initial surface mesh created on the model. The program may report a number of different types of error, **Errors** these are explained in the following: Surface mesh failure: CHECK the bodies for errors This means that errors have been found in the surface mesh that are probably caused by errors in the model geometry. The CHECK command should be used to find these errors and possibly to repair them. Repairing is not always possible, in some cases it may be necessary to find a better way of constructing the model - for example avoiding glancing contacts between two curved surfaces. ACIS surface mesh contains Faces with no mesh - CHECK body for errors This again means that errors have been found in the surface mesh that are probably caused by errors in the model geometry. The CHECK command should be used to find these errors and possibly to repair them as above. This error may also arise if a vanishingly small face has been created by small errors in the dimensions of the model. ACIS surface mesh contains holes - CHECK body for errors or ACIS surface mesh is degenerate - change mesh size This means that holes have been found in the surface mesh that may be caused by errors in the model geometry or by the surface meshing of the geometry. The CHECK command should be used first to see if there are errors in the geometry, and these should be repaired. If no errors are found, a more rigorous surface meshing procedure can be used that tests for loss of internal edges and points. The string variable ACISMESH-CHECK can be used to control the level of testing applied to the surface meshing process, the default is:

\$STRING ACISMESHCHECK NONE

The following options may be used

String Variable	ACISMESHCHECK
Option	Description
NONE	No additional tests performed
PLANARVERTEX	Check for duplicate vertices only on planar faces
PLANAREDGE	Check for duplicate edges only on planar faces
PLANARALL	Check for duplicate edges and vertices on planar faces
ANYVERTEX	Check for duplicate vertices on all faces
ANYEDGE	Check for duplicate edges on all faces
ANYALL	Check for duplicate vertices and edges on all faces

This checking can slow the surface meshing significantly, so should only be used where problems occur. PLANARVERTEX is the fastest option, and ANYALL is the slowest.

• Unable to allocate memory required

This means that a gigantic surface mesh was created and no more memory was available for storage. The mesh size controls applied to the model are probably not realistic.

The **MODEL** Command

Summary Creates a single body model suitable for meshing and analysis.

Menu Route	Model \downarrow
	Create model body
	Model \downarrow
	Delete model body
	Properties \downarrow
	Set model components

Command Line Parameters

Notes

Command	MODEL		
Parameter	Default	Function	
OPTION	CREATE	CREATE	Creates a model body for meshing and analysis
		DELETE	Deletes the model body
		DEFAULT	Sats the status of picked hodies for
		INCLUDE	inclusion during create model body
		EXCLUDE	inclusion during create model body.
		LOAD	Loads the status of the picked bodies

The MODEL OPTION=CREATE command is used to generate a single body model suitable for meshing. It must be used before a surface or volume mesh can be generated.

The status of picked component bodies can be set by using:

- OPTION=DEFAULT so that bodies will be included in the model body unless they are formed from wire edges only.
- **OPTION=INCLUDE** so that bodies will always be included
- **OPTION=EXCLUDE** so that bodies will always be excluded.

During the operation, if no bodies are picked, the union without regularisation of all bodies, whose status indicates that they are to be included, is created to form the model body. If bodies are picked, their union is formed without regularisation to form a single *model body* (and the status of the bodies is ignored) If any of the bodies have the name **BACKGROUND**, the *model body* is trimmed to the boundary of the union of these bodies, before being unioned with them. This is useful when only a section of the model is needed, but when the geometry is easier to build completely, e.g. for axi-symmetric models where the construction is best done using cylinders.

Any cell within the model body, that has material label name NULL is deleted from the model body.

After forming the model body, any layering data applied using the FACE-DATA command, is used to generate thin regions from these faces.

Once created, the model geometry cannot be changed. Properties associated with parts of the model can be adjusted, but these changes are not applied to the components used to form the model, and hence changes will be lost when reverting to component view.

When the model body has been created, this is the only body that can be seen. Component bodies used to form the model body still exist, but may not be manipulated or viewed until the model body has been deleted using MODEL OPTION=DELETE or with the DELETE command.

The model body can be saved, and the component bodies will be saved when using SAVE OPTION=ALL or OPTION=NEW. When loading, the view will be of the model body, rather than the components. If loading with OPTION=INSERT, the model body is not loaded.

Conductors and Local Coordinate Systems are unaffected by the MODEL command and can still be created and modified.

The reason for the model command is that when meshing, it is necessary to have a single body, so that all volumes are correctly bounded, and the mesh is continuous throughout the model. However, a single body model makes it difficult to adjust the dimensions and positioning of sections of the model, which is simple when the model is made up of several independent bodies.

The **MORPH** Command

Summary	Morphs the geometry of bodies by applying a functional transformation.
Menu Route	Operations \downarrow Morphing $ ightarrow$ General morph

Command Line Parameters

Command	MORPH	
Parameter	Default	Function
UMAPPING	U	New value for the U coordinates
VMAPPING	V	New value for the V coordinates
WMAPPING	V	New value for the W coordinates
KEEP	NO	Keep the picked items for further operations: YES or NO.

Notes

The MORPH command allows bodies to be transformed to a new shape. The functions specified for UMAPPING, VMAPPING and WMAPPING must be functions of the bodies original spacial location U,V and W.

For example the command

Morph umapping=u vmapping=v+sin(w) wmapping=w

would transform the body by effectively corrugating it.

Any function can be specified, although a function whose input is invalid for certain values (e.g. **SQRT** is not valid for any negative number) may not be accepted.

The morph command will not change the underlying topology of a body (i.e. it will still have the same number and connectivity between faces, edges etc.). Morphing can cause e.g. self-intersection which will result in bodies with topological errors. These are very likely to cause problems in later operations.

Note that any morph that does not make use of all 3 values of U,V and W will be transformed into a plane (only 2 values used) or an edge (only one value given). The body formed from these operations will not be valid as the topology will not be updated to match the new dimensionality.

The **MOUSE** Command

Summary	Swap	the	function	ality	of middle	and	right	mouse	buttons.
Summury	Dinup	une	runetion	unity	or innuare	unu	ingin	mouse	outtons.

Icon



Command Line				
Parameters	Command	MOUS	E	
	Parameter	Default	Funct	tion
	NORMAL	YES	Set st	ate of right mouse button:
			YES	Normal behaviour.
			NO	Emulates the middle button.

Notes The MOUSE command with **normal=no** swaps the functionality of the middle and right mouse buttons so that the software can be used with a 2-button mouse.

The default (**normal=yes**) mouse functions in the 3D Graphics Window are:

- left: rotate
- middle: zoom
- right: translate

The **PERIODICITY** Command

Summary	Defines the set of transformations for creating periodic symmetry bound-
	ary pairings within the model.

Menu Route	$\texttt{Model} \downarrow$
	a

Symmetry conditions...

Command Line Parameters

Command	PERIODICITY			
Parameter	Default	Function		
NSETS	0	Defines the number of periodicity transforms defined.		
DX1	0	Defines the x,	y, z displacement of the first	
DY1	0	periodicity trai	nsform.	
DZ1	0			
ROTX1	0	Defines an axi	s, based on the Global Coordi-	
ROTY1	0	nate System, a	bout which a rotational com-	
ROTZ1	0	ponent of the f	irst periodicity transform can	
ANGLE1	0	The angle of ro	otation about the axis.	
TYPE1	POSITIVE	POSITIVE	Set the potentials on paired faces to be the same.	
		NEGATIVE	Set the potentials on paired faces to have the opposite sign.	
OPTION1	MATCH	MATCH	Force the geometry of the model to match the specified periodicity transform.	
		EXTERNAL	Consider the faces on the external surface of the model when pairing faces.	
		SYMMETRY	Consider faces which have been assigned the symmetry boundary condition.	
DX2	0	Defines the x, y	y, z displacement of the second	
DY2	0	periodicity trai	nsform.	
DZ2	0	1		

Command	PERIODICIT	Y (continued)
Parameter	Default	Function
ROTX2	0	Defines an axis, based on the Global Coordi-
ROTY2	0	nate System, about which a rotational com-
ROTZ2	0	ponent of the second periodicity transform can be defined
ANGLE2	0	The angle of rotation about the axis.
TYPE2	POSITIVE	See TYPE1
OPTION2	MATCH	See OPTION1
DX3	0	Defines the x, y, z displacement of the third
DY3	0	periodicity transform.
DZ3	0	
ROTX3	0	Defines an axis, based on the Global Coordi-
ROTY3	0	nate System, about which a rotational com-
ROTZ3	0	ponent of the third periodicity transform can be defined
ANGLE3	0	The angle of rotation about the axis.
TYPE3	POSITIVE	See TYPE1
OPTION3	MATCH	See OPTION1

Notes

The **PERIODICITY** command is used to define periodic or symmetry boundaries. Periodic boundaries are recognised by the TOSCA, SCALA and TEMPO analysis packages.

Periodic boundaries link the potential values, and hence the fields, over one surface to the potential values over another surface within the model. More precisely, the potential values on one surface are set equal to the values on the other surface (with or without a change of sign).

The **PERIODICITY** command is used to define which faces are linked or paired within the model. This pairing is achieved by specifying a periodicity transform which maps one face, defined as the master face, onto its paired face, the slave face. Edges and vertices are also paired by the periodicity transform. A maximum of 3 separate periodicity transforms can be specified. The actual number of sets used is controlled by the **NSETS** parameter.

Each transform is defined by a displacement and a rotation about the origin. For the first periodicity transform the displacement is defined by the parameters DX1, DY1 and DZ1, which correspond to the x, y and z components of the displacement. The rotational component of the transform is specified by the vector ROTX1, ROTY1, ROTZ1 and the parameter ANGLE1. The vector defines the axis of rotation and the parameter ANGLE1 determines the angle of rotation about this axis.

The parameter **TYPE1** specifies whether the potentials over paired faces have the same (**TYPE1=POSITIVE**) or opposite (**TYPE1=NEGATIVE**) sign. The parameter **OPTION1** defines the method by which vertices, edges and faces are paired.

- To crop the geometry of the model to match the periodicity conditions defined, OPTION1=MATCH can be specified.
- To pair only the external faces, OPTION1=EXTERNAL, can be specified. When an exact match is not possible between faces, faces are subdivided to ensure that the paired faces have the same geometry
- To pair faces with the SYMMETRY boundary condition applied OPTION1=SYMMETRY can be used. This option can be selected when only certain faces on the external surface of the model have to be paired. Both faces to be paired need to have the SYMMETRY boundary condition applied.

The second and third periodicity transforms can be defined in a similar way.

The pairing of faces is performed when the model body is created. Once the model body has been formed, paired vertices, edges and faces are assigned labels which can be displayed. A label is also given to faces with positive symmetry and to faces with negative symmetry.

When the surface mesh is generated, the mesh on paired faces will be identical. A consequence of this is that any surface mesh refinements specified on one vertex, edge or face of a pair will also be applied to the mesh refinement on the second vertex, edge or face of the pair.

Warnings are given if:

- A defined periodicity transform does not result in the pairing of any faces within the model.
- A face with the **SYMMETRY** boundary condition applied is located within the interior of the model. Periodic boundaries can only be defined on the external surfaces of the model.
- A face with the **SYMMETRY** boundary condition is not paired by the periodicity transforms specified.

The **PICK** Command

Summary Picks items for modification.

Icons



Menu Route

Picking

Command Line Parameters

Command	PICK				
Parameter	Default	Function			
OPTION	ADD	RESET	Clears all picked items		
		ADD	Adds items into the list of		
			picked objects		
		REMOVE	Removes items from the list		
		TOGGLE	Adds if the items are not picked, otherwise removes them		
		FILTER	Removes any picked entities		
			that are not of the current filter		
			type		
		CHANGE	Replaces picked entities with		
			those of the current filter type		
		ALL	Picks all entities of the current		
			filter type		
PROPERTY		Property of items including MATERIAL,			
		Let have the the the second se			
		Label associated	with the specified property type		
ТҮРЕ	none	BODY	Type of entity whose data is to		
		CELL	be shown		
		FACE			
		EDGE			
		VERTEX			
		LCS			
		CONDUCTOR			
NUMBER		Identifier for the	item of the entity type		

Notes

Command	PICK (continued)			
Parameter	Default	Function		
PTU		Coordinate in or near to an entity to be picked		
PTV				
PTW				

The **PICK** command allows objects to be picked for modification. Objects picked are stored in an ordered list and can then be used in many operations.

The list can be cleared using **OPTION=RESET**.

OPTION=ADD adds new entities to the list.

OPTION=REMOVE removes entities from the list

OPTION=TOGGLE adds an entity if it is not in the list, or removes it if it is in the list.

With OPTION=ADD, OPTION=REMOVE and OPTION=TOGGLE there are 3 ways of specifying items:

The first uses an entity TYPE with a NUMBER. This method is used during graphical selection of objects. The entity type can be BODY, CELL, FACE, EDGE, VERTEX, LCS or CONDUCTOR. Each object has a unique integer identifier and this is specified in the NUMBER parameter. If number is zero, all objects of the given type are picked.

This method should not be used in scripts if possible. After commands such as **COMBINE** or **DELETE**, the numbering of the items will be changed. The method in which the numbering is applied cannot be guaranteed to be the same on subsequent runs, so use of this method may mean that the wrong items are picked if rerunning a script.

- If no number is specified an entity of the given type can be picked by providing a coordinate (PTU, PTV, PTW), specified in the Working Coordinate System. This method will pick a body or cell that contains the point. If the type is face, edge or vertex, the nearest entity will be found. Local Coordinate Systems and conductors cannot be picked using this method. If the pick is ambiguous, no item is picked.
- The third method uses the data attached to each of the entities. The **PROPERTY** parameter can be used to select a type of data item that is stored. For example **PROPERTY=MATERIAL** will pick objects that have a material label attached. This will be all cells, as they all have this information.

By specifying a label the **PICK** command can be more refined, e.g. **PROPERTY=MATERIAL LABEL=AIR**, would pick only cells with a material label of air.

If more than one item is picked in a single **PICK** command, the order in which items are added to the list is undefined. In most cases this is unimportant. For **COMBINE OPERATION=SUBTRACT**, **OPERATION=TRIM** or **OPERATION=CUTAWAY**, the order is critical to the result and more care must be taken in picking the objects.

Subsequent PICK commands always add picked items to the end of the list of picked items.

Picked faces can be displayed with direction arrows showing the normal direction of the face. This normal direction is used by various commands, e.g. **SWEEP OPTION=DISTANCE**. The direction of the normal can be reversed by double clicking over one of the direction vectors (with **FIL-TER TYPE=FACE**). This has the effect of generating a **PICK** command with a negative entity **NUMBER**.

OPTION=FILTER filters the list of picked items with the current filter entity type, and removes any that do not match.

OPTION=CHANGE changes the list of picked items to those of the current filter entity type. If this type is **CELL**, this option will pick all cells in all picked bodies. If type is **FACE**, this option will pick all faces in all picked bodies and cells. The filter option is then applied to remove any entities not of this type.

The TYPE parameter of the PICK command will be used. If it is unset, the TYPE parameter of the FILTER command will be used instead.

- TYPE=CELL converts all picked bodies to picked cells, and unpicks any other entity type (such as faces, edges and vertices).
- **TYPE=FACE** converts all picked bodies and cells to picked faces, and unpicks any other entity type.
- **TYPE=EDGE** converts all picked bodies, cells and faces to picked edges, and unpicks any other entity type.
- TYPE=VERTEX converts all picked bodies, cells, faces and edges to picked vertices, and unpicks any other entity type.

As explained above, the process only works in one direction (from edges to vertices) not the other way.

OPTION=ALL selects all items of the filter entity type.

The **PICTURE** Command

Summary Saves the current display in a file or to the clipboard.

Icons



Menu Route	Edit \downarrow	
	Copy to file	
	Edit \downarrow	
	Copy to clipboard	ł

Command Line Parameters

Notes

Command	PICTURE			
Parameter	Default	Function		
SAVE	NO	NO	Copy file to clipboard	
		YES	Save picture in a file	
FILENAME		The n	ame of the file to be saved	
TYPE	PNG	PNG BMP XPM	Type of image file format used to save the picture	

This command allows the current display to be stored to the clipboard, or in a file.

If **SAVE=NO** the image is placed on the clipboard and can then be pasted into another application. Under UNIX, the image is placed on the X-Selection.

With **SAVE=YES**, the image is stored in a file. The filename can be specified in the **FILENAME** parameters, and the format can be selected with the **TYPE** parameter.

An image can be printed directly using the **PRINT** command (see "The **PRINT** Command" on page 3-119).

The **PRECISIONDATA** Command

|--|

Edit \downarrow

Menu Route

Tolerances

Command Line Parameters

Command	PRECISIONDATA			
Parameter	Default	Function		
ABSTOL	1.0e-6	The minimum distance between geometric objects within the model		
SMALLFACE	1.0e-6	The area of a face that is considered to be of negligible area		

Notes

This command controls the tolerance of operations within the Modeller. These tolerances should be set according to the size of the model being created. In general, these parameters should be set before building the model. Changing the minimum dimension when parts of the model have already been defined (especially if these are now outside the defined range) may cause some operations to fail.

The ABSTOL parameter sets the minimum distance at which points are treated as being independent. The ABSTOL parameter also defines the maximum dimension of the model. This is set to allow a dynamic range of 10^{10} for double precision data (with allowances for calculation tolerances), giving a maximum dimension of 10^{10} *ABSTOL, i.e. 10^4 by default. If the model does not sit comfortably within this dimension range, this tolerance should be adjusted accordingly.

During some Boolean operations, the tolerancing may be such that very small faces are created. The SMALLFACE parameter defines the area considered by the Modeller to be negligible. Any face of area less than this value is unwanted, and the Modeller tries to remove it. In some cases, this may fail and the user should take steps to improve the model. Small faces are likely to cause problems to the mesh generation, and if successful, the mesh is likely to be of lower quality near such a face.

The **PRINT** Command

Summary	Prints the current display.
Icon	
Menu Route	File↓ Print
Command Line Parameters	Command PRINT No parameters
Notes	This command allows the current display to be printed. A dialog of the available printers and their options for printing is opened. The current display can then be printed on the selected printer.

The printers available are determined from the system.

An image can be saved to file or copied to the clipboard using the PIC-TURE command (see "The PICTURE Command" on page 3-117).

The **PRISM** Command

Summary Creates an *n*-sided prism or pyramid, with points evenly distributed around a circular or elliptic base.

Icon



Menu Route

 $\begin{array}{c} \texttt{Create} \ \downarrow \\ \texttt{Object} \ \rightarrow \ \texttt{Prism/pyramid} \end{array}$

Command Line Parameters

Command	PRISM	
Parameter	Default	Function
NAME		Attaches this name to the body formed
SIDES		Number of sides on the base of the prism
HEIGHT		Height along the axis of the prism
MAJORRADIUS		Major radius of the base of the prism
MINORRADIUS		Minor radius of the base of the prism
TOPRADIUS		Radius at the top of the prism

Notes

All values must be given. At least 3 sides must be specified. The HEIGHT must be greater than zero.

If the **TOPRADIUS** is given as zero, the body formed is a pyramid.

The points needed to form the *n*-sided polygon are regularly positioned around a base ellipse given by the MAJORRADIUS and MINORRADIUS, and the sides of the polygon are connected by straight edges. The prism is formed by sweeping along the z-axis of the working coordinate system, a distance of HEIGHT. If TOPRADIUS is different from the major radius of the base, the prism is tapered. If the TOPRADIUS is zero, an *n*-sided pyramid is formed.

The body formed is centred on the origin of the working coordinate system (it extends from -height/2 to +height/2 in the z-direction).

OPERA-3d Reference Manual

The **RACETRACK** Command

Create or modify racetracks.		
0		
Create \downarrow Conductor $ ightarrow$ Racetrack Operations \downarrow Modify conductors $ ightarrow$ Racetrack		

Command Line Parameters

Command	RACET	TRACK			
Parameter	Default	Function			
OPTION	NEW	NEW	Creates a new racetrack con- ductor		
		MODIFY	Modifies properties of the picked racetrack conductors		
		LOAD	Loads defaults from picked conductors		
DRIVELABEL		Name for	the racetrack drive label		
LCNAME		Name for Local Coordinate System for coordinate system 1			
SYMMETRY		Rotational symmetry about global Z axis			
XCEN2		Origin of coordinate system 2			
YCEN2		†			
ZCEN2		†			
THETA2		Euler angles defining orientation of coor-			
PHI2		dinate system 2			
PSI2		†			
RXY		Reflection	symmetries in XY, YZ and ZX		
RYZ		planes			
RZX					
XP1		Inside low	ver on the racetrack cross-sec-		
YP1		tion			
Α		Cross-sect	tional width		

Command	RACET	RACK (continued)			
Parameter	Default	Function			
В		Cross-sectional height			
H1		Half-lengt	h of the straight		
R1		Radius of	the arc		
CURD		Current density in the conductor. This can be defined in terms of the current as cur- rent/AREA			
TOLERANCE		Field calcu	ulation tolerance		
INCIRCUIT		Is the conductor part of an external cir- cuit:			
		NO The conductor has defined c rent density.			
		YES	The current in the conductor is determined by an external circuit.		
REVERSE		Reverse the connections to this conductor in its circuit: YES or NO.			
CIRCUITELEMENT		The name of circuit element this conductor is part of.			
KEEP	NO	NO	Clear the list of picked items		
		YES	Keep the list of picked items for further modification		

Notes

This command creates a new racetrack conductor when using OPTION=NEW.

OPTION=MODIFY it is used to operate on the list of picked racetracks, changing geometry, coordinate systems, symmetries, current density, tolerance etc. The current common values of the picked racetrack conductors are available by calling **CONDUCTOR OPTION=LOAD**.

The default values of the command parameters are updated to match common values shared by all of the picked racetracks. If the picked racetracks do not have a common value for a parameter, that parameter is left unset. Using OPTION=MODIFY will change the conductor data of all of the picked racetracks to the new values given in the parameters, but will not affect the conductor data of any unset parameter.

For more information on the parameters, see "Racetracks" on page 4-29. For more information on connecting conductors to external circuits, see "The CIRCUIT Command" on page 3-51.

The **REDO** Command

Summary	Redo previously undone commands		
Icon	C		
Menu Route	Edit ↓ Bedo		
	Edit ↓ History		

Command Line Parameters

Notes

Command	REDO	
Parameter	Default	Function
STATE	none	Name of the state to which the history is to be undone

This command allows an UNDO command to be reversed, by forwarding the state of the model to a former position.

If no value for **STATE** is given, the operation forwards one state.

The REDO facility is available to go to future positions in the history stream after the UNDO command. The REDO command is available until a new entry is added into the history stream, i.e. a new command changes the current state of the model.

Note: If generating command scripts from log files generated during sessions, it is advisable to remove any UNDO or REDO commands that specify a state name, as the state name is dependent upon the number of commands issued. This makes it *very sensitive* to changes earlier in the script.

The **RENAME** Command

Summary	Changes the name on picked bodies or Local Coordinate System.

Menu Route Properties ↓ Rename

Command Line Command RENAME Parameters Command RENAME Parameter Default Function NAME Name to be used to replace the existing name Notes The RENAME command changes the name associated with the picked bodies or with a single Local Coordinate System.

If more than one Local Coordinate System is selected, the operation has no effect. Two Local Coordinate Systems may not have the same name. Renaming a Local Coordinate System will not affect any conductors that use this Local Coordinate System for describing coordinate system 1.

Bodies with the name **BACKGROUND** define the extent of the model space when creating the model body with the **MODEL** command.

The **SAVE** Command

Summary	Saves model data to file.		
Icon			
Menu Route	File ↓ Save Save as new model data Save model with mesh Operations ↓ Save picked components Export picked bodies		

Command Line					
Parameters	Command	SAVE			
	Parameter	Default	Function		
	OPTION	ALL	ALL	Saves all model data to the current file	
			NEW	Creates a new file containing all the model data	
			PICKED	Saves all picked bodies, Local Coordinate Systems and conduc- tors	
			EXPORT	Exports picked bodies to <i>sat</i> format file	
			MESH	Saves all model data and mesh data in binary format	
			IGES	Exports the picked bodies to IGES format file	
	FILE		File name	to be opened	
	FILEVERSION	12.0	SAT file v	ersion for exporting	

Note

I

SAVE will save a file containing all the model data. A saved file can be opened with the LOAD command (see page 3-96).

To save all the model data in a new .*opc* file, use **SAVE OPTION=NEW**. To overwrite a previously saved or loaded .*opc* file, use **SAVE OPTION=ALL**. Note that **OPTION=ALL** can only be used once the open file has been established by **LOAD OPTION=NEW** or **SAVE OPTION=NEW** commands.

To save a subset of the model, pick the bodies, Local Coordinate System or conductors and use OPTION=PICKED to save them to file.

An ACIS *.sat* file of any valid version number can be exported using **SAVE OPTION=EXPORT**.

An IGES .igs file can be exported using **SAVE OPTION=IGES**.

To save the whole model and the finite element mesh in a binary (*.opcb*) file, use **SAVE OPTION=MESH**. This does not change the open file name used by **SAVE OPTION=ALL**.

The **SELECT** Command

Summary Sciects items for display.

Icon



Menu Route

View \downarrow Selection

Command Line		
Parameters		

Command	SELEC	ELECT			
Parameter	Default	Function			
OPTION	ADD	ADD Selects an item for d			
			play		
		DEFAULT	Sets the default view options		
		HIDE	Explicitly hides an item from the display		
		PICKEDREMOVE	Temporarily hides all picked entities		
		REMOVE	Removes the hidden or displayed flag from an item		
		RESET	Clears all selected items		
		UNHIDE	Unhides all entities that have been hidden		
PROPERTY		Property of data for	selection		
LABEL		Label in the data of t	he specified type		
TYPE	none	BODY	Type of entity whose		
		CELL	data is to be shown		
		FACE			
		EDGE			
		VERTEX			
		LCS			
		CONDUCTOR			
NUMBER		Identifier for the item of the entity type			

Command	SELEC	(continued)		
Parameter	Default	Function		
PTU		Point nearest to or co	ontained within an entity	
PTV		of the selection TYPE		
PTW				
WAIT		YES	Do not update the dis-	
			play from the selection	
			change	
	ļ	NO	Update the display from	
	ļ		this selection	
AUTOUPDATE	YES	YES	Refresh the display after	
			every change	
	ļ	NO	Never update the display	
	ļ	CLOSEWINDOW	Closes the display win-	
	ļ		dow	
SELECTLEVEL	1	Set the options from	which display items can	
	ļ	be chosen		

Notes

The **SELECT** command allows a list of objects to be selected for display and a list of objects to be explicitly hidden. It provides facilities for viewing the different data attached to entities, differentiating between different data with different colours. It also allows parts of the model to be hidden. This is a stronger option than not being displayed, as it forces items to be hidden from the display.

OPTION=DEFAULT will clear all display items and reset the data to display any entities with properties of *Material*, *Coiltype*, *LCName* or *System*.

The selected and hidden items are completely cleared using OPTION=RESET.

OPTION=ADD selects an item or data characteristic for display.

OPTION=HIDE selects an item or data characteristic to be explicitly hidden from the display.

OPTION=REMOVE removes the display or hide selection from an item or data characteristic.

With OPTION=ADD, OPTION=REMOVE or OPTION=HIDE, there are three ways of specifying the selection:

• Data is attached to each of the entities. The TYPE parameter can be used to select a type of data item that is stored. For example PROP-

I

3-129

ERTY=BOUNDARY will select objects that have a boundary label attached.

By specifying a label the **SELECT** command can be refined, e.g. adding **LABEL=FACE1** in the above example would restrict the selection to entities with attached boundary condition label **FACE1**.

When using **OPTION=REMOVE** with only the **PROPERTY** parameter set, the selection of all individually selected items in that property list will also be removed.

It should be noted that different entity types will be selected by different properties. For example the *Material* property is attached to cells, so selecting **PROPERTY=MATERIAL** will select cells. The *boundary* property is attached to faces, so **PROPERTY=BOUNDARY** will select faces.

The colour with which each object is displayed will indicate the specific label of the property. Where the colour for the display of a section is ambiguous, the part is displayed with the specific colour indicating the ambiguity. Colours can be changed using the COLOUR command (see page 3-56).

 The second method for selection uses an entity TYPE with a number. This method is used with graphical selection when hiding an entity. The entity type can be BODY, CELL, FACE, EDGE, VERTEX, LCS or CONDUCTOR. Each object has a unique integer identifier and this number is specified in the NUMBER parameter. If NUMBER is not given, all objects of the given type are selected.

It is not recommended that this method be used in scripts. After operations such as **COMBINE** or **DELETE**, the numbering of the items will be changed. The method in which the numbering is applied cannot be guaranteed to be the same on subsequent runs or versions of the software, so use of this method may mean that the wrong items are selected.

 A third method uses an entity TYPE and a point coordinate (PTU, PTV, PTW), defined in the current Working Coordinate System. Only entities of type body, cell, face, edge or vertex can be selected by this method. For cells and bodies, the entity that contains the point is selected. For others, the entity nearest to the supplied point is selected. If there is more than one possible selection, the result is undefined.

OPTION=UNHIDE unhides all objects that have been hidden by the second and third methods described above. Objects hidden by **PROPERTY** or **LABEL** will not be affected.

The different selections are stored and can be used in any combination. For example,

SELECT OPTION=ADD TYPE=MATERIAL

will display all entities with material data (i.e. all cells). A second command of

SELECT OPTION=HIDE TYPE=MATERIAL LABEL=AIR

will mean that all materials are displayed except those with material label AIR. A further command of

SELECT OPTION=HIDE TYPE=POTENTIAL LABEL=REDUCED

will mean that any reduced potential regions are also excluded from the display.

The faces of objects are used to display the objects, i.e. bodies, cells and faces are displayed by drawing their faces. This means that the selection of the display is quite complex, as the choice of selected and hidden items will often conflict. The following rules are used to choose the colour and faces to be shown:

- 1. Highlight any picked item.
- 2. Determine any other parts of the model that have been hidden and flag these for exclusion from the display.
- 3. Include faces with associated data that have been selected for display, which are not hidden and which are not already displayed.
- 4. Include bodies with associated data that have been selected for display, which are not hidden and which are not already displayed.
- 5. Include cells with associated data that have been selected for display, which are not hidden and which are not already displayed.

It can be seen from the above order that the display will always include all picked items. **OPTION=PICKEDREMOVE** can be used to override this, so that any hidden object is temporarily removed from the display. Subsequent displays after issuing this command will revert to the normal display mode with picked items included.

A further option, **SELECTLEVEL=1** restricts the options to the basic information attached to the different entities. More options, e.g. the boundary condition type that is associated to a boundary label, are available with **SELECTLEVEL=2**.

By default any change to the model will cause the display to be refreshed. This behaviour can be changed using **AUTOUPDATE=NO** or **AUTOUPDATE=CLOSEWINDOW**. In the first case, the **THREED** command must be called to refresh the display to show changes. This should not normally be used during an interactive session as it will mean there is no visual feedback of any actions performed. The second option (**AUTOUPDATE=CLOSEWINDOW**.) closes the display window, and releases any memory being used by the display (for example to allow larger result files to be processed).

When making multiple selections the WAIT parameter allows the AUT-OUPDATE to be overridden until the next change. The WAIT parameter is always reset to NO, so must be explicitly declared with any SELECT command when needed.

The **SOLENOID** Command

Summary	Create or modify solenoids.

Icon



Menu Route	Create \downarrow
	$\texttt{Conductor} \ \rightarrow \ \texttt{Solenoid}$
	Operations \downarrow
	Modify conductors \rightarrow Solenoid

Command Line Parameters

Command	SOLENOID			
Parameter	Default	Function		
OPTION	NEW	NEW	Creates a new solenoid con-	
			ductor	
		MODIFY	Modifies properties of the	
			picked solenoid conductors	
		LOAD	Loads defaults from picked	
			conductors	
DRIVELABEL		Name for	the solenoid drive label	
LCNAME		Name for Local Coordinate System for		
		coordinate	e system 1	
SYMMETRY		Rotational	symmetry about global Z axis	
XCEN2		Origin of	coordinate system 2	
YCEN2				
ZCEN2				
THETA2		Euler angl	es defining orientation of coor-	
PHI2		dinate sys	tem 2	
PSI2				
RXY		Reflection	symmetries in XY, YZ and ZX	
RYZ		planes		
RZX				
XP1		First point	on the solenoid cross-section	
YP1				
Command	SOLEN	OID (continued)		
----------------	---------	---	--	--
Parameter	Default	Function		
XP2		Second po	int on the solenoid cross-sec-	
YP2		tion		
XP3		Third poin	t on the solenoid cross-section	
YP3				
XP4		Fourth poi	nt on the solenoid cross-section	
YP4				
CP1		Curvature	s of the 4 sides of the cross-sec-	
CP2		tion		
CP3				
CP4				
CURD		Current density in the conductor. This can		
		be defined	in terms of the current as cur -	
		rent/AR	EA	
TOLERANCE		Field calculation tolerance		
INCIRCUIT		Is the conductor part of an external cir-		
		cuit:		
		NO The conductor has defined current density.		
		YES	The current in the conductor is determined by an external circuit.	
REVERSE		Reverse the connections to this conductor		
		in its circuit: YES or NO.		
CIRCUITELEMENT		The name of circuit element this conduc-		
		tor is part of.		
KEEP	NO	NO	Clear the list of picked items	
		YES	Keep the list of picked items	
			for further modification	

Notes

This command creates a new solenoid when using OPTION=NEW.

OPTION=MODIFY it is used to operate on the list of picked solenoids, changing geometry, coordinate systems, symmetries, current density, tolerance etc. The current common values of the picked solenoid conductors are available by calling **CONDUCTOR OPTION=LOAD**.

The default values of the command parameters are updated to match common values shared by all of the picked solenoids. If the picked solenoids do not have a common value for a parameter, that parameter is left unset. Using OPTION=MODIFY will change the conductor data of all of the picked solenoids to the new values given in the parameters, but will not affect the conductor data of any unset parameter.

For more information on the parameters, see "Solenoids" on page 4-27. For more information on connecting conductors to external circuits, see "The CIRCUIT Command" on page 3-51.

The **SOLVERS** Command

<i>Summary</i> Creates a new database, or adds a new simulation	to an existing database.
---	--------------------------

Menu Route	Model \downarrow		
	Create	analysis	database

Command Line Parameters

Command	SOLVERS			
Parameter	Default	Function		
PROGRAM	TOSCAMAGN	Analysis program typ	be	
		TOSCAMAGN Tosca: magnetic,		
		TOSCAELEC	trostatic or current	
		TOSCACURRENT	flow	
		SCALA	Scala: Space charge	
		ELEKTRASS	ELEKTRA: Steady	
		ELEKTRATR	state harmonic, tran-	
		ELEKTRAVL	sient or velocity (lin-	
		ELEKTRARO	ear and rotational)	
		SOPRANOSS	Soprano: Steady state	
		SOPRANOEV	harmonic or eigen-	
			value	
		TEMPOST	TEMPO: thermal	
		TEMPOTR	analysis, steady-state	
		Nama of the detabase	of transferit	
OPTION			file	
		ADD	Add a new simulation	
			to an existing database	
			file	
UNITS	CGS	Specify the unit set to be used to write the		
		database		
		CGS	CGS units	
		METRE	SI units	
		MM	SI units with MM	
		MICRON	SI units with microns	
		INCH	SI units with inches	

Command	SOLVERS (con	ntinued)			
Parameter	Default	Function			
ELEMENT	MIXED	Type of elements to b	be created		
		MIXED	Use element type		
			defined by cell prop-		
			erty		
		LINEAR	Use all linear elements		
		QUADRATIC	Use all quadratic ele-		
			ments		
SURFACE	CURVED	Type of elements to b	be created in all ele-		
		ments that touch surf	aces		
		MIXED	Use element type		
			defined by face prop-		
			erty		
		CURVED	As mixed, but curved		
			surfaces default to		
			quadratic		
		LINEAR	Use all linear elements		
		QUADRATIC	Use all quadratic ele-		
			ments		

Notes

The solvers command creates a new database or adds a new simulation to an existing database. When adding a simulation to a database, the UNITS, ELEMENT and SURFACE parameters are not used.

The type of analysis to be used is set by the **PROGRAM** parameter.

The type of elements are controlled by the ELEMENT and SURFACE parameters. The ELEMENT parameter allows the type of volume elements in cells to be controlled. The SURFACE parameter allows the type of volume elements that touch a face to be controlled.

The **SPHERE** Command

Summary	Creates a sphere.
Icon	

Menu Route

 $\begin{array}{c} \texttt{Create} \ \downarrow \\ \texttt{Object} \ \rightarrow \ \texttt{Sphere} \end{array}$

Command Line Parameters

	-	
Command	SPHER	E
Parameter	Default	Function
NAME		Attaches this name to the body formed
X0		X coordinate of centre
Y0		Y coordinate of centre
Z0		Z coordinate of centre
RADIUS		Radius of the sphere

Notes All values must be given. A sphere of the given RADIUS is formed, centred on the coordinate (Z0, Y0, Z0).

The coordinates specified are in the Working Coordinate System.

The **STRAIGHT** Command

Summary sharging.

Icon

 \mathbf{N}

Menu Route	Create \downarrow
	Conductor $ ightarrow$ Straight bar
	Operations \downarrow
	Modify conductors $ ightarrow$ Straight bar

Command Line Parameters

Command	STRAIC	SHT	
Parameter	Default	Function	
OPTION	NEW	NEW	Creates a new straight conduc-
			tor
		MODIFY	Modifies properties of the
			picked straight conductors
		LOAD	Loads defaults from picked
			conductors
DRIVELABEL		Name for	the drive label
LCNAME		Name for	Local Coordinate System for
		coordinate	e system 1
SYMMETRY		Rotational	symmetry about global Z axis
XCEN2		Origin of	coordinate system 2
YCEN2			
ZCEN2			
THETA2		Euler angl	es defining orientation of coor-
PHI2		dinate sys	tem 2
PSI2			
RXY		Reflection	symmetries in XY, YZ and ZX
RYZ		planes	
RZX			
A		Cross-sect	tional width
В		Cross-sect	tional height
H1		Length of	the straight

Command	STRAIG	GHT (cont	HT (continued)		
Parameter	Default	Function			
CURD		Current de	ensity in the conductor. This can		
		be defined in terms of the current as			
		current/AREA			
TOLERANCE		Field calcu	ulation tolerance		
INCIRCUIT		Is the conductor part of an external cir-			
		cuit:			
		NO	The conductor has defined current density.		
		YES	The current in the conductor is determined by an external circuit.		
REVERSE		Reverse the connections to this conductor in its circuit: YES or NO.			
CIRCUITELEMENT		The name of circuit element this conductor is part of.			
KEEP	NO	NO Clear the list of picked items			
		YES	Keep the list of picked items for further modification		

Notes

This command creates a new straight conductor when using OPTION=NEW.

OPTION=MODIFY it is used to operate on the list of picked straights, changing geometry, coordinate systems, symmetries, current density, tolerance etc. The current common values of the picked straight bar conductors are available by calling **CONDUCTOR OPTION=LOAD**.

The default values of the command parameters are updated to match common values shared by all of the picked straights. If the picked straights do not have a common value for a parameter, that parameter is left unset. Using OPTION=MODIFY will change the conductor data of all of the picked straights to the new values given in the parameters, but will not affect the conductor data of any unset parameter.

For more information on the parameters, see "Straight Bars" on page 4-35. For more information on connecting conductors to external circuits, see "The CIRCUIT Command" on page 3-51.

3-140

The **STRETCH** Command

Summary	Stretches the	geometry o	f bodies
		0 2	

 $\begin{array}{ccc} Menu \ Route & & \\ &$

Command Line Parameters

Command	STRETC	H		
Parameter	Default	Function		
U1		Position of	f the starting point of the stretch	
V1				
W1				
DISP1		Displacement of the starting point		
U2		Position of the end point of the stretch		
V2				
W2				
DISP2		Displacement of the end point		
KEEP	NO	NO	Keep the picked body for further	
		YES	operations	

Notes The **STRETCH** command allows bodies to be transformed to a new shape by stretching space. The stretch is defined by 2 coordinates and a displacement at each. A positive displacement indicates a shift along the vector from point 1 to point 2.

Any part of picked bodies that lie between these 2 coordinates will be stretched.

The **SWEEP** Command

Summary	Sweeps a picked face through space to form a new body, or to extend the
	existing body owning the face.

Menu Route	Operations \downarrow
	Sweep face

Command Line Parameters

Command	SWEEP			
Parameter	Default	Function		
TYPE	DISTANCE	DISTANCE Sweep a fixed distance		
			along the face normal	
		VECTOR	Sweep along a specified	
			vector	
		ROTATION	Sweep a fixed angle about	
			an axis	
		PATH	Sweep along a path given	
			by edge or wire body.	
DISTANCE		Distance to be sw	vept through, TYPE=DIS-	
		TANCE only		
DU		Vector to sweep	along, TYPE=VECTOR	
DV		only		
DW				
ROTU		Axis of rotation, TYPE=ROTATION only		
ROTV				
ROTW				
POSU		Point on the axis	of rotation, TYPE=ROTA-	
POSV		TION only		
POSW				
ANGLE		Angle of rotation	, TYPE=ROTATION only	
REGULARISE	NO	NO	Leaves the starting face	
		YES	Removes the original face	
			if part of a body	
DRAFTTYPE	NONE	NONE	No draft during the sweep	
		ANGLE	The area of the swept face	
			increases with the draft	
			angle	

Command	SWEEP (c	ontinued)	
Parameter	Default	Function	
DRAFTANGLE		The draft angle	
TWISTTYPE	NONE	NONE	No twist
		ANGLE	Twist through a fixed
			angle
		PARAMETRIC	Twist through a angle
			specified as a function of
			position along the edge
TWISTANGLE			Fixed angle of twist
TWISTEXP			Parametric expression
			defining twist
RIGID	NO	NO	Keep the face normal
		YES	fixed during the sweep.

Only a single planar face may be swept in the command.

The sweep is controlled by the **TYPE** parameter.

If TYPE=DISTANCE, the face is swept along its normal. Vectors representing the normal direction can be viewed on picked faces with the VEC-TOR command. This normal direction can be toggled by picking one of these vectors.

TYPE=VECTOR will sweep along the specified vector.

TYPE=ROTATION allows sweeping about an angle around an axis passing through a point.

TYPE=PATH allows sweeping along an edge or wire body (i.e. a body comprised of only wire edges). To perform this operation both a face and edge or face and wire body should be picked.

For TYPE=VECTOR, TYPE=ROTATION and TYPE=PATH, care should be taken to avoid sweeping a face where the normal to the face is almost perpendicular to the direction.

For **TYPE=PATH**, if the path being swept does not start on the plane of the face, the sweep will occur in both directions up to the start and finish of the path. If the path does not intersect the plane of the face, unpredictable results may occur.

A twist may be specified for TYPE=PATH. This twist uses the path as the centre of twist, and may be specified as an angle (in degrees) or as a parametric expression of the distance along the edge, U. It is not possible to

Notes

describe the underlying algorithm used by the geometric modelling kernel when sweeping with parametric twists, so this facility should be used with care.

Care should also be taken to avoid sweeping a face into another part of the same body. This can cause errors in the body formed. Some of these difficulties may be resolved, but other operations may progress and form inconsistent bodies. Such bodies will be highlighted using the CHECK command, but cannot be easily corrected.

When sweeping, the face can be changed in area by using the draft angle.

The coordinates specified are in the Working Coordinate System.

The TANGENTIALCPE Command

Summary	Create or modify tangential constant perimeter end (CPE) conductors.
---------	--

Icon

0

Menu Route	Create \downarrow
	Conductor $ ightarrow$ Tangential constant perimeter end
	Operations \downarrow
	Modify conductors $ ightarrow$ Tangential constant perimeter end

Command Line Parameters

I

Command	TANGE	ENTIALCPE		
Parameter	Default	Function		
OPTION	NEW	NEW	Creates a new tangential CPE conductor	
		MODIFY	Modifies properties of the picked tangential CPE conductors	
		LOAD	Loads defaults from picked conductors	
DRIVELABEL		Name for	the drive label	
LCNAME		Name for Local Coordinate System for coordinate system 1		
SYMMETRY		Rotational symmetry about global Z axis		
XCEN2		Origin of	coordinate system 2	
YCEN2		-		
ZCEN2		-		
THETA2		Euler angl	les defining orientation of coor-	
PHI2		dinate sys	tem 2	
PSI2		-		
RXY		Reflection symmetries in XY, YZ and ZX		
RYZ		planes		
RZX		-		
ALPHA		Azimutha	l angular position of the straight	
BETA		Cutter ang	gle	

Command	TANGE	NTIALCPE (continued)		
Parameter	Default	Function		
A		Cross-sect	tional width	
В		Cross-sect	tional height	
H1		Half lengt	h of the straight	
R1		Radius of	forming cylinder	
R2		Radius of	cross-over arc	
CURD		Current de	ensity in the conductor. This can	
		be defined	l in terms of the current as	
		current	AREA	
TOLERANCE		Field calcu	ulation tolerance	
INCIRCUIT		Is the conductor part of an external cir- cuit:		
		NO The conductor has defined current density.		
		YES The current in the conductor is determined by an external circuit.		
REVERSE		Reverse the connections to this conductor		
		in its circuit: YES or NO.		
CIRCUITELEMENT		The name of circuit element this conduc-		
		tor is part of.		
KEEP	NO	NO Clear the list of picked items		
		YES Keep the list of picked items		
		for further modification		

Notes

This command creates a new tangential CPE conductor when using OPTION=NEW.

OPTION=MODIFY it is used to operate on the list of picked tangential CPE conductors, changing geometry, coordinate systems, symmetries, current density, tolerance etc. The current common values of the picked tangential CPE conductors are available by calling **CONDUCTOR OPTION=LOAD**.

The default values of the command parameters are updated to match common values shared by all of the picked tangential CPE conductors. If the picked tangential CPE conductors do not have a common value for a parameter, that parameter is left unset. Using OPTION=MODIFY will change the conductor data of all of the picked tangential CPE conductors to the new values given in the parameters, but will not affect the conductor data of any unset parameter. For more information on the parameters, see "Constant Perimeter Ends" on page 4-33. For more information on connecting conductors to external circuits, see "The CIRCUIT Command" on page 3-51.

The THREED Command

Summary	Control	the d	isplay	of the	geometry.
					0 1

Icons



Menu Routes

View↓ Refresh Set view... Views

Command Line Parameters

Command	THREED			
Parameter	Default	Function		
OPTION		Command option:		
		GETVIEW	Retrieve view parameters	
			after mouse interaction.	
		INIT	Refresh picture without	
			changing the view.	
		SETVIEW	Refresh picture using the	
			view parameters.	
SIZE	10	Display exte	ends from the origin by SIZE	
		in each direc	ction. SIZE=0 requests the	
		initial view	of the model	
ROTX	20	Rotation of model around X axis.		
ROTY	20	Rotation of model around Y axis.		
ROTZ	0	Rotation of model around Z axis.		
XORIGIN	0	X coordinate	e at centre of picture	
YORIGIN	0	Y coordinate	e at centre of picture	
ZORIGIN	0	Z coordinate	e at centre of picture	
PERSPECTIVE	YES	Perspective	switch:	
		YES	Perspective view.	
		NO	Orthographic view.	
LINECOLOUR	YES	Colour used for outlines:		
		YES	Material colour.	
		NO	Edge colour.	

Notes The **THREED** command updates the 3D picture of the model. The pictures consists of the three dimensional geometry of the model and conductors. The view can be adjusted using the mouse buttons or by setting explicitly using this command.

- OPTION=GETVIEW: updates the values of the parameters SIZE, ROTX, ROTY, ROTZ, XORIGIN, YORIGIN and ZORIGIN.
- OPTION=SETVIEW: uses the current values of SIZE, ROTX, ROTY, ROTZ, XORIGIN, YORIGIN and ZORIGIN.
- OPTION=INIT: updates the picture without changing the view. If automatic update is switched off using the command SELECT AUTOUP-DATE=NO, this command option must be called to update the display.

Perspective view can be switched off using **PERSPECTIVE=NO**.

The TITLE Command

Summary Add title, date and time to the display.

Menu Route

View↓ Title

Command Line Parameters

I

Command	TITLE				
Parameter	Default	Function			
STRING	none	A graphics window tit	le.		
POSITION	NONE	Title position:			
		BOTTOMCENTRE	Bottom centre		
		BOTTOMLEFT	Bottom left		
		BOTTOMRIGHT	Bottom right		
		NONE	No title		
		TOPCENTRE	Top centre		
		TOPLEFT	Top left		
		TOPRIGHT	Top right		
DATE	TOPLEFT	Time/date position:			
		BOTTOMCENTRE	Bottom centre		
		BOTTOMLEFT	Bottom left		
		BOTTOMRIGHT	Bottom right		
		NONE	No date and time		
		TOPCENTRE	Top centre		
		TOPLEFT	Top left		
		TOPRIGHT	Top right		

Notes

The TITLE command controls the display of a title and the date and time. There is a choice of 6 positions for each. If the same position is chosen for both the title and the date, the title appears above the date and time. The version number of the software can be included in the title using the string and system variables, VERSION:

title string='Version: &version&'

or

```
title string='Version: %real(version,5)'
```

The **TORUS** Command

Summary	Creates a torus.
Icon	0
Menu Route	$\begin{array}{l} \texttt{Create} \ \downarrow \\ \texttt{Object} \ \rightarrow \ \texttt{Torus} \end{array}$

Command Line Parameters

Command	TORUS	
Parameter	Default	Function
NAME		Attaches this name to the body formed
X0		X coordinate of centre
Y0		Y coordinate of centre
Z0		Z coordinate of centre
MAJORRADIUS		Major radius of the torus
MINORRADIUS		Minor radius of the torus

Notes All values must be given. A torus with the given radii is formed, centred on the coordinate (Z0,Y0,Z0).

The coordinates specified are in the Working Coordinate System.

The **TRANSFORM** Command

Summary Transforms or copies bodies, cells, edges, local coordinate systems and conductors.

Menu Route	Operations \downarrow
	Transform
	Operations \downarrow
	Сору

Command Line

Parameters

Command	TRANSFOR	RM	
Parameter	Default	Function	
OPTION	APPLY	APPLY	Apply the transformation to the picked items
		COPY	Creates a copy of the new item and applies the transformation to it
TYPE	DISPLACE	DISPLACE	Use a displacement transforma- tion
		ROTATE	Use rotation about an axis
		REFLECT	Reflect in a plane
		SCALE	Apply anisotropic scaling
		EULER	Rotate by Euler angle set
DU	0	Translation c	component in the working coordi-
DV	0	nate system	
DW	0		
ROTU	0	Rotation axis	s in the working coordinate system
ROTV	0		
ROTW	1		
ANGLE	0	Angle of rota	ation about the given axis
NU	0	Normal vector	or to the plane of reflection
NV	0		
NW	1		
SCU	1	Scaling facto	ors in the working coordinate sys-
SCV	1	tem	
SCW	1	1	

3-153

Command	TRANSFORM (continued)		
Parameter	Default	Function	
THETA	0	Values of Eu	ler angle rotations
PHI	0		
PSI	0		
LABEL		Adds a label	to each item transformed or copied.
COUNT	1	Number of co	opies to be created
KEEP	NO	NO	Reset the list of picked objects
		YES	Keep the list of picked objects

Notes

The **TRANSFORM** command is used to transform the geometry of existing items, or to create a copy and transform this.

The **TYPE** of transformation determines which of the parameter values are used. All others are ignored. The **COUNT** parameter is only used when copying, and allows multiple copies to be produced from a single command.

Tuno	Deremators
Type	Farameters
DISPLACE	DU, DV, DW
ROTATE	ROTU, ROTV, ROTW, ANGLE
REFLECT	NU, NV, NW
SCALE	SCU, SCV, SCW
EULER	THETA, PHI, PSI

If bodies are picked, they can be transformed using OPTION=APPLY, or OPTION=COPY can be used to generate a copy of these bodies that are then transformed.

If any Local Coordinate Systems or conductors are picked, the SCALE and REFLECT transformations will not have any effect on these objects, as they must remain with the same handedness and size.

Picked cells, faces, edges and vertices are unaffected by transformations.

If only cells have been picked, the OPTION=COPY operation can be used to create a new body for each cell that is copied.

If only faces have been picked, the OPTION=COPY operation can be used to create a new body for each face. This body then contains only a copy of this face. The body formed will enclose no volume, but the face may be selected and used in sweep or other operations.

If only edges have been picked, the OPTION=COPY operation can be used to create a new body for each edge. This body then contains only a

copy of this edge. The body formed will enclose no volume, but the edge may be selected and used in sweep or other operations.

When transforming or copying conductors, the operator modifies either Local Coordinate System 1 or Local Coordinate System 2 of the conductor:

- If Local Coordinate System 1 is the current Working Coordinate System, Local Coordinate System 2 is changed.
- If Local Coordinate System 1 is not the Working Coordinate System, a copy of Local Coordinate System 1 is created and transformed, and the conductor's Local Coordinate System 1 is changed to point to the new Local Coordinate System.

The transformed items will all have the LABEL attached as a user label. If copying, the original items will be given the label. All new items created by copying will be given the label suffixed by a count from 1 to COUNT, i.e. labeltext1, labeltext2 etc. This allows the items created by copying to be easily grouped according to the number of times that the transformation was applied to the original items.

The KEEP parameter can be used if more than one transformation is to be applied to the picked set of objects, to avoid needing to re-pick them.

The **TWIST** Command

Summary	Twists the	geometry	of bodies
	I WISto the	Scomeny	01 000000

Menu Route	Operations \downarrow
	Morphing $ ightarrow$ Twist

Command Line Parameters

Command	TWIST		
Parameter	Default	Function	
U1		Position of	the starting point of the twist
V1			
W1			
ANGLE1		Displacem	ent of the starting point
U2		Position of	the end point of the twist
V2			
W2			
ANGLE2		Displacem	ent of the end point
CONTINUITY	0	Level of co	ontinuity at the ends of the twist
KEEP	NO	NO	Keep the picked body for further
		YES	operations

Notes

The **TWIST** command allows bodies to be transformed to a new shape by twisting space. The twist is defined by 2 coordinates and a twist at each of them. The twist takes place around an axis directly between the 2 points given.

Any part of picked bodies that lie between these 2 coordinates will be twisted. A continuity level (0, 1 or 2) can be specified to control the level of surface continuity if the twist region does not enclose all of the body. Level 0 will have a sharp join, while level 2 will enforce continuity of the second derivatives at the interface.

The UNDO Command

Summary	Rewinds the history to a specified position.
Icon	2
Menu Route	Edit↓ Undo
	History

Command Line Parameters

Command	UNDO	
Parameter	Default	Function
STATE	none	Name of the state to which the history is to be undone

Notes

This command returns the state of the model to a former position.

If no value for **STATE** is given, the operation rewinds one previous state.

Most operations add an entry to the history stream. This entry is given a unique **STATE** name (based upon the command name and the number of commands so far). Any state name within the list can be specified in the **STATE** parameter. This allows the model to be returned to any previous state by a single command.

After an UNDO command, a REDO facility is available to go back to future positions in the history stream (see "The REDO Command" on page 3-123). The REDO command is available until a new entry is added into the history stream.

Note: If generating command scripts from log files generated during sessions, it is advisable to remove any UNDO or REDO commands that specify a state name, as the state name is dependent upon the number of commands issued. This makes it *very sensitive* to modifications earlier in the script.

The VARIABLE Command

Summary Sets a user defined variable.

Icon



Menu Route

Create \downarrow Variables

Command Line Parameters

Command	VARIABLE		
Parameter	Default	Function	
OPTION	none	CONSTANT	Value is constant
		PARAMETER	Expression is stored for later evaluation
		DELETE	Deletes the named variables
		LIST	Lists all stored variables
NAME	none	Variable name	•
VALUE	none	Value or expres	sion assigned to the variable

Notes

I

This command creates and changes the values of user defined variables. Such variables must begin with the '#' character and be a total length of not more than 16 characters.

The command replaces the use of the **\$CONSTANT** and **\$PARAMETER** commands, and allows history facilities for **UNDO/REDO**, as well as storage of the variables when loading and saving models.

OPTION=DELETE removes a stored variable, although this may not be possible if the variable forms part of an expression which is in use in the Modeller.

OPTION=LIST shows the current set of variables and their values.

The **\$CONSTANT** and **\$PARAMETER** commands can be used, but the effects are not stored immediately within the history, so cannot be explicitly undone.

These variables can be used as part of any expression, and so can help to automate command scripts or to assist in model definition by allowing common values to be defined.

The **VECTOR** Command

Summary Displays vectors of attached data and properties on the model.

Menu Route

View \downarrow Vectors

Command Line

Parameters

Command	VECTO	R	
Parameter	Default	Function	
COMPONENT	NONE	А	Magnetic vector potential
			of a face boundary condi-
			tion
		CONDUCTOR	Conductor current
		FACEDIRECTION	Direction of faces for use
			with sweeping and emit-
			ters
		INA	Incident magnetic vector
			potential of a face bound-
			ary condition
		J	Current density set in the
			volume property of a cell
		NONE	No vectors will be dis-
			played
		ORIENTATION	Local orientation set in
			the volume property of a
			cell
		V	Linear velocity set in the
			volume property of a cell
SCALE	1	Scale factor for sizin	g the vectors
MAGNITUDE	YES	NO	Display all vectors with
			LENGTH=SCALE
		YES	Display vectors with their
			magnitude set by
			COMPONENT*SCALE

Notes

This command displays vectors of the **COMPONENT** on displayed objects in the model. If an object has data containing the component, then

the vectors of the data will be displayed on it. Other objects will be displayed normally.

COMPONENT=FACEDIRECTION vectors will only be displayed on selected faces.

The vectors displayed can be scaled in size by the value of SCALE. Those vectors that have a magnitude can be displayed either with size given by the magnitude of the COMPONENT multiplied by SCALE (MAGNI-TUDE=YES), or with a constant size of SCALE (MAGNITUDE=NO).

The VERTEXDATA Command

Summary Sets properties of picked vertices.

Menu Route Properties ↓ Vertex properties

Command Line				
Parameters	Command	VERTEX	DATA	
	Parameter	Default	Function	
	OPTION	MODIFY	MODIFY	Applies new values to the picked verti-
				ces
			RESET	Clears all data from the picked vertices
	SIZE	See notes	Mesh cont	trol size
	LEVEL	See notes	Data stora	ge level for the vertex data

Notes

This command is used to set or clear the properties of all picked vertices. Vertices initially have no data assigned to them.

If issuing the command with OPTION=MODIFY, the new value of parameters that have been set replace the existing values of data on the vertices. The value of data, associated with any of the unset parameters, is left unchanged.

The default value of each of the parameters is updated to be the common value of all the picked vertices. If the data of one of these parameters is unset, or the picked vertices do not share the same value, then the parameter value is left clear.

SIZE sets the maximum element side length of any element touching that vertex.

Upon issuing the command with OPTION=MODIFY, all picked items are de-selected. Issuing the command with OPTION=RESET keeps the same set of picked objects.

The LEVEL parameter controls the storage of data when there is a conflict during the merging of multiple cells. The data set with the greater level will be maintained. The result of merging 2 cells with the same level is indeterminate.

The VOLUME Command

abel.
5

Menu Route	Model \downarrow		
	Set	volume	properties

Command Line

Parameters

Command	VOLUN	JUME				
Parameter	Default	Function				
OPTION		PICK	Adds a volume label to a list of labels to be set			
		UNPICK	Clears the list of volume labels to be set			
		RESET	Clears the data from the			
			picked labels			
		MODIFY	Sets the data for the picked labels			
		LIST	Lists the properties of picked labels			
		DELETE	Deletes the picked labels			
VOLUMELABEL		Volume label to b	e picked for modification			
VX		Components of lin	near velocity for			
VY		ELEKTRA-VL				
VZ						
JX		Components of vo	olume source current density.			
JY						
JZ						
THETA		Components of or	ientation for anisotropic			
PHI		material propertie	s and easy direction of mag-			
PSI		netisation in perm	anent magnets			
PACKING		Packing factor for	packed materials			
ROTATION		Rotational velocity				
CHARGE	ARGE Volume charge density					

Command	VOLUN	IE (continued)				
Parameter	Default	Function				
HEATTYPE		Source of heat for TEMPO analysis				
		NONE	No heat source			
		VALUE	Specified functional heat source			
		NODALTABLE	Heat source is to be read from a table of values in the <i>OP3</i> database, RNODALHEAT			
		ELEMENTTABLE	Heat source is to be read from a table of values in the <i>OP3</i> database, RELE- MENTHEAT			
HEATVALUE		The functional speci	ification of the heat source			

Notes This command defines the volume characteristics for use by the analysis programs.

A set of volume labels is picked using the command repeatedly, with OPTION=PICK and a VOLUMELABEL specified. A volume label can be removed from the set using OPTION=UNPICK. If no VOLUMELABEL is given, the set is emptied.

Issuing the command with OPTION=MODIFY will modify the properties of the set of picked volume labels to the new values given in the parameters. The value of a property associated with the volume labels is unchanged if the parameter is unset.

The default value of each of the parameters is updated to be the common value of all the picked volume labels. If the data of one of these parameters is unset, or the picked volume labels do not share the same value, then the parameter value is left clear.

OPTION=RESET will clear the properties associated with all of the picked volume labels.

The properties of all volumes with the label VOLUMELABEL can be listed using OPTION=LIST.

Volume labels that are not used, i.e. have no cell referencing them can be deleted using OPTION=DELETE. Deleting a volume label that is in use will reset its properties.

All parameters can be specified. Which of these values will be used by the analysis will depend upon the analysis module being used, e.g. the velocity components will only be used by ELEKTRA-VL.

When HEATTYPE=NODALTABLE, the TEMPO analysis will require that a vector of heat density values at every node in the model is added into the database before the analysis. This can be done using the TABLE command in the post processor. When HEATTYPE=ELEMENTTABLE, the value of heat density values at the centroid of each element must be added into the database.

The WCS Command

Summary	Sets the	Working	Coordinate	System	(WCS).
---------	----------	---------	------------	--------	--------

Menu Route	Operations \downarrow	
	Set working coordin	nate system

Command Line Parameters

Command	WCS		
Parameter	Default	Function	
OPTION		SET	Set Working Coordinate System to Local
			Coordinate System given by LCNAME
		PICKED	Set Working Coordinate System to be the
			picked Local Coordinate System
		UNSET	Use the Global Coordinate System as the
			Working Coordinate System
LCNAME		Name of	the Local Coordinate System to be set as
		Working (Coordinate System

Notes

This command controls the Working Coordinate System. The Working Coordinate System is a coordinate system in which many commands operate, and it can be used to assist in the definition of geometries that do not lie neatly in the Global Coordinate System. The commands affected are those that create new objects, i.e. BLOCK, CYLINDER, PRISM, SPHERE, TORUS and LCS. It also operates with the TRANSFORM and SWEEP commands (when specifying positions, vectors and axes of rotation) and the morphing commands MORPH, TWIST and STRETCH.

Local Coordinate Systems can be defined with the LCS command. If OPTION=SET, a value must be given for LCNAME and it must correspond to the name of a Local Coordinate System within the model.

If a single Local Coordinate System has been picked, the OPTION=PICKED will set it to be the current Working Coordinate System.

OPTION=UNSET will unset the Working Coordinate System, so that the Global Coordinate System will be used.

3-166

The WINDOW Command

Show of mac parts of the alspia

Icons



Menu Route	View	\downarrow			
		Parts	of	the	display

Command Line Parameters

Command	WINDO	W
Parameter	Default	Function
AXES	YES	Show coordinate axes: YES or NO
SOLID	YES	Show solid view of model: YES or NO
OUTLINE	YES	Show outline view of model: YES or NO
VECTORS	YES	Show vectors on the surface of the model: YES or NO
MESH	YES	Show surface mesh YES or NO

The WINDOW command can be used to hide or show again parts of the display which exist. For example, if a surface mesh exists, it can be hidden using **WINDOW MESH=NO** or shown again using **WINDOW MESH=YES**.

Chapter 4 OPERA-3d Pre Processor

Introduction

The OPERA-3d pre processor can prepare data for the electromagnetic field analysis programs including CARMEN, ELEKTRA, SCALA, SOPRANO and TOSCA. The analysis programs use finite elements to model three dimensional electromagnetic devices, with the added facility for direct evaluation of fields from conductors carrying prescribed currents. The pre processor provides full support for all the features of the analysis programs.

The OPERA-3d pre processor is an interactive program that is used to create and edit three dimensional finite element models, define material characteristics for non-linear magnetic or dielectric components, assign boundary conditions, specify complicated conductor geometries with prescribed excitation, show the models using wire frame and hidden surface displays and output data files in the formats accepted by the analysis programs.

When the pre processor starts, on UNIX operating systems, the first input the user must give is 'device nomination'. This tells the program what graphics options should be used. This is described with the DEVICE command (page 4-94) which can be used at any time to reset the device, or change graphics options.

After device nomination, or when the OPERA-3d pre processor is restarted with the CLEAR command, the program looks for a file called *oppre.comi*, first in the current file directory and then in the user's home directory. If such a file exists it is read into the program as a \$ COMINPUT file (page 2-25). This allows the user to reset the default values of certain commands, e.g. COLOUR (page 4-22) or define frequently used

I

\$ CONSTANTS and **\$ PARAMETERS** (page 2-21) each time the program is started.

Two methods of command and data entry are available:

- 1. Menu system or GUI (Graphical User Interface) command selection and data specification are carried out under mouse control.
- 2. Command line input command selection and data specification are carried out from the keyboard.

Under normal operation, the pre processor starts in the GUI mode.
The Graphical User Interface

The GUI is built from 8 types of input window which are selected and controlled by pointing with the cursor and clicking a mouse button. Some input windows accept characters typed at the keyboard. The input windows are:

- Horizontal menu: Only used for top level menu.
- Vertical menu: For selecting commands and options.
- ParameterBox: For entering numerical or character data.
- **DialogBox**: Combination of text inputs and switches.
- FileBox: For selection of files.
- **CDBox**: For selection of current directory or folder.
- ColourBox: For re-defining colours.

The GUI also uses MessageBoxes to display messages and questions on the display.

Menus

Menus are horizontal or vertical lists of keywords which indicate actions to be performed. Menu items are selected by pointing with the mouse and clicking its left button. When the mouse is pointing at a menu item, that item is highlighted.

Alternatively, menu items can be chosen using the keyboard arrow keys: \leftarrow and \rightarrow for a horizontal menu or \uparrow and \downarrow for a vertical menu. When the required item is highlighted, it can be selected using the **<Enter>**, **<Return>** or \dashv keys.

Selecting a menu item can have one of several effects; the action is indicated by a symbol at the right-hand side of the item:

Symbol	Action	
Ŷ	Drop Down: this activates a sub-menu. It only exists in the top- level horizontal menu.	
	Pull Right: activates a sub-menu.	
\rightarrow	Pick and Pull: activates a sub-menu after a selection from the displayed model (see Pick below).	
	Return: returns to higher-level menu.	

Symbol	Action (continued)		
	Toggle: swaps between 2 options and the symbol changes		
	between I and I . The current state of the program is displayed.		
	Option: chooses one from a set of options. The current choice		
	is indicated by 📕 (filled with red).		
+	Pick: must be followed by a selection from the displayed model. This is done by positioning the cursor (which changes		
	shape to $+$) over the required part of the model and pressing the		
	left mouse button.		
	Rubber-box: must be followed by selection of diagonally oppo- site corners of a rectangle. This is done by pressing the left mouse button with the cursor at one corner and dragging the mouse, with the button held down, releasing it at the opposite corner. The menus are automatically hidden while the rubber- box is being used		
*	Action: executes a command or requests additional information via a ParameterBox or both; sometimes the menu will close after the specified action.		

The **<Esc>** key can be used to escape from a menu without any actions. If the menu does not allow Pick operations, selecting from a higher level menu can also be used to close it.

Not all menu items can be used at all times. For example, it is not possible to create a mesh before any volumes have been defined. Unavailable menu items are displayed in pale-blue rather than white until they become available as the result of other commands.

Parameter Boxes

ParameterBoxes (Figure 4.1) are used to input information from the key-

Size	= 10
X coordinate of eye	= 2
Y coordinate of eye	= 1
Z coordinate of eye	= 3
Accept	Dismiss

Figure 4.1 A Typical ParameterBox

board. Default values (if they exist) are displayed and are initially highlighted. When a value is highlighted it can be replaced by the first characters typed. A value can be edited by moving the text cursor before typing any characters keys.

Most ParameterBoxes have **Accept** and **Dismiss** below the list of parameters. These can be selected with the mouse to execute or escape from the command. The mouse can also be used to identify a parameter to be edited.

Editing parameter values and controlling the execution of the command can be achieved with the following keys:

- \uparrow , \downarrow and **<Tab>** can be used to move between the parameters.
- \leftarrow and \rightarrow move the text cursor within the value being edited.
- **<Enter>**, **<Return>** or → move to next parameter. If **Accept** is highlighted or there is only one parameter, the command is executed.
- **<Esc>** escapes from the command.
- <Back-space> or <Delete> delete characters.

To toggle insert mode: **<Insert>** (Windows) or function keys **<F2>** or **<PF2>** (X-windows)

- To move to start of the value <Home> (Windows) or function keys
 <F3> or <PF3> (X-windows)
- To move to end of the value: <End> (Windows) or function keys <F4> or <PF4> (X-windows)

FileBoxes and CDBoxes

FileBoxes (Figure 4.2) are used for selecting a file name for reading or writing. The FileBox contains a filter string which specifies a subset of all files. If the filter is edited, it can be acted on by typing **<Enter>**,

<Return> or → or by selecting the Filter button.

If there are more files than can be displayed in the FileBox a scroll-bar is displayed at the left side of the list of names. Similarly, if the longest file name is too long for the FileBox a scroll bar is displayed below the list of names. The size of the slider within the scroll bar indicates the proportion of the text which is displayed. The text can be scrolled in two ways:

• clicking above or below the slider in the vertical scroll bar, or to the left or right of the slider in the horizontal scroll bar, scrolls the text by one page in the direction indicated.



Figure 4.2 A File Selection Box

• dragging the slider, by pressing and holding the left mouse button while moving the mouse scrolls the text in the direction indicated while the mouse is moving.

One file should be selected from the list of files. Double-clicking (selecting the file twice in quick succession) confirms the selection. Alternatively, the

selection can be confirmed by selecting the **Accept** button. The required file name can also be typed into the selection box and accepted by typing **<Enter>**, **<Return>** or \dashv .

The current directory or folder name is shown and its sub-directories are also displayed in a second selection area. The current directory can be changed by using a double-click selection in the same way as for a file name, or by typing the directory name into the selection box and typing **<Enter>**, **<Return>** or \dashv . Any change of directory in a FileBox is remembered for the next time the FileBox is used, unless the current directory is changed using the CDBox which resets the directories for all FileBoxes.

The CDBox implements the Change Directory command within the GUI. It displays a list of subdirectories, which can be selected by double-click or typing in the same way as within the FileBox. If the new directory includes a disk or device name, it can only be selected by typing the full name into the selection box. When the current directory is as required, the CDBox can

be closed by typing **<Esc>** or by selecting the **Quit** button.

In FileBoxes and CDBoxes, file tree-names and directory names can be given using environment variables (UNIX and Windows only). Environ-

ment variables **\$VFDIR** (on UNIX systems) and **%VFDIR%** (on Windows systems) are defined by the software as the parent directory of the software.

DialogBoxes

DialogBoxes are used to input information using a combination of keyboard and mouse operations. Within a DialogBox there can be:

• Text-inputs: black rectangles. Initially the first text-input is selected and any characters typed will appear there. Any text-input can be selected with the mouse before typing. The **<Enter>**, **<Return>**, ⊣, ↓ or **<Tab>** keys can be used to move to the next text-input. The ↑ key can be used to move to the previous text input. Within a text-input the value can be edited using the editing and function keys defined for Parameter-

Boxes (see page 4-4). Selecting a down arrow button \checkmark to the right of a text input activates a FileBox to supply a file name for the text-input (see page 4-5).

• Switches: small squares or 'radio-buttons'. The switches can be on

(red) or off (blue). The state of a switch can be changed by selecting with the mouse pointer. Turning on one switch might turn others off if the options are mutually exclusive.

- Buttons: labelled rectangles. These are used to action the selected options, or exit without issuing a command. A button can be selected using the mouse or, if it is high-lighted, with the <Enter>, <Return> or ↓ key. The <Esc> can also be used to exit without issuing a command.
- Scrolling lists: a list of items from which one or many can be selected using the mouse. If the list is long or wide, scroll-bars can be used to view other parts of the list in the same was as for FileBoxes (page 4-5).

It is important to remember that DialogBoxes often contain more items than are needed at a particular time. Only those items required should be selected.

MessageBoxes

MessageBoxes are used by the GUI to communicate important information to the user. There are 5 types of MessageBox each containing black text on a grey background.

• Information: the results of commands, warnings etc. These boxes are labelled with a large i. Information MessageBoxes can be dismissed by

typing any key on the keyboard (except **<F1>**) or with the left mouse button.

- If the quantity of information exceeds the size of the window, a scrollbar is displayed to enable the whole message to be viewed. The window will show the top of the message as the scroll is generated, but other parts of the message can be viewed by dragging the scroll bar up or down.
- Errors: these include Pick operations outside the model space and bad values in ParameterBoxes. The program gives the user another chance to perform the input if an error occurs. Error MessageBoxes are labelled with a large !. They can be dismissed by typing any key on the keyboard (except <F1>) or with the left mouse button.
- Questions: these always require a choice between **YES** and **NO**. QuestionBoxes are labelled with a large ?. They can be dismissed by

selecting either the **YES** box or the **NO** box.

- Input: these always require additional information to be given by the user via a ParameterBox, a DialogBox or a FileBox which appears below the MessageBox. The boxes disappear when the information has been supplied.
- Timer: these indicate how much of an operation has been completed. Timers are only displayed for operations for which the estimated elapsed time is greater than 5 seconds. Timer boxes cannot be dismissed, but will automatically disappear when the operation is complete.

ColourBoxes

The ColourBox (Figure 4.3) is used to redefine colours used on the display. It consist of 3 horizontal slider bars, one each for red, green and blue, a square showing the colour as it is changed and three buttons,

Accept, Cancel and Quit.

The colour can be adjusted by moving the sliders in two ways:

- clicking to the left or right of the slider decreases or increases the amount of a colour by 10%.
- dragging a slider by pressing and holding the left mouse button and moving the mouse to the left or right decreases or increases the amount of a colour while the mouse is moving in proportion to the distance between the cursor and the slider.



Figure 4.3 A ColourBox

The buttons Accept a colour by changing the display to use the colour in the colour square; Cancel the changes by restoring the colour to what it was when the ColourBox was opened; and Quit the ColourBox.

Note that on some types of display, the new colours are not shown until the picture is redrawn.

Hiding and Leaving the Menus

Sometimes it is necessary to hide the menus so that the complete picture can be seen. This can be done using the **<F1>** function key. bring the menus back again.

To leave the menus completely the top-level menu item **MENU-OFF** should be selected. To return to menu-mode, the keyboard command ^ should be given. (This is the caret character followed by **<Enter>**, **<Return>** or \dashv .)

Pre Processor Quick Reference Guide

The following is a complete list of the 'top-level' keyboard commands (which can be entered in response to the prompt 'OPERA >') and menu items. Following sections contain complete descriptions of all the keyboard commands, sub-commands and modes of the OPERA-3d pre processor. The commands are described in alphabetical order. The modes of the DEFINE command are described in the order they occur when using the program.

Commands for Keyboard Entry – Command Line

Command line entry is carried out in the text window.

• Help Command:

HELP	Obtain System overview, help on command interpreter,
	Euler angles, units and new features in this version.

•	Mesh	definition	and	editing	commands:
---	------	------------	-----	---------	-----------

	8
DEFINE	Enter the define sequence to input points, facets, subdi-
	visions, extrusions, materials and boundary conditions
	of the finite element mesh. DEFINE can also be used to
	define geometry of 20 node 'brick' conductor elements.
REDEFINE	Remove all extrusions and enter DEFINE sequence to
	modify base-plane and redefine extrusions.
EXTEND	Add more extrusion layers to a mesh.
MODIFY	Modify mesh data: points, subdivisions, materials and
	boundary conditions.
LABEL	Add extra labels to points, lines, facets and volumes.
MATERIALS	Define material properties.
TRANSFORM	Transform a labelled set of points
SLIP	Add or remove a slip surface for CARMEN models.
CHECK	Examine the shape of the finite element volumes, add-
	ing label DEBUG. Find the exterior facets of the finite
	element volumes, adding label EXTERNAL. Report
	whether hexahedral meshing is possible.

• Conductor definition and editing command:

CONDUCTOR	Enter conductor sub-command mode to define, modify
	and list conductor data.

• Finite element mesh commands:

MESH	Generate a mesh on the surfaces of the volumes.
FILL	Generate a mesh inside each volume.

• Display command:

DISPLAY	Display some or all of the mesh and conductors in 3D with optional hidden surface removal.
THREED	Start or update the 3D Viewer

• Material B-H characteristic, definition and editing command:

BHDATA	Enter the BH of	data definition	and editing mode.
--------	-----------------	-----------------	-------------------

•	Commands to	read and write data files:
R	EAD	Read a file of OPERA-3d pre

READ	Read a file of OPERA-3d pre processor data.
EDIT	Read a file of OPERA-3d pre processor data allowing data to be edited at break points.
WRITE	Write a file of OPERA-3d pre processor data for use with READ and EDIT commands.
TABLE	Write a table file containing all the nodes of the model.
SOLVERS	Prepare an analysis database.
IDEAS	Read a finite element mesh from an I–DEAS universal file.

• Program management commands:

DEVICE	Reset or change graphics device.
COLOUR	Enquire and set colours for the display.
TITLE	Control screen titles.
CLEAR	Clear program data and re-initialize all commands.
DUMP	Write a picture file containing the current display.
END	End OPERA-3d pre processor.

Menu System – GUI

The following is an overview of the first level menu structure. A brief description of the functionality of each item is also given and the page number of the section of this chapter which describes it in detail.

$file \downarrow$

File Options		
Read pre processor file	read a pre processor command file	page 4-146
Write pre processor file	write a pre processor command file	page 4-177
I-DEAS universal file	read a finite element mesh from an I–DEAS universal file	page 4-117
Analysis	Create/edit an analysis database	page 4-150
create new database	Create an analysis database	
use existing database	Edit an analysis database	
Start analysis now	Start an analysis program using OPERAANL	page 2-31
Write node table file	Write a file in table format containing all the node coordinates	page 4-168
Commands in	read a command input file	page 2-25
Dump picture	send current graphics display to a file	page 4-103
Change directory	change the current directory	page 2-30
System command	run an operating system command	page 2-30
Return	close menu	
End OPERA-3d/Pre	end pre processing session	page 4-111

 $options\downarrow$

Options		
Calculator		
Parameters	set a user defined parameter	page 2-21
Constants	set a user defined constant	page 2-21
List variables	list user defined variables	page 2-21
Colour settings	modify colours in graphics palette	page 4-22
Clear and Reset	clear all mesh data and restart	page 4-21
Dump picture	send current graphics display to a file	page 4-103
Graphics output	select type of graphics output	page 4-94
Title options	specify title options	page 4-174
Title position	specify title position in window	page 4-174
Display title	enter text of title	page 4-174
Return	close menu	

$\mathtt{display} {\downarrow}$

3D Viewer		
style	specify wire frame, solid etc. display of model	page 4-171
select parts	specify components to be displayed	page 4-170
refresh display	start or update 3D Viewer	page 4-169
Display Command		
view	specify display size and orientation of model	page 4-98
style	specify wire frame, solid etc. display of model	page 4-101
select parts	specify components to be displayed	page 4-99
refresh display	clear and re-draw graphics window	page 4-96
copy 3D view	copy view parameters from 3D Viewer	page 4-99
Return	close menu	
	—	

help \downarrow

Нејр		
System Overview	general description of pre processor	page 4-116
The GUI	assistance with menu system	
Command line	assistance with command line entry	
Return	close menu	

 $\texttt{mesh} \downarrow$

Mesh Generator		
Surface mesh		
options	set options for surface mesh generator	page 4-131
triangles	generate a surface mesh of triangles	page 4-131
quadrilaterals	generate a surface mesh of quadrilaterals	page 4-131
Volume mesh		
options	set volume mesh generator options	page 4-114
Mesh	generate a volume mesh	page 4-114
Return	close menu	

$\texttt{MODIFY} \downarrow$

Data Modification		
Mesh number	specify mesh to be modified	page 4-133
No 3D Viewer	choose whether to use 3D Viewer	page 4-133
Point coordinates	modify point coordinates	page 4-133
Subdivisions	modify subdivisions	page 4-133
Material properties	modify element/potential/material types	page 4-133
Boundary conditions	modify boundary conditions	page 4-133
Add slip surface	add slip surface to CARMEN models	page 4-149
Remove slip surface	remove slip surface	page 4-149
Labels	modify entity labels	page 4-123
Transform labelled points	apply transformation to a group of points	page 4-176
Conductors	modify conductor definitions	page 4-24
Return	close menu	

$\texttt{define} \downarrow$

Data Definition		
Define new mesh	start creating a new mesh	page 4-49
Extend existing mesh	continue extending an existing mesh	page 4-112
Redefine a mesh	remove extrusions, modify baseplane and re- extrude	page 4-148
Check a mesh	check existing mesh geometry	page 4-18
Conductors	create a conductor	page 4-24
Read conductor data	read a file of conductor definition commands	page 4-146
Material properties	define material properties	page 4-127
BH tables	define/edit a B–H curve	page 4-15
Return	close menu	

The **BHDATA** Command

Menu Route

DEFINE \downarrow BH tables

Command Line Parameters

Command	BHDAT	A	
Parameter	Default	Functi	on
UNITS	CGS	The units of B and H	
		CGS	gauss and oersted
		SI	tesla and ampere metre ⁻¹

The TOSCA, ELEKTRA and CARMEN programs use material characteristics to relate flux density and field intensity of all materials (except for AIR and NULL). For soft magnetic materials these should be defined in the first quadrant, with the first values of B and H both zero. The curve should not extend beyond saturation magnetisation; the program extrapolates correctly. True hysteresis cannot be modelled, but a limited form, particularly applicable to ferrites, is available in ELEKTRA-SS using complex permeability in the MATERIALS command (page 4-127) or the MATERIAL subcommand of SOLVERS (page 4-161).

Anisotropic materials can be treated as a stack of laminations or by use of multiple BH curves. For hard magnetic materials the demagnetisation curve in the second and third quadrant is used. The orientation of laminations, anisotropic materials and permanent magnets is set by the VECTOR property, set in Materials Definition Mode of the DEFINE command (page 4-49). The PACKING factor of laminations is set there also.

In electrostatics and current flow problems, similar curves can be used to relate the electric field and displacement current or electric field and current density.

The BHDATA command is used to create, edit or check tables of pairs of values that define the non-linear BH, DE or JE characteristics of magnetic, dielectric or conducting materials. There must be at least 5 and not more than 50 entries in each table. Material characteristics are not stored with the models in pre processor data files. After a table has been defined or edited the new data should be stored in a materials library file. A separate system directory could be created to contain all such files. Such a directory is supplied with the software and can be accessed using the directory name

\$VFDIR/bh (UNIX) or *%VFDIR%\bh* (Windows) in the FileBox Selected directory input box.

Names of BH files are requested by the MATERIALS sub-command of the SOLVER command for all material names used (page 4-161).

The BHDATA command has one parameter, UNITS, which allows the user to define the data in either CGS units (gauss and oersted) or SI units (tesla and ampere metre⁻¹). Files of BH data stored in one set of units can be loaded and edited in either set of units.

On entering the BHDATA command, the user is presented with a default BH curve. In general it will be necessary to delete this curve and replace it with the desired characteristic. In order to use the default curve it must be also stored in a file.

The BHDATA command has 8 sub-commands allowing addition of points on the current curve, modification of existing points, access to files, and data checking. The commands do not have named parameters but rely on keywords and values being supplied in the correct order. In the description below the keywords are given in upper case and the variable values in lower case. Keywords can be abbreviated to single letters. In any case only the first 4 characters are decoded.

In the REPLACE sub-command, the system variables B and H, the values of the current point, can be used in expressions. For example, an existing curve can be degraded in a command loop such as:

OP-BH > \$ do #i 1 10 OP-BH > repl #i h+(b-h)*0.9 h -redr OP-BH > \$ end do

The boolean **-REDRaw** prevents the graph of B against H being re-drawn each time round the loop.

BHDATA Sub-commands

The sub-commands of the BHDATA command are described in the following table:

Sub-commands	Function
ADD B H	Add a new point to the end of the table. B and
	H are numeric values of B and H .
CHECK M N	Check the data and display the interpolations
	of the data used in analysis. M and N specify
	the first and last point displayed.

Sub-commands	Function (continued)
DELETE M N	Deletes the points M to N of the curve. N can have the value * to indicate the last point.
INSERT NUMBER B H	Inserts a new point after the point NUMBER of the curve. B and H are numeric values of B and H .
LOAD FILE	Loads a curve from a file. The file name extension <i>.bh</i> is added to the name if no extension is given. Any points already defined are deleted.
QUIT	Leave the BHDATA command.
REPLACE N B H	Replaces the Nth point of the curve. B and H are the new values of B and H. \pm REDRAW can be used to control whether the curve is redisplayed.
STORE FILE	Stores the curve in a file. The file name extension <i>.bh</i> is added to the name if no extension is given.

The CHECK Command

Menu Route

 $\text{DEFINE} \downarrow$ Check a mesh

Command Line Parameters

Command	CHECK		
Parameter	Default	Function	
MESH	1	Number of the finite element mesh to be checked.	
VERBOSE	NO	Verbose reporting of errors:	
		NO	Summary of errors.
		YES	Detail of error given for each volume.

The CHECK command examines the shape of the volumes of the MESH number given, adding the label DEBUG to any which may cause errors in the mesh generator or analysis programs. It also gives the label EXTER-NAL to all facets which are not shared by two volumes. There is no checking between facets of different meshes.

The type of checking depends on whether the model is capable of being meshed with hexahedra or can only be meshed with tetrahedra. Hexahedral meshes can be generated in models which obey the following rules:

- All volume facets must be triangles or quadrilaterals.
- The subdivisions on opposite lines of quadrilateral facets must be the same.
- The subdivisions of two lines of triangular facets must be the same.

The CHECK command first examines the model to see whether it can be meshed with hexahedra. It then checks each volume as described below.

In verbose mode, the unique labels assigned to volumes have the form VNNN, where NNN is the internal volume number. This allows individual volumes to be displayed.

If the CHECK command is used again, after correcting the volumes in error, the label DEBUG is removed from any volumes previously in error and the unique labels, VNNN are also removed.

• Example: to display volumes with label DEBUG in layer 3:

OPERA > check OPERA > disp type=volu labe=debug l1=3 3

Checks on Hexahedral Models

If the model can be meshed with hexahedra, the following checks are performed.

The program calculates the determinant of the Jacobian which defines the isoparametric mapping from a hexahedron to a unit cube. For the mapping to be unique this must be greater than zero everywhere within the hexahedron. The Jacobian determinant is evaluated at every point defining the volume. In some circumstances the checks may be more strict than necessary. For example, a quadratic volume is checked using quadratic shape functions, but in analysis, the volume may be discretised into linear elements in which the errors do not show up.

If the Jacobian determinant changes sign within a volume, this may be due to any of the errors listed below:

- faces which cut through each other.
- mid-side points which are too close to the corners (closer than 0.25 times length of side).
- excessive curvature or distortion.

The Jacobian can also be used to evaluate the size of each volume. If the Jacobian goes to zero or is negative throughout a volume, this is also reported.

The number of volumes in error is reported for each layer and since they have the label DEBUG they can be displayed separately using the DIS-PLAY command.

If **+VERBOSE** is selected, the CHECK command reports an error message and assigns a unique label to each DEBUG volume. The errors reported are:

- Jacobian changes sign.
- Very small volume.
- Negative volume.
- Mid-side node too close to corner.

The CHECK command also counts the total numbers of nodes and elements in the mesh. These numbers are approximate if degenerate volumes (with facets with less than 4 sides) or NULL materials are used.

Checks on other Models

For models with facets with more than 4 sides or irregular subdivision, the following checks are performed.

The equation of the plane of each facet with more than 4 sides is calculated. An error is reported if any points defining the facet do not lie in the plane.

The **CLEAR** Command

Menu Route

 $options \downarrow$ Clear and Reset

Command Line Parameters

Command CLEAR No Parameters

The CLEAR command puts OPERA-3d back to the state it was in when it first started. It deletes all the data including conductors and construction lines, re-initializes all variables and sets all parameters back to their default values.

User variables are not deleted.

The **COLOUR** Command

Menu Route

OPTIONS↓ Colour settings

Command Line Parameters

Command			
Commanu	COLOUR	-	
Parameter	Default	Function	
OPTION	ENQUIRE	Option: CONFIGURE, ENQUIRE or SET.	
		CONF	Re-configure the colour map for a different number of distinct MATE- RIALS.
		ENQU	Enquire which colour numbers are used for each material and part of the display.
		SET	Reset colour number CODE to new values of RED, GREEN and BLUE.
CODE	none	Colour map number to be redefined.	
RED	none	Amount of red for colour CODE.	
GREEN	none	Amount of green for colour CODE.	
BLUE	none	Amount of blue for colour CODE.	
MATERIALS	5	Number of distinct material colours.	

The COLOUR command informs the user which colours are used for each part of the display and enables the definitions of the colours to be changed.

There are three OPTIONS: CONFIGURE, ENQUIRE and SET.

- **CONFIGURE** sets the number of distinct material colours to be made available. The operation of this option depends on the total number of colours available on the display being used. A minimum of 5 and a maximum of 100 distinct material colours are allowed. If there are more materials than colours, the colours are re-used cyclically for higher material numbers. Increasing the number of material colours reduces the number of light-source shading levels.
- ENQUIRE lists the colour numbers used for each part of the display.
- SET displays the definition of the colour number given by CODE in terms of its red, green and blue components. The colour can be changed by giving values to RED, GREEN and BLUE which should each be in the range 0 to 1. The current values of RED, GREEN and BLUE can be used in expressions to define the new values.

- If the colour CODE to be reset is the main colour for air, conductors or a material, etc., the corresponding light-source shading colours are reset to different shades of the new colour.
- On some types of display, the new colours will not be shown until the picture is refreshed.

The **CONDUCTOR** Command

Menu Route

DEFINE↓ Conductors

Command Line Parameters

Command	CONDUCTOR	
No Parameters		

This command enters the conductor sub-command mode of the pre processor. In this mode new conductors may be defined, existing conductors edited or erased, the parameters of the current set of conductors may be listed or they may be stored in a *cond* data file. The sub-commands are:

CONDUCTOR Sub-commands			
Sub-command	Function		
DEFINE	Define a new conductor		
ERASE	Erase one or several conductors		
EXTERNAL	Define uniform external field		
MODIFY	Modify one or several conductors		
PRINT	Print details of one or several conductors to the terminal		
	and the log file.		
QUIT	Leave the CONDUCTOR command.		
WRITE	Write a data file with commands to define all the conduc-		
	tors.		

The programs have a wide range of pre-defined conductor geometries, ranging from simple solenoids to bedsteads and racetracks wound on the surface of cylinders. There are also primitive conductor elements which can be joined together to build up conductor circuits.

The following conductors are available:

Conductor Shapes		
SOLENOID	Solenoid around Y' axis.	
GSOLENOID	Generally orientated solenoid.	
RACETRACK	Racetrack around <i>Y</i> ' axis.	
GRACETRACK	Generally orientated racetrack.	
BEDSTEAD	Bedstead around <i>Y</i> ' axis.	
GBEDSTEAD	Generally orientated bedstead.	
HELIX	Helical end racetrack.	

Conductor Shapes (continued)		
CPEND Constant perimeter end racetr		
ARC or GARC	Circular arc element.	
STRAIGHT or GSTRAIGHT	Straight bar element.	
BR8	8 noded brick element.	
BR20	20 noded brick element.	

To enable conductors to be oriented in space correctly, local coordinate systems can be defined. To reduce the amount of data necessary, symmetry and reflection codes can be used to replicate a basic shape.

Parameters common to all conductors including the local coordinate systems and replication parameters will be described first, before details of the conductor shapes are given and lastly the parameters of the sub-commands are listed.

Current Density, Drive Label and Tolerance

The current flowing in the conductors is defined by the current density (CURD). For conductors with changing cross section the current density applies to the first face of the conductor. The system variable AREA holds the correct value of area so that the current density can be given in terms of the total current as current/AREA.

The PHASE parameter is used to specify a drive label. The SOLVERS command (page 4-150) allows each value of drive label to be associated with a scaling factor (statics), phase angle (steady-state ac) or a drive function (transient).

The fields from some conductors are calculated using an adaptive integration method which requires the user to supply a TOLERANCE. This specifies the error tolerance on the flux density in the units system being used. The field from conductors without a TOLERANCE parameter, or with TOLERANCE set equal to zero is calculated to a tolerance of 10 gauss.

Single FilamentA negativeConductorsapproximais then use

A negative value of **TOLERANCE** can be used to request a single filament approximation to the conductors. The absolute value of the **TOLERANCE** is then used to control numerical integration along the trajectory of the current filament (fields are evaluated just by a line integral with no integration over the cross-section).

Local Coordinate System 1

The local coordinate system 1 is formed by displacing the origin with respect to the global origin to coordinates (XCENTRE, YCENTRE, ZCENTRE) and rotating by Euler angles (PHI1, THETA1, PSI1) (see "Euler Angles" on page 2-34 and Figure 4.4). ANGLE is an alias for PHI1.



Figure 4.4 Local Coordinate System 1

Local Coordinate System 2

The local coordinate system 2 is formed by displacing the origin with respect to local coordinate system 1 by (X0, Y0, Z0), and rotating by Euler angles (T, P, S) (see "Euler Angles" on page 2-34 and Figure 4.5).

Symmetries and Reflections

SOLENOIDS, RACETRACKS and BEDSTEADS can be reflected in the global ZX plane, with the currents in the reflection flowing in the same sense around the local Y' axis (SYMM=2 or -2). Any other values of SYMMETRY give no reflection.

All other conductors can have more general symmetries. **SYMMETRY=N** gives N copies of the basic shape each rotated by 360/n degrees about the global Z axis with respect to the previous copy. If N is negative then the sign of the current alternates from one copy to the next.

The other conductors can also have reflections in local(1) coordinate planes. The parameters IRXY, IRYZ and IRZX can have values 0, 1 or -1



Figure 4.5 Local Coordinate System 2

giving no reflection, reflection with the same sign of current or reflection with reverse sign of current.

Solenoids

Two types of solenoid are available (Figure 4.6). The first (SOLENOID) has a restricted set of symmetries and can only be orientated with local coordinate system 1 without the Euler angles (PHI1, THETA1, PSI1). The second (GSOLENOID) can be generally orientated and replicated with any of the symmetries and reflections.



Figure 4.6 A Solenoid

The quadrilateral cross section of the solenoids in the local XY plane is defined by the coordinates of the 4 corners. (X1, Y1, ..., X4, Y4) and the curvatures of the sides (CU1, ..., CU4). If the 4 corners are entered in a clockwise sense when viewed from the positive Z axis, then positive curvature gives an increase in the area of the cross section.

Positive currents flow in the positive Z direction across the positive X half of the XY plane, assuming that the vertices of the cross section have been defined in a clock-wise sense; otherwise the direction of current flow is reversed.

Parameters for SOLENOID XCENTRE, YCENTRE, ZCENTRE Local coordinate system 1 X1, Y1, X2, Y2 Solenoid cross section in X'Y' plane X3, Y3, X4, Y4 Solenoid cross section in X'Y' plane CU1, CU2, CU3, CU4 Solenoid cross section curvatures CURD, SYMMETRY, PHASE Current density, symmetry and drive label

GSOLENOID	
XCENTRE, YCENTRE, ZCENTRE PHI1, THETA1, PSI1	, Local coordinate system 1
X0, Y0, Z0	Local coordinate system 2 (ori- gin)
T, P, S	Local coordinate system 2 (Euler angles)
X1, Y1, X2, Y2	Solenoid cross section in X'Y' plane
X3, Y3, X4, Y4	Solenoid cross section in X'Y' plane
CU1, CU2, CU3, CU4	Solenoid cross section curvatures
CURD, SYMMETRY, PHASE	Current density, symmetry and drive label
IRXY, IRYZ,IRZX	Reflections in local coordinate system 1 coordinate planes
TOLE	Field tolerance

Solenoids

Parameters for General **Solenoids**

Racetracks



Two types of racetrack are available (Figure 4.7). The first (RACE-

Figure 4.7 A Racetrack

TRACK) has the restricted set of symmetries and can only be orientated with local coordinate system 1 without the Euler angles (PHI1, THETA1, PSI1). The second (GRACETRACK) can be generally orientated and replicated with any of the symmetries and reflections.

The racetrack is made up of 4 straight sections and four 90 degree arcs. The cross section is rectangular, defined by its width in local X direction (A) and thickness in local Y direction (B). The coordinates of the bottom inside edge as it crosses the XY plane are given by X1 and Y1. The half-length of the z-directed straight is H1 and the inside radius of the corners is R1. R1 must be greater than zero.

Positive currents flow in the positive Z direction across the positive X half of the XY plane (local coordinates).

RACETRACK	
XCENTRE, YCENTRE, ZCENTRE	Local coordinate system 1
А, В	Conductor cross section
X1, Y1	Local coordinates of bottom
	inside corner

Parameters for Racetracks

RACETRACK (continued)	
H1, R1	Half length and corner radius
CURD, SYMMETRY, PHASE	Current density, symmetry and drive label

Parameters fo	r
General	
Racetracks	

GRACETRACK	
XCENTRE, YCENTRE, ZCENTRE,	Local coordinate system 1
PHI1, THETA1, PSI1	
X0, Y0, Z0	Local coordinate system 2 (ori-
	gin)
T, P, S	Local coordinate system 2 (Euler
	angles)
А, В	Conductor cross section
X1, Y1	Local coordinates of bottom
	inside corner
H1, R1	Half length and corner radius
CURD, SYMMETRY, PHASE	Current density, symmetry and
	drive label
IRXY, IRYZ,IRZX	Reflections in local coordinate
	system 1 coordinate planes
TOLERANCE	Field tolerance

Bedsteads

Two types of bedstead are available (Figure 4.8). The first (BEDSTEAD) has the restricted set of symmetries and can only be orientated with local coordinate system 1 without the Euler angles (PHI1, THETA1, PSI1). The second (GBEDSTEAD) can be generally orientated and replicated with any of the symmetries and reflections.

The bedstead is made up of 8 straight sections and eight 90 degree arcs. The cross section is rectangular, defined by its width in local X direction (A) and thickness in local Y direction (B). The coordinates of the bottom inside edge as it crosses the XY plane are given by X1 and Y1. The half-length of the z-directed straight is H1 and the inside radius of the corners at the ends of the z-directed straights is R1. The length of the straights in the local y direction is H2 and the inside radius of the arcs connecting these straights to the straight bars which cross the YZ plane is R2. R1 and R2 must be greater than zero.

Positive currents flow in the positive Z direction across the positive X half of the XY plane (local coordinates).



Figure 4.8 A Bedstead

Parameters for Bedsteads		
	BEDSTEAD	
	XCENTRE, YCENTRE, ZCENTRE	Local coordinate system 1
	A, B	Conductor cross section
	X1, Y1	Local coordinates of bottom inside corner
	H1, H2	Lengths of straight sections
	R1, R2	Inside radii
	CURD, SYMMETRY, PHASE	Current density, symmetry and drive label

Parameters for General Bedsteads

GBEDSTEAD	
XCENTRE, YCENTRE, ZCENTRE,	Local coordinate system 1
PHI1, THETA1, PSI1	
X0, Y0, Z0	Local coordinate system 2 (ori-
	gin)
T, P, S	Local coordinate system 2 (Euler
	angles)
A, B	Conductor cross section
X1, Y1	Local coordinates of bottom
	inside corner
H1, H2	Lengths of straight sections

GBEDSTEAD (continued)	
R1, R2	Inside radii
CURD, SYMMETRY, PHASE	Current density, symmetry and drive label
IRXY, IRYZ,IRZX	Reflections in local coordinate system 1 coordinate planes
TOLERANCE	Field tolerance

Helical Ends

The helical end (HELIX, Figure 4.9) is the first of two conductors which



Figure 4.9 A Helical End

are wound on the surface of a cylinder. It is made up of two straight sections parallel to the axis of the cylinder (the local Z axis). The half length of the central filament of the straights is H1. The cross section is rectangular, defined by the thickness in the radial direction (A) and width in the azimuthal direction (B). The azimuthal position of the straights on the cylinder is defined by the angle from the mid i.e ZX, plane (ALPHA).

Each end of the conductor consists of two helices which extend to azimuthal angle BETA measured from the mid-plane and Z coordinate H2. The width can change along the helices so that the arc joining the helices together has a width R2. R2 is normally set the same as B.

Positive currents flow in the positive Z direction across the positive X half of the XY plane (local coordinates).

Parameters for Helical Ends

HELIX	
XCENTRE, YCENTRE, ZCENTRE,	Local coordinate system 1
PHI1, THETA1, PSI1	
X0, Y0, Z0	Local coordinate system 2 (ori-
	gin)
T, P, S	Local coordinate system 2 (Euler
	angles)
А, В	Conductor cross section
H1, H2	Half lengths of conductor
R1, R2	Radius of forming cylinder;
	thickness of cross-over arc
ALPHA, BETA	Angles to central filament of
	straight and end of helix
CURD, SYMMETRY, PHASE	Current density, symmetry and
	drive label
IRXY, IRYZ,IRZX	Reflections in local coordinate
	system 1 coordinate planes
TOLERANCE	Field tolerance

Constant Perimeter Ends

The constant perimeter end (CPEND, Figures 4.10 and 4.11) is also wound



Figure 4.10 A Constant Perimeter End

on the surface of a cylinder. It is made up of two straight sections parallel to the axis of the cylinder (the local Z axis). The half length of the central



Figure 4.11 Another view of a Constant Perimeter End

filament of the straights is H1. The cross section is rectangular, defined by the thickness in the radial direction (A) and width in the azimuthal direction (B). The surface which touches the cylinder can be flat and TANGENTIAL to the cylinder or curved and FITTING the cylinder. (N.B. The FITTING conductors are more accurate in geometry and field than the TANGEN-TIAL.) The azimuthal position of the straights on the cylinder is defined by the angle from the mid. i.e ZX, plane (ALPHA).

The ends of the conductor form a smooth curve over the cylinder. In manufacture they are machined by a cutter which has an angle **BETA** to the local Z axis of the cylinder and traverses a circular path on a cylinder of radius **R2**.

Positive currents flow in the positive Z direction across the positive X half of the XY plane (local coordinates).

CPEND	
XCENTRE, YCENTRE, ZCENTRE, PHI1, THETA1, PSI1	Local coordinate system 1
X0, Y0, Z0	Local coordinate system 2 (ori- gin)
T, P, S	Local coordinate system 2 (Euler angles)
А, В	Conductor cross section
ALPHA, BETA	Angles to central filament of straight and of cutting tool

Parameters for Constant Perimeter Ends

CPEND (continued)	
H1	Half length of straight
R1, R2	Radii of construction cylinders
FIT	Conductor fit to mandrel
CURD, SYMMETRY, PHASE	Current density, symmetry and drive label
IRXY, IRYZ,IRZX	Reflections in local coordinate system 1 coordinate planes
TOLERANCE	Field tolerance

Straight Bars

The STRAIGHT bar (or GSTRAIGHT), Figure 4.12 is a simple rectangu-



Figure 4.12 A Straight Bar

lar cross section conductor. Its cross section is A in the local X direction and B in the local Y direction. Its central filament starts at the local origin and extends H1 in the local Z direction. Positive currents flow in the positive local Z direction.

Parameters for Straight Bars	STRAIGHT or GSTRAIGHT	
	XCENTRE, YCENTRE, ZCENTRE, PHI1, THETA1, PSI1	Local coordinate system 1
	X0, Y0, Z0	Local coordinate system 2 (ori- gin)

STRAIGHT or GSTRAIGHT (continued)	
T, P, S	Local coordinate system 2 (Euler angles)
А, В	Conductor cross section
H1	Length of straight
CURD, SYMMETRY, PHASE	Current density, symmetry and drive label
IRXY, IRYZ,IRZX	Reflections in local coordinate system 1 coordinate planes
TOLERANCE	Field tolerance (GSTRAIGHT only)

Circular Arcs

The circular ARC (or GARC), Figure 4.13 is similar to the straight bar. Its



Figure 4.13 An Arc

cross section is A in the local X direction and B in the local Y direction. Its central filament starts at the local origin and moves in the local Z direction. It bends through an angle PHI towards the positive local Y direction. Positive currents flow from the starting coordinates in the positive local Z direction.

Parameters for Circular Arcs

ARC or GARC	
XCENTRE, YCENTRE, ZCENTRE,	Local coordinate system 1
PHI1, THETA1, PSI1	
X0, Y0, Z0	Local coordinate system 2 (ori-
	gin)
T, P, S	Local coordinate system 2 (Euler
	angles)
А, В	Conductor cross section
R1, PHI	Inside radius and angle
CURD, SYMMETRY, PHASE	Current density, symmetry and
	drive label
IRXY, IRYZ, IRZX	Reflections in local coordinate
	system 1 coordinate planes
TOLERANCE	Field tolerance (GARC only)

Bricks

Two brick elements can be used to defined more complex shapes, especially those which involve changes in cross section. They have 8 nodes (BR8) or 20 nodes (BR20, Figure 4.14).



Figure 4.14 A 20-Node Brick

The bricks are defined by the coordinates of the nodes (X1, Y1, Z1, ..., X20, Y20, Z20. The current in a brick flows from face 1 (nodes 1, 2, 3, 4) to face 2 (nodes 5, 6, 7, 8). The total current in the brick is calculated from

node 9	between nodes 1 and 2
node 10	between nodes 2 and 3
node 11	between nodes 3 and 4
node 12	between nodes 4 and 1
node 13	between nodes 1 and 5
node 14	between nodes 2 and 6
node 15	between nodes 3 and 7
node 16	between nodes 4 and 8
node 17	between nodes 5 and 6
node 18	between nodes 6 and 7
node 19	between nodes 7 and 8
node 20	between nodes 8 and 5

the current density (CURD) multiplied by the area of face 1. The 20 noded brick has mid-edge nodes:

If the mid-edge nodes are not co-linear with the corner nodes, then the surfaces are parabolic. If the mid-edge nodes are not at the centre of the edges, the current density will vary over the cross section. The mid-edge nodes should not be displaced outside the ¹/₄ and ³/₄ points.

The top-level DEFINE command of the pre processor (page 4-49) can be used to define the geometry of brick conductors. This allows triangular or quadratic cross sections, which may be curved, to be extruded in the third direction. When the extrusions are complete the geometry is copied into the conductor storage area so that other conductor parameters, local coordinate systems, replications, current density etc. can be set to their correct values with the MODIFY sub-command.

BR8	
XCENTRE, YCENTRE, ZCENTRE, PHI1, THETA1, PSI1	Local coordinate system 1
X0, Y0, Z0	Local coordinate system 2 (ori- gin)
T, P, S	Local coordinate system 2 (Euler angles)
X1, Y1, Z1	Bottom right corner of front face
X2, Y2, Z2	Top right corner of front face
X3, Y3, Z3	Top left corner of front face
X4, Y4, Z4	Bottom left corner of front face
X5, Y5, Z5	Bottom right corner of back face
X6, Y6, Z6	Top right corner of back face

Parameters for 8-Noded Bricks
BR8 (continued)	
X7, Y7, Z7	Top left corner of back face
X8, Y8, Z8	Bottom left corner of back face
CURD, SYMMETRY, PHASE	Current density, symmetry and drive label
IRXY, IRYZ,IRZX	Reflections in local coordinate system 1 coordinate planes
TOLERANCE	Field tolerance

Parameters for 20-Noded Bricks

BR20	
XCENTRE, YCENTRE, ZCENTRE,	Local coordinate system 1
PHI1, THETA1, PSI1	
X0, Y0, Z0	Local coordinate system 2 (ori-
	gin)
T, P, S	Local coordinate system 2 (Euler
	angles)
X1, Y1, Z1	Bottom right corner of front face
X2, Y2, Z2	Top right corner of front face
X3, Y3, Z3	Top left corner of front face
X4, Y4, Z4	Bottom left corner of front face
X5, Y5, Z5	Bottom right corner of back face
X6, Y6, Z6	Top right corner of back face
X7, Y7, Z7	Top left corner of back face
X8, Y8, Z8	Bottom left corner of back face
X9, Y9, Z9	Mid-point of right edge of front
	face
X10, Y10, Z10	Mid-point of top edge of front
	face
X11, Y11, Z11	Mid-point of left edge of front
	face
X12, Y12, Z12	Mid-point of bottom edge of
	front face
X13, Y13, Z13	Mid-point of bottom right edge
X14, Y14, Z14	Mid-point of top right edge
X15, Y15, Z15	Mid-point of top left edge
X16, Y16, Z16	Mid-point of bottom left edge
X17, Y17, Z17	Mid-point of right edge of back
	face
X18, Y18, Z18	Mid-point of top edge of back
	face

BR20 (continued)	
X19, Y19, Z19	Mid-point of left edge of back face
X20, Y20, Z20	Mid-point of bottom edge of back face
CURD, SYMMETRY, PHASE	Current density, symmetry and drive label
IRXY, IRYZ,IRZX	Reflections in local coordinate system 1 coordinate planes
TOLERANCE	Field tolerance

The CONDUCTOR Sub-command DEFINE

Menu Route

 $\texttt{define} \downarrow$

 $\texttt{Conductors} \ \rightarrow \ \texttt{Define} \ \texttt{a} \ \texttt{conductor}$

Command Line Parameters

Sub-command	DEFINE		
Parameter	Default	Function	
END	none	Conductor type	
		SOLENOID	Simple solenoid
		GSOLENOID	Generally orientated
			solenoid
		RACETRACK	Simple racetrack
		GRACETRACK	Generally orientated
			racetrack
		BEDSTEAD	Simple bedstead
		GBEDSTEAD	Generally orientated
			bedstead
		HELIX	Helical end racetrack
		CPEND	Constant perimeter end
			racetrack
		STRAIGHT or	Straight bar
		GSTRAIGHT	
		ARC or GARC	Circular arc
		BR8	8-noded brick
		BR20	20-noded brick

S

Sub-command	DEFINE (continued)	
Parameter	Default	Function	
DEFAULT	max	Number of an exist reference default v given, then the high of the same type is	ting conductor used to alues. If no number is nest numbered conductor used.

The program prompts the user to provide values for all the parameters appropriate for the conductor type. The input lines are decoded in the same way as top-level commands without a command name. This means that the help character, !, can be used to get more information about the parameters and can be used twice, **!!**, to get individual parameter prompting.

The default value is given in parentheses following each parameter name. If no value is given, there is no default and a value must be supplied.

The full parameter lists were given above with the descriptions of the conductors, in the order in which they are requested from the user.

Examples In the example which follows various forms of input are used during the definition of a solenoid.

> OPERA > cond Define and list conductor data Conductor definition and editing.

Sub-commands: Define, Modify, Erase, Print, Write and Quit; ! for help. COND > define solenoid Define conductors. Specify new values by: giving values positionally OR assigning PARAMETER=value OR hitting <return> to accept all defaults.

Conductor 1: Solenoid around Y' axis. Local coordinate system 1 XCENTRE(0), YCENTRE(0), ZCENTRE(0) COND > xcen=5 ycen=3 zcen=2 Solenoid cross section in X'Y' plane. X1(), Y1(), X2(), Y2() COND > ! Parameter Value Meaning X1 Local X coordinate of bottom inside corner

4-42

Local Y coordinate of bottom Υ1 inside corner X2 Local X coordinate of top inside corner Local Y coordinate of top inside Y2 corner COND > 3 3 3 5 Solenoid cross section in X'Y' plane. X3(), Y3(), X4(), Y4()COND > 5 5 5 3 Solenoid cross section curvatures. CU1(0), CU2(0), CU3(0), CU4(0)COND > cu3=0.05 Current density, symmetry and drive label CURD(), SYMMETRY(1), PHASE(ONE) COND > !! There are 3 Parameters For each parameter: hit return to accept default OR enter new value OR type \$H for help OR type \$S to skip remaining parameters and execute command OR type \$A to skip remaining parameters and abort command NO. Value Meaning Name 1 CURD Current density !! > 10 2 SYMMETRY 1 Symmetry code (1 or 2) !! > 2 3 PHASE ONE Drive label !! > Type return to obey command or \$A to abort !! > Sub-commands: Define, Modify, Erase, Print, Write and Quit; ! for help. COND >

 $\mathtt{define} \downarrow$

 $\texttt{Conductors} \ \rightarrow \ \texttt{Erase} \ \texttt{conductors}$

Command Line Parameters

Sub-command	ERASE	
Parameter	Default	Function
C1	none	Number of first conductor to be erased.
C2	none	Number of last conductor to be erased. C2 can be set to * to indicate the highest numbered conductor.

The ERASE sub-command removes a range of conductors from the conductor database. For example, if C1 is set to 3 and C2 to 6 then conductors with numbers 3, 4, 5 and 6 will be erased. C2 can be set to * to indicate the highest numbered conductor.

The remaining conductors with numbers greater then C2 are renumbered to give a contiguous set.

The CONDUCTOR Sub-command EXTERNAL

Menu Route

 $\mathtt{DEFINE} \downarrow$

Conductors \rightarrow Define external field

Command Line Parameters

Sub-command	EXTERNAL	
Parameter	Default	Function
HXEXT	0	X-component of external field.
HYEXT	0	Y-component of external field.
HZEXT	0	Z-component of external field.
DRIVELABEL		Drive label.

The EXTERNAL sub-command defines a uniform external field which can be used in addition to field from conductors in TOSCA (magnetostatics), SCALA, CARMEN and ELEKTRA analyses. A drive label must also be given so that multiple case scaling factors or a time variation of the external field can be defined (page 4-156).

The CONDUCTOR Sub-command MODIFY

Menu Routes

 $\texttt{define}^{\downarrow}$

 $\texttt{Conductors} \ \rightarrow \ \texttt{Modify} \ \texttt{conductors}$

 $\begin{array}{c} \texttt{MODIFY} \\ \texttt{Conductors} \end{array}$

Command Line Parameters

Sub-command	MODIF	Y
Parameter	Default	Function
C1	none	Number of first conductor to be modified.
C2	none	Number of last conductor to be modified. C2 can
		be set to * to indicate the highest numbered con-
		ductor.
END	none	Conductor type: SOLENOID, GSOLENOID,
		RACETRACK, GRACETRACK, BED-
		STEAD, GBEDSTEAD, HELIX, CPEND,
SVMMETRV	nona	Symmetry code
	none	X apprding to of origin of local system 1
	none	X coordinate of origin of local system 1.
TUENTRE	none	Y coordinate of origin of local system 1.
ZCENTRE	none	Z coordinate of origin of local system 1.
CURD	none	Current density.
PHI1	none	Euler angle phi (local system 1).
TOLERANCE	none	Tolerance on fields.
X0	none	X coordinate of origin of local system 2.
Y0	none	Y coordinate of origin of local system 2.
Z0	none	Z coordinate of origin of local system 2.
Т	none	Euler angle theta (local system 2).
Р	none	Euler angle phi (local system 2).
S	none	Euler angle psi (local system 2).
IRXY	none	Reflection code in xy plane of local system 1.
IRYZ	none	Reflection code in yz plane of local system 1.
IRZX	none	Reflection code in zx plane of local system 1.
X1	none	X coordinate of corner of conductor cross sec-
		tion.
Y1	none	Y coordinate of corner of conductor cross sec-
		tion.
X2	none	X coordinate of corner of solenoid cross section.

Sub-command	MODIFY (continued)		
Parameter	Default	Function	
Y2	none	Y coordinate of corner of solenoid cross section.	
X3	none	X coordinate of corner of solenoid cross section.	
Y3	none	Y coordinate of corner of solenoid cross section.	
X4	none	X coordinate of corner of solenoid cross section.	
Y4	none	Y coordinate of corner of solenoid cross section.	
А	none	Thickness of conductor in x or radial direction.	
В	none	Width of conductor in y or azimuthal direction.	
H1	none	Length of straight section.	
H2	none	Length of upright (BEDSTEAD)	
		Local Z coordinate of midpoint of cross-over (HELIX).	
R1	none	Radius: inner radius of arc (RACETRACK, BEDSTEAD, ARC).	
		Radius of cylinder (HELIX, CPEND).	
R2	none	Radius: inner radius of arc (BEDSTEAD).	
		Width of cross-over (HELIX).	
		Radius of generating cylinder (CPEND).	
PHI	none	Angle of ARC.	
ALPHA	none	Angle of straight from mid plane of cylinder (HELIX and CPEND).	
BETA	none	Angle of end of helix (HELIX), or cutter (CPEND).	
CU1	none	Curvature of cross section of solenoid (points 1 to 2).	
CU2	none	Curvature of cross section of solenoid (points 2 to 3).	
CU3	none	Curvature of cross section of solenoid (points 3 to 4).	
CU4	none	Curvature of cross section of solenoid (points 4 to1).	
FIT	none	Fit of straight section to cylinder: TANGEN- TIAL or FITTING (CPEND).	
PHASE	none	Drive label.	
THETA1	none	Euler angle theta (local system 1).	
PSI1	none	Euler angle psi (local system 1).	

The MODIFY command can be used in two ways. If only the range of conductors is specified on the MODIFY sub-command then the program issues the same prompts as are used by the DEFINE sub-command (page 4-40). The displayed values are taken from the first conductor in the range and any changes made by the user are applied to all the conductors within the range which are of the same type.

The second way of using MODIFY makes use of the other parameters of the MODIFY command. The values of the parameters used on the command are applied to all the conductors in the range. This way of using MODIFY should be used with care, especially if more than one conductor is specified by C1 and C2, and the range includes conductors of more than one type. It is possible to create conductors with invalid data, since some of the parameters have different meanings for different conductor types. However it is very powerful for changing parameters which apply to all conductors, such as the current density (CURD).

N.B. Only the second way of using **MODIFY** is available from the GUI.

Expressions can also be used to good effect in this second way of using MODIFY. The expressions are evaluated for each conductor; within each evaluation any parameters which are referenced have the correct values for the conductor concerned. However, the original values are used in each expression. Thus setting A=A*2, CURD=CURD/A/B would set the current density using the original value of A, not the updated value.

```
• Example: MODIFYing the conductor defined above.
Sub-commands: Define, Modify, Erase, Print, Write
and Quit; ! for help.
COND > modi 1 1
Modify conductors.
Modifying conductor 1.
Specify new values by:
giving values positionally
OR assigning PARAMETER=value
OR hitting <return> to accept all defaults.
Conductor 1: Solenoid around Y' axis.
Local coordinate system 1
XCENTRE(5), YCENTRE(3), ZCENTRE(2)
COND >
Solenoid cross section in X'Y' plane.
X1(3), Y1(3), X2(3), Y2(5)
COND > x1=2 x2=2
```

Solenoid cross section in X'Y' plane. X3(5), Y3(5), X4(5), Y4(3) COND > x3=4,,4 Solenoid cross section curvatures. CU1(0), CU2(0), CU3(0.05000), CU4(0) COND > Current density, symmetry and drive label

```
CURD(10), SYMMETRY(1), PHASE(ONE)
COND >
Sub-commands: Define, Modify, Erase, Print, Write
and Quit; ! for help.
COND > modi 1 * curd=curd/2
Modify conductors.
```

The CONDUCTOR Sub-command PRINT

Menu Route	$\texttt{DEFINE}_{\downarrow}$
	Conductors $ ightarrow$ Print data

Command Line Parameters

Sub-command	PRINT	
Parameter	Default	Function
C1	1	Number of first conductor to be printed.
C2	*	Number of last conductor to be printed. C2 can be set to * to indicate the highest numbered con- ductor.

The **PRINT** sub-command lists the parameters of the selected range of conductors to the terminal and to the log file. It uses the same format for the prompts in **DEFINE** and **MODIFY**.

The CONDUCTOR Sub-command QUIT

Menu Route

 $\mathtt{define} \downarrow$

 $\texttt{Conductors} \ \rightarrow \texttt{Return}$

Command Line				
Parameters	Sub-command	QUIT		
	No Parameters			
	The OLUT sub c	ommand	leaves the CO	

The QUIT sub-command leaves the CONDUCTOR command and returns to the top-level commands.

The CONDUCTOR Sub-command WRITE

Menu	Route
------	--------------

 $\mathtt{DEFINE} \downarrow$

Conductors \rightarrow Write data to a file

Command Line Parameters

Sub-command	WRITE	
Parameter	Default	Function
FILE	none	Name of file.

The WRITE sub-command creates a command input FILE containing the CONDUCTOR command, the set of DEFINE sub-commands for all the conductors currently defined and the QUIT sub-command.

In the pre processor, conductor files are useful for copying the conductors from one data set to another, or for transferring the data into the post processor. The actual commands used to define the conductors are also included in any file created by the pre processor top level command WRITE.

If no file name extension is given, the program adds the extension *cond*.

Conductor definition (.*cond*) files can be read:

- into the pre-processor using the READ command (page 4-146) or the 'built-in' command, \$ COMINPUT (page 2-25),
- into the Modeller using the IMPORT command (page 3-90)
- into the post processor using the **CONDUCTOR IMPORT** command (page 5-44).

For hints on how to write compatible command scripts for all 3 programs see "Conditional commands" on page 2-20.

The **DEFINE** Command

Menu	Route
------	-------

 $\texttt{define} \downarrow$

Define new mesh \rightarrow Finite element mesh \rightarrow 8 or 20 node conductors

Command Line Parameters

Command	DEFINE		
Parameter	Default	Function	
TYPE	MESH	Type of elements to be defined.	
		CONDUCTOR	8 or 20 node conductor elements
		MESH	Finite elements mesh
KEEP	YES	Keep existing con	nstruction lines
		NO	Construction lines removed at start of DEFINE sequence.
		YES	Construction lines kept from any previous DEFINE commands.
THREED	NO	Use 3D Viewer	
		NO	No 3D Viewer
		YES	3D View starts when first layer is complete.

DEFINE is the command which puts the OPERA-3d pre processor into its finite element mesh creation mode. The pre processor only supports one type of finite element mesh construction: a surface defined by a set of facets is specified and is then extruded or swept through space to create a volume discretisation. The user input is structured by the program. Points used to define the geometry are defined first, followed by the surface facets and the element subdivision of these facets. It is possible to define more points while facets are being defined, and more facets while subdivisions are being defined. The surface of facets is then extruded or swept through space thus creating a series of layers of volumes. The topology of the initial surface is maintained during the extrusion operations, but the point coordinates can be changed in the new surface created by each extrusion or sweep. Once the complete model has been created, the volumes in the layers of the mesh can be assigned material attributes and boundary conditions can be assigned to the facets of the volumes.

The **TYPE** parameter allows definition of the finite element mesh for the solution of the model (**TYPE=MESH**) or 8 or 20 node brick conductor elements (**TYPE=CONDUCTOR**). After definition conductor elements are cop-

ied into the conductor storage area as BR8 or BR20 conductors with total current of 1 Amp and can only be modified by use of the CONDUCTOR sub-commands. (N.B. The cross sections of conductors can be only triangles or quadrilaterals.) The finite element mesh can be modified using MODIFY and have extra layers added using EXTEND.

The KEEP parameter affects construction lines defined during a previous use of the DEFINE command. If **KEEP=NO**, any construction lines which already exist are removed before any new data is defined.

The **THREED** parameter can be used to request the 3D Viewer to be used during the **DEFINE** command to display the outline of the model. When **THREED=YES** is selected, the viewer is started when the first layer has been created and is updated whenever the two dimensional display is refreshed.

Labels are assigned to each entity (i.e. point, line, facet or volume) in the mesh. All entities have the label ALL, facets have boundary condition names and volumes have material names, potential types and element types. Additional labels can be added or removed with the LABEL command (page 4-123). These labels can be used to select subsets of the entities for DISPLAY.

Each time the **DEFINE** command is used to create a finite element mesh, a new mesh is started. Up to 100 meshes can be defined, each one forming part of any data set prepared for analysis. The user is responsible for ensuring that the meshes are contiguous at their interfaces.

Finite Element Meshing

The elements used by the analysis programs are created by subdividing volumes created by the pre processor. These volumes are formed by extruding facets on the base plane through space. There are two types of mesh available.

- If all the facets are made up from 3 or 4 lines with regular subdivisions, the volumes can be meshed using hexahedra (which can be degenerate). Regular subdivision means that for 4 sided facets, opposite sides must have the same subdivision and for 3 sided facets, two sides must have the same subdivision.
- Any model can be meshed with tetrahedra. Tetrahedral meshes can be generated in extruded polygonal volumes with no restrictions on the subdivision of the edges. The points defining any polygonal facet must be coplanar.

In either type of mesh, the lines can be straight (2 points) or quadratic curves (3 points). The position of the mid-side point of quadratic curves

affects the subdivision of the volumes. If it is not central there will be smaller elements at the end of the side to which it is closer. Any combination of straight and curved lines may be used. Thus, facets can be anything from 3 noded triangles to many-noded curved sided polygons. In the same way extrusions can be linear or quadratic. Quadratic extrusions have midside points which can be moved independently, allowing definition of curves or non-uniform subdivision in the extrusion direction.

DEFINE start up sequence and modes

		1 5
Pre-defined Coordinate Systems		
Keyword	Menu item	Coordinate system
XY	XY plane, extrude in Z	Same as global, extrusion direction
		Ζ.
ΥZ	YZ plane, extrude in X	U=Y, V=Z, W=X, extrusion direc-
		tion X.
ZX	ZX plane, extrude in Y	U=Z, V=X, W=Y, extrusion direc-
		tion Y.
NEW	(keyboard only)	Define a new coordinate system.

However any other right handed system can be defined to give the orientation of the base plane and the default extrusion direction. The coordinate system is defined by its origin, and Euler angles (page 2-34). (This coordinate system can be changed during DEFINE using Change View menu item or the V cursor hit.) The most appropriate system should be selected. The default extrusions will be normal to the plane selected and unless a coordinate triple is specified for a point, the point will be given the default coordinate of the plane. Once a plane has been selected, subsequent input of point coordinates is specified in a local U, V, W coordinate system, where U and V are in the plane and W is out of the plane.

W coordinate The user is next asked to input the default W coordinate of the plane, for points whose W coordinate is not explicitly defined.

Window size Finally, the user is asked to input window size to be used for subsequent graphical display of the point and surface data. This is only an initial specification and can be changed at any time subsequently. The input request is for *umin*, *umax*, *vmin* and *vmax* which may be supplied in free format. The

Coordinate system

After the **DEFINE** command has been issued, the user will be asked to select the default coordinate system to be used for input and display of the initial surface (base plane). There are 3 pre-defined systems

screen is cleared at this point and the program enters Point Definition Mode.

The graphics display indicates, at the top, the mesh number and the plane or extrusion layer number currently being worked on, and, at the bottom, the definition or name of the local (UVW) coordinate system.

Aspect Ratio Searching

When the user selects a point or line on the screen, the program searches for the point or line nearest to the cursor position in real coordinate space. If the horizontal and vertical axes limits are very different, i.e. if

$$\alpha = \frac{u_{\max} - u_{\min}}{v_{\max} - v_{\min}}$$
(4.1)

and $\alpha \gg 1$ or $\alpha \ll 1$ it might be difficult to select the intended object.

To make it possible to select correctly in such circumstances, aspect ratio searching can be switched on or off using cursor command \mathbf{Z} or menu item **Aspect ratio search**. When it is switched on, the program makes use of the window aspect ratio (α), to find the nearest object.

Defining Meshes with the GUI

When the **DEFINE** command is used with the GUI, the Base Plane Definition Modes are presented on one menu:

De	efine Baseplane Menu
Fast Polygon Input	To define points line and facets using grids or
	construction lines (see page 4-59).
Point Input	To define points using keyboard input of coor-
	dinates, grids and construction lines (see
	page 4-56).
Facet Input	To select points to form facets (see page 4-62).
Subdivision	To subdivide lines (facet sides) to set the finite
	element mesh size (see page 4-69).
Extrude	To create and edit the first layer (coordinates,
	materials and boundary conditions (see page 4-
	72).
Escape from baseplane	To create the first layer of size 1 with no editing
	(if there is at least one facet).

The two input methods, Fast Polygon Input and Point Input with Facet Input can be used separately or together to create the base plane facets.

There are several additional points which should be noted:

- All the data for the first layer should be defined first. The **Define new mesh** option should be used to define the first layer and the **Extend existing mesh** option for one or more additional layers (see "The EXTEND Command" on page 4-112).
- Not all options are available from the menus. For example, the coordinate system menu only allows the 3 pre-defined systems.
- It is not possible to abort the definition; the layer must be completed. However, as soon as there is at least one facet, the menu option **Escape from baseplane** is available. This completes all remaining modes of the baseplane definition and extrudes by one unit. The model can be subsequently edited using the **REDEFINE** command (page 4-148).

Defining Meshes with the Keyboard

In keyboard mode, the program steps through the modes of the DEFINE command sequentially, each one being terminated with a Q cursor hit. During Point Definition Mode, Fast Polygon Input can be accessed; during Facet Definition Mode, Group Operations can be accessed. There are cursor and text commands which can be used to move between the various modes in almost any order.

Construction Lines and Grids

Whenever point coordinates can be defined, they can be supplied graphically using construction lines or grids.

Construction lines

Construction lines are straight lines (LINE) or arcs (ARC). The arc is in fact a linear interpolation in the cylindrical polar coordinate system. It will only be an arc of a circle if the radial coordinates of the end points are the same. The definitions can be given numerically or by choosing points which have already been defined: LINES are defined by two **space>** cursor hits; ARCS are defined by either a **C** and two **space>**s to specify the centre and two end points or by 3 **space>** cursor hits to specify points on a circle. Cursor defined LINES extend beyond the points which define them. Cursor defined ARCS are minor arcs if defined by centre and end points or complete circles if defined by 3 points on the circumference.

The menu item **Enter C_lines** or cursor hit I can be used for define construction lines.

- *In keyboard mode* this produces a request to specify data defining construction lines, together with the format of the specification.
- *In menu mode* the type of construction line can be selected from a menu and the values supplied via a ParameterBox:

	Construction Line Sub-commands
Command	Parameters and Function
ARC	uc vc r1 t1 r2 t2
	An arc centred on (uc, vc) starting at polar coordinates $(r1, t1)$ and ending at $(r2, t2)$. Both r and t vary linearly between the end points.
ARC	CURSOR
	An arc defined by cursor hits:
	Either: select points for centre with C and end points with
	<space>.</space>
	or: select 3 points on circumference of circle with <space></space> .
LINE	u1 v1 u2 v2 angle
	A straight line from $(u1, v1)$ to $(u2, v2)$ rotated by <i>angle</i> around $(u1, v1)$.
LINE	CURSOR
	A straight line through 2 existing points chosen by space > cursor hits.
QUIT	End the definition of construction lines.

Points can be defined on the nearest construction line using menu item On nearest C_line or cursor hit N, or at the nearest intersection of construction lines using menu item At C_line intersection or cursor hit X. The set of intersections also includes the end points of the construction lines. In each case the cursor hit only specifies the U and V coordinates. The W coordinate used is the default coordinate of the plane.

The nearest construction line can be erased using menu item **Remove** C_{lines} or cursor hit E and the lines can be listed at the terminal using menu item List C_{lines} or cursor hit L.

Grids

Grids are two-dimensional arrays of points in cartesian or polar coordinates. A grid can be switched on or off following menu item **Grid** or **G** cursor hit.

Grid Sub-commands		
Command	Parameters and Function	
CARTESIAN	du dv Define grid with spacing du in the horizontal direction and dv in the vertical.	
POLAR	$dr d\theta$ Define grid with spacing dr in the radial direction and $d\theta$ in the azimuthal direction.	
NONE	Remove any existing grid.	

When a grid exists, and menu item At the mouse or cursor hit <space> selects the closest grid point.

Point Definition Mode

The user must specify points to define the corners and mid-side points of the facets in the base plane. Additional points can be entered by returning to Point Definition Mode from Facet Definition Mode. It is also possible to copy points and facets in the Facet Group Operations Mode.

Points are defined using the graphics cursor or numerically using the keyboard. The cursor can be used to position points at the cross-hairs position, or at points on construction lines or grids. Keyboard input can be in cartesian or cylindrical polar coordinates with respect to the current local coordinate system origin.

When enough points have been defined, the user should leave the Point Definition Mode and move on to Facet Definition Mode.

Full details of construction lines and grids are give above and the keyboard input and the cursor commands are described in the following sections.

Keyboard input Keyboard input mode can be used to specify coordinates of points in

- cartesian coordinates: menu item Give U, V, W or cursor hit C
- cylindrical polar coordinates: menu item Give R, Theta, W or cursor hit P. The origin of the polar coordinate system is the same as the UVW system, with the axis in the W direction and zero azimuthal angle (T) in the U direction.

OPERA-3d Reference Manual

For each point 1, 2 or 3 values or expressions should be specified. The first defines the U or R coordinate. If it is omitted by using a comma at the start of the command line, the value of U or R will be the last value given for U or R. The second value defines the V or T coordinate. If it is omitted the value of V or T will be the last value used for V or T. When keyboard input mode is entered, U and V have default values taken from the cursor position, and R and T are both zero. The third value defines the W coordinate and always defaults to the default value for the plane.

To return to cursor mode type the keyword CURSOR or select the Quit button in the DialogBox.

	Point Mode Menu a	nd Cursor Hits
Cursor hit	Menu item	Function
<space></space>	At mouse	Define a point at the cursor cross
		hair position or closest grid point.
A		Abort the DEFINE command. If
		Point Definition Mode has been
		re-entered from Facet Definition
	· .	Mode then A acts like Q .
C	Give U, V, W	Switch to input from keyboard in
		cartesian coordinates U v w.
D	Delete point	Delete point closest to cursor
		cross hair.
Е	Remove C_line	Erase construction line closest to
		cursor cross hair.
F		Enter Fast Polygon Input Mode
		(see page 4-59).
G	Grid	Define or remove a grid (see
		page 4-55).
н		Display menu help message
		explaining all the cursor options.
I	Enter C_lines	Input construction line specifica-
		tions (see page 4-53).
L	List C_lines	List construction line specifica-
		tions.
м	Move point	Move the nearest point. The point
		can be repositioned using cursor
		hits which have the same mean-
		ings as for point definition. See
		page 4-58.

The Point Mode Menu and Cursor Hits

P	oint Mode Menu and Cu	rsor Hits (c	ontinued)
Cursor hit	Menu item	Function	
N	On nearest C_line	Define a point on the nearest con- struction line with minimum nor- mal distance.	
Ρ	Give R, Theta, W	Switch to inp local cylindri nates R0W.	out from keyboard in ical polar coordi-
Q	Go to Facet Definition	Leave Point Definition Mode and move on to Facet Definition Mode.	
R	Re-draw picture	Reconstruct gram request replies are: 4 numeric values	the display. The pro- s a new size. Valid umin, umax, vmin, vmax Default values are
		CURSOR	Select diagonally opposite corners of the display area with <space></space> cursor hits.
		RESTORE	Return to previous screen size.
		BOUND	Use bounding rec- tangle of geometry.
		<return></return>	Reconstruct at the same size.
Т	Show coordinates	Type the coo closest to the coordinates a (UVW) and p nates.	rdinates of the point cross-hairs. The re given in cartesian polar (RθW) coordi-
υ	Undo	Undo the last move.	
v	Change view	Change the UVW coordinate sys- tem. This affects the view and the extrusion direction.	
x	At C_line intersection	Define a point at the closest con- struction line intersection or end.	
Z	Aspect-ratio search	Switch aspector of f (page 4	t ratio searching on 4-52).

The Point
Repositioning
Mode Menu and
Cursor Hits

Point Repositioning Mode Menu and Cursor Hits			
Cursor hit	Menu item	Function	
<space></space>	At mouse	Reposition the point at the cursor cross hair position or closest grid point.	
С	Give U, V, W	Switch to input from keyboard in cartesian coordinates UVW. Coordinates should be entered in free format. Default values of U, V and W are the values prior to the move. They can be accessed via the variables #1 , #2 and #3 . Type Q to leave point in its present position.	
Е	Remove C_line	Erase construction line closest to cursor cross hair.	
G	Grid	Define or remove a grid (see page 4-55).	
н		Display menu help message explaining all the cursor options.	
I	Enter C_lines	Input construction line specifica- tions.	
L	List C_lines	List construction line specifica- tions.	
N	On nearest C_line	Reposition the point on the near- est construction line with mini- mum normal distance. This only affects the U and V coordinates; W remains unchanged.	
P	Give R, Theta, W	Switch to input from keyboard in local polar coordinates $R\theta W$. Coordinates should be entered in free format. Default values of R, ϑ and W are the values prior to the move. They can be accessed via the variables #1 , #2 and #3 . Type Q to leave point in its present position.	
Q	Return without moving	Leave the point at its previous position.	

Cursor hit	Menu item	Function	
R	Re-draw picture	Reconstruct a gram request replies are:	the display. The pro- s a new size. Valid
		4 numeric values	<i>umin, umax, vmin, vmax</i> <i>vmax</i> Default values are
		CURSOR	Select diagonally opposite corners of the display area with <space></space> cursor hits.
		RESTORE	Return to previous screen size. Use bounding rec-
		<return></return>	tangle of geometry. Reconstruct at the same size.
T	Show coordinates	Type the coo closest to the select it for r of the point a coordinates a (UVW) and p nates.	rdinates of the point cross-hairs, and epositioning instead lready selected. The re given in cartesian polar ($R\theta W$) coordi-
X	At C_line intersection	Reposition the construction end. This onl coordinates; unchanged.	e point at the closest line intersection or y affects the U and V W remains
Z	Aspect-ratio search	Switch aspector of f (page 4	t ratio searching on 4-52).

Fast Polygon Input Menu and Cursor Hits

In Fast Polygon Input Mode, points, lines and facets are all defined together. Points can be positioned at the cursor position and can be accurately positioned using construction lines or grids. Duplicate points are removed and new polygons can be defined using the points of existing polygons.

Mid-side points can be used to define polygon sides which are curved or have non-uniform element sizes. The mid-side points must be between the 1/4 and 3/4 points along the length of the side (this is verified by the CHECK

command, page 4-18). If the mid-side point is not half way along the side the discretisation will also vary quadratically, with smaller elements near the corner which is closer to the mid-side point. This can be used to grade the mesh even for straight sided facets.

A mid-side point is defined after menu item **Next point is mid-side** or cursor hit **M** and can be defined using the cursor position, construction lines, grids or points of existing polygons. Mid-point definition can be cancelled using menu item **Cancel** or repeating cursor hit **M**. Points which have been used as corners cannot subsequently be used as mid-side points, and vice-versa.

Although the name *Polygon* is used, 3 or 4 sided facets can be defined for quadrilateral meshing.

	Fast Polygon Input Men	u and Cursor Hits
Cursor hit	Menu item	Function
<space></space>	At mouse	Define a polygon corner at the cursor cross hair position or closest grid point.
А		Abort the DEFINE command.
В	Backtrack	Remove the last corner from the current polygon.
D	Delete facet	Delete the facet containing the cursor cross hair.
Е	Remove C_line	Erase construction line closest to cursor cross hair.
F	Close polygon	Close an open polygon.
G	Grid	Define or remove a grid (see page 4-55).
н		Display menu help message explaining all the cursor options.
I	Enter C_lines	Input construction line specifications (see page 4-53).
L	List C_lines	List construction line specifica- tions.
м	Next point is mid-side	The next point will be a mid- side point. It can be defined using <space></space> , O , N or X .
N	On nearest C_line	Define the next corner on the nearest construction line with minimum normal distance.

Fast l	Fast Polygon Input Menu and Cursor Hits (continued)		
Cursor hit	Menu item	Function	
0	At nearest old point	Define next co be at the near already been	orner of polygon to est point which has defined.
Q	Return	Return to Poi Mode.	nt Definition
R	Re-draw picture	Reconstruct t program requ Valid replies	he display. The lests a new size. are:
		4 numeric values	<i>umin, umax,</i> <i>vmin, vmax</i> Default values are the current set- tings.
		CURSOR	Select diagonally opposite corners of the display area with <space></space> cursor hits.
		RESTORE	Return to previ- ous screen size.
		BOUND	Use bounding rectangle of geometry.
		<return></return>	Reconstruct at the same size.
x	At C_line intersect	Define the ne nearest constr section or end	ext corner at the ruction line inter-

Facet Definition Mode

Facets are defined by connecting the points on the base plane. Facets can be triangles, quadrilaterals or higher-order polygons with straight or curved edges. Points are selected in sequence around each facet:

- corners: menu items Corner ... no auto-close or polygon corner (cursor hits C or P). These are equivalent except that if a polygon corner is defined, the facet is not limited to four sides.
- mid-side points: menu item Mid-side (cursor hit M)

Close or cursor hit \mathbf{F} can be used to close the facet. To extend a polygon to more than 4 sides, **polygon corner** or cursor hit \mathbf{P} must be used for one of the first 4 corners.

Facets with 4 straight sides can be defined more quickly by selecting the 4 corner points with Corner ... auto-close after 4 or cursor hit <space>.

Sides with mid-side points are quadratic. The mid-side points must be between the ¼ and ¾ points along the length of the side (this is verified by the CHECK command, page 4-18). If the mid-side point is not half way along the side the discretisation will also vary quadratically, with smaller elements near the corner which is closer to the mid-side point. This can be used to grade the mesh even for straight sided facets.

Points which have been used as corners cannot subsequently be used as mid-side points, and vice-versa.

Additional points can be defined by returning to Point Definition Modes (menu item **Return to points** or cursor hit **N**).

Facets can be copied and points transformed using the Facet Group Operations Mode which can be entered using menu item Go to Group Ops. or cursor hit G.

Care must be taken to ensure that the entire base plane is covered with facets. When the base plane is complete, menu item **Go to Subdivisions** or cursor hit **Q** leaves Facet Definition Mode and moves on to Base Plane Subdivision Mode. It is possible to return to Facet Definition Mode from Base Plane Subdivision Mode in order to define more facets.

Full details of the cursor commands are given in the following section.

	Facet Mode Menu and Cursor Hits		
Cursor hit	Menu item	Function	
<space></space>	auto-close after 4	Select nearest point as a corner. Facet closes automatically after 4 sequential <space></space> hits.	
A		Abort DEFINE command.	
C	no auto-close	Select nearest point as a corner.	
D	Delete facet	Delete the facet containing the cross-hairs.	
E	Erase last point	Forget the last point selected for the current face.	

Facet Mode Menu and Cursor Hits

I

Fa	Facet Mode Menu and Cursor Hits (continued)		
Cursor hit	Menu item	Function	
F	Close	Close the current face (after at least	
		3 corners).	
G	Go to Group Ops.	Enter Facet Group Operations	
		Mode to group points for transfor-	
		mations or facets for copying	
		(page 4-64).	
н		Display menu help message	
		explaining all the cursor options.	
м	Mid-side	Select nearest point as a mid-side	
		point. This cannot be used for the	
		first point of a facet. The facet is	
		closed automatically if the mid-	
		side point is on the fourth side	
		unless P has been used to define a	
		corner.	
N	Return to points	Enter Point Definition Mode to	
		define or move points.	
Р	polygon corner	Select nearest point as a corner.	
		Using this, rather than <space></space>	
		or C identifies the facet as a poly-	
		gon which can have more than 4	
		sides.	
Q	Go to Subdivisions	Leave the facet definition mode	
		and move on to Base Plane Subdi-	
		vision Mode.	

F	Facet Mode Menu and Cursor Hits (continued)			
Cursor hit	Menu item	Function		
R	Re-draw picture	Reconstruct t gram request replies are:	Reconstruct the display. The pro- gram requests a new size. Valid replies are:	
		4 numeric values	<i>umin, umax, vmin, vmax</i> Default values are the current settings.	
		CURSOR	Select diagonally opposite corners of the display area with <space></space> cur- sor hits.	
		RESTORE	Return to previous screen size.	
		BOUND	Use bounding rec- tangle of geometry.	
		<return></return>	Reconstruct at the same size.	
v	Change view	Change the U tem. This aff extrusion dire	JVW coordinate sys- ects the view and the ection.	
Z	Aspect-ratio search	Switch aspec off (page 4-5	t ratio searching on or 2).	

Facet Group Operations Mode

There are two Group Operations which can be performed on the base plane: the coordinates of groups of points can be transformed and groups of facets can be copied:

- A point is added into the point group using menu item Select/deselect point or cursor hit N. This selects the nearest point or if it is repeated for a selected point it removes the point from the group.
- The coordinates of points in the point group can be transformed using menu item **Transform points** or cursor hit **T**. If there is no point group then all the points are transformed.
- A facet is added to the facet group using menu item Select/de-select or cursor hit **F** cursor hit. This selects the facet containing the crosshairs or if it is repeated for a selected facet it removes the facet from the group.

- The facets in the facet group can be copied using menu item Copy facets or cursor hit C. If there is no facet group then all the facets are copied. Multiple copies can be made. The program prompts the user for the number of new copies, e.g. to end up with 8-fold symmetry it is necessary to create 7 new copies.
- The copy operation creates new facets, lines and points, and then transforms the coordinates of the points. For multiple copies the transformation applies between the original points and the first copy, between the first and second copies, between the second and third, etc. The index number of the copy (**#COPY**) can be used in transformation options CARTESIAN and POLAR.

At any one time there can be either a point group **or** a facet group, not both.

The transformation options are: CARTESIAN, DISPLACE, MANGLE, MIRROR, POLAR, PROJECT, ROTATE and SCALE.

When all transformations and copies are complete, menu item Return to facets or cursor hit **Q** is used to return to Facet Definition Mode. The program prompts for a tolerance which is used to coalesce coincident points. Points are coalesced if the differences in U, V and W coordinates are all less than the tolerance.

Full details of the menu items, cursor commands and the transformation options are given in the following sections.

	Group Operations Menu and Cursor Hits		
Cursor hit	Menu item	Function	
С	Copy facets	Copy the facet group, or all the facets if there is no group.	
F	Select/de-select facet	Select or de-select the facet con- taining the cross-hairs as a mem- ber of the facet group.	
н		Display menu help message explaining all the cursor options.	
N	Select/de-select point	Select or de-select the nearest point as a member of the point group.	
Q	Return to Facets	Leave the Facet Group Operations Mode and return to Facet Defini- tion Mode.	

on Menu and **Cursor Hits**

Group Operation
Monu and

Grou	Group Operations Menu and Cursor Hits (continued)		
Cursor hit	Menu item	Function	
R	Re-draw picture	Reconstruct th gram requests replies are:	ne display. The pro- a new size. Valid
		4 numeric values	<i>umin, umax, vmin, vmax</i> <i>vmax</i> Default values are the current settings.
		CURSOR	Select diagonally opposite corners of the display area with <space></space> cursor hits.
		RESTORE	Return to previous screen size.
		BOUND	Use bounding rec- tangle of geometry.
		<return></return>	Reconstruct at the same size.
Т	Transform points	Transform the the points if the	e point group, or all here is no group.
υ	Undo transform	Undo a point ation.	transformation oper-
Z	Aspect-ratio search	Switch aspect or off (page 4	ratio searching on -52

Group Transformation Options The transformations can be any combination of the following:

	Base Plane Transformation Commands		
Command	Parameters and Function		
CARTESIAN	$exp_u exp_v exp_w$ The points are moved to new positions defined by expressions for their (<i>u</i> , <i>v</i> and <i>w</i>) coordinates in terms of their old coordinates: U, V, R, TH and W. The copy number, (#COPY), can also be used.		
DISPLACE	du dv dw Displace points in the current viewing local coordinate system by adding (du , dv , dw) to their coordinates (u , v , w).		
MANGLE	<i>u1 v1 angle</i> Reflect point coordinates in line specified by one point (<i>u1</i> , <i>v1</i>) and an <i>angle</i> . The W coordinate of the points is not affected.		

Base Plane Transformation Commands (continued)		
Command	Parameters and Function	
MIRROR	u1 v1 u2 v2 Reflect point coordinates in line specified by its end points $(u1,v1)$, $(u2, v2)$. The W coordinate of the points is not affected.	
POLAR	$exp_r exp_{\theta} exp_w$ The points are moved to new positions defined by expressions for their (r , θ and w) coordinates in terms of their old coordinates: U, V, R, TH and W. The copy number, (#COPY), can also be used.	
PROJECT	<i>du dv dw ucentre vcentre wcentre uangle vangle wangle</i> Project the points in the direction (<i>du</i> , <i>dv</i> , <i>dw</i>) until they intersect the XY plane of a coordinate system specified by its origin (<i>ucentre</i> , <i>vcentre</i> , <i>wcentre</i>) and axis rotation angles (<i>uangle</i> , <i>vangle</i> , <i>wangle</i>).	
QUIT	End the sequence of transformations.	
ROTATE	<i>ucentre vcentre wcentre uangle vangle wangle</i> Rotate points by angles (<i>uangle, vangle, wangle</i>) around axes parallel to the local coordinate system and passing through the point (<i>ucentre, vcentre, wcentre</i>).	
SCALE	<i>ucentre vcentre factor</i> Scale points by multiplying the distance from local coordinate point (<i>ucentre</i> , <i>vcentre</i>) by <i>factor</i> . The W coordinate of the points is not affected.	

• Example: to rotate about the origin by angle 30 degrees about the W axis:

OP-TRANS > r 0 0 0 0 30

- Example: to scale the model to half its size: OP-TRANS > **s** 0 0 1/2
- Example: to move all points onto a circle, radius 10, leaving the azimuthal coordinates the same:

```
OP-TRANS > polar 10 atan2(v;u)*180/pi w
```

Base Plane Subdivision Mode

This mode is omitted if **DEFINE** is being used for conductor elements.

The volumes created by extruding the initial surface plane (base plane) are treated as super-elements. They are normally divided into smaller elements that are used for the actual finite element approximation. The division of the volume into elements is determined by the subdivision defined for the facet edges and the subdivision specified for each edge created by an extrusion operation.

A default subdivision of 1 is initially set for all facet edges. When all the edges have been updated, the program will reply that the subdivision is complete.

- If there are no polygonal facets, regular subdivision is assumed. In this case, when the subdivision is given for a single edge and that edge is one side of a quadrilateral facet, then the subdivision of the opposite edge will be set to the same value. The subdivision will be carried through the mesh until a triangular facet, the edge of the mesh or a facet already set in this operation is reached.
- In models with regular subdivision, triangular facets are meshed by mapping onto a quadrilateral with 2 corners coincident. Therefore triangles must have 2 sides with the same number of subdivisions. The program can check whether this rule has been satisfied using menu item **Check for hex meshing** or cursor hit **C**. The check is also made if Subdivison Mode is left using cursor hit **Q**.
- If there are polygonal facets, the subdivision of each edge must be set individually unless all edges are set to the same subdivision.

The subdivision of each edge is uniform, unless the edge is a quadratic line with the mid-point not at the geometric mid-point of the line. In that case the elements at the end of the edge closer to the mid-point will be smaller than those at the other end.

- *In keyboard mode* a facet is selected for subdivision by positioning the cross-hairs just inside a facet, close to the edge to be divided and pressing the **<SPACE>** bar. The program prompts for the number of subdivisions. Subdivisions can also be set globally, i.e. all edges set to the same subdivision number, using the **G** cursor hit.
- In menu mode the subdivision can be set first using menu item **Set sub**division and can then be applied to a single edge or globally to the entire mesh.

Subdivisions of edges can be changed any number of times, until the required pattern is achieved.

Additional facets can be defined by returning to Facet Definition Mode with menu item **Return to Facets** or menu hit **F**. If this facility is used it is essential to ensure that the mesh subdivision is complete before finally leaving Facet Subdivision Mode. Menu item **Go to Extrusions** or cursor hit **Q** leaves Facet Subdivision Mode and proceeds to Extrusion Mode. It is necessary to make sure that the extrusion direction required has been set using menu item **Change view** or cursor hit **V** before leaving the Subdivision Mode.

I

Facet Subdivision Menu and Cursor Hits

Facet Subdivision Menu and Cursor Hits				
Cursor hit	Menu item	Function		
<space></space>	Apply to line	Select closest edge for its subdivi-		
		sion to be set.		
A		Abort the DEFINE command.		
C	Check for hex meshing	If there are no polygons, check that		
		the subdivisions obey the rules for		
		meshing with quadrilaterals/hexa-		
		hedra.		
F	Return to Facets	Return to Facet Definition Mode		
		to define additional facets.		
G	Apply globally	Select all edges for their subdivi-		
		sions to be set to the same value.		
н		Display menu help message		
0	Co to Entruciona	explaining all the cursor options.		
Q	GO LO EXCRUSIONS	Leave the Facet Subdivision Mode		
		Mode If there:	are no polygons the	
		subdivisions ar	e checked for quad-	
		rilateral/hexahe	edral meshing.	
R	Re-draw picture	Reconstruct the	e display. The pro-	
		gram requests a new size. Valid		
		replies are:		
		4 numeric val-	umin, umax, vmin,	
		ues	vmax	
			Default values are	
			the current set-	
			ungs.	
		CURSUR	Select diagonally	
			the display area	
			with <space></space>	
			cursor hits.	
		RESTORE	Return to previous	
			screen size.	
		BOUND	Use bounding rec-	
			tangle of geome-	
			try.	
		<return></return>	Reconstruct at the same size.	

Full details of the cursor commands are given in the following section.

Facet Subdivision Menu and Cursor Hits (continued)				
Cursor hit	Menu item	Function		
v	Change view	Change the UVW coordinate sys- tem. This affects the view and the extrusion direction.		
x		Leave the Facet Subdivision Mode and proceed to the Extrusions Mode. No checking is done.		
Z	Aspect-ratio search	Switch aspect ratio searching on or off (page 4-52).		

Extrusions Mode

Once the initial surface set of facets has been defined and subdivisions assigned to the edges, the program moves on to applying extrusion operations to the set of facets. There must be at least one extrusion operation applied to the set of facets, but many others may be needed to define the complete model. It is also possible to add more extrusions to a completed mesh using the EXTEND command. The EXTEND command must be used to add the second and subsequent extrusions if the data is defined in menu mode (page 4-112).

- In keyboard mode a single line command defines each extrusion. It consists of two compulsory numeric values with optional keywords. The first numeric value is interpreted as the W-coordinate for the points at the top of the new extrusion layer. The second numeric value gives the number of subdivisions between the planes. The keywords LINEAR, QUADRATIC, GLOBAL and RELATIVE specify the type of extrusion. The keyword NOEDIT indicates that no editing is required for points on the new mesh plane(s). The keyword SUBDIVISION request the program to return to Facet Subdivision Mode and ignore all other items on the command line. SUBDIVISION is only available before the first extrusion has been created.
- In menu mode a LINEAR or QUADRATIC extrusion can be chosen from a menu. The DialogBox shown in the section "Extrusion Dialog-Box" on page 4-72 allows the user to give the coordinate of the next plane, the number of subdivisions and to select GLOBAL or RELA-TIVE.

The values and options are interpreted as follows:

• **Coordinate of next plane.** The W-coordinate can be interpreted as GLOBAL or RELATIVE. When a GLOBAL coordinate is given for the new plane, the extrusion is formed by projecting the current facet set in the W direction until the W-directed lines intersect with the plane defined by the global value of W. The W-coordinate can also be inter-

preted as a **RELATIVE** move in the W direction, in which case the coordinate value is added to the W-coordinates of the points in the current plane.

• **Subdivision.** This specifies the number of layers of elements there will be between the two planes. This can be adjusted for the whole layer or for individual W-directed lines using the MODIFY command (page 4-133).

The subdivision is ignored when DEFINE is being used for conductor elements.

• Linear or Quadratic. The W-directed lines can be LINEAR or QUAD-RATIC. The lines are created initially linear, i.e. straight, but if QUAD-RATIC is selected, a mid-extrusion plane of points is also created, at the geometric mid-points of the lines. This means that QUADRATIC lines can be changed to be curved, or to have non-uniform subdivision by moving the points on the mid-extrusion plane away from the geometric mid-points of the lines. The mid-points should be between the ¹/₄ and ³/₄ points along the length of the extrusion.

Unless **NOEDIT** has been requested, the coordinate values of points in the new plane, and in the mid-extrusion plane if **QUADRATIC** is selected, can be changed after the layer has been created. In some cases it is better to make the new plane(s) exactly the same as the current plane by using a **RELATIVE** coordinate of zero, and subsequently modifying the coordinates to the points using the Point Movement cursor options which are presented next. In other cases, it can be better to request **NOEDIT** and subsequently use the **MODIFY** (page 4-133) command to move the points.

For editing the point coordinates, the program draws a picture of the new plane and invites points to be moved individually or in groups. Individual points are selected using menu item **Move point** or cursor hit **<space>** and can be repositioned using the same options as offered for the base plane. Points can be grouped using menu item **Select/de-select point** or cursor hit **K** and transformed using menu item **Transform points** or cursor hit **T**. If no points have been grouped all the points of the plane will be transformed. The transformations are similar to those available in Facet Group Operations Mode (page 4-64).

The original coordinates are stored for all points moved or transformed so that they can be put back using the menu item Undo move or transform or cursor hit U.

 In keyboard mode, after the extrusion has been created and all necessary points have been moved, cursor hit Q moves on to the next extrusion. The user is given the option of creating another extrusion or not (valid replies are YES or NO). After a NO response, the program moves on to Material Definition Mode, or, when DEFINE is being use for conductor elements, copies the data into the conductor database for modification with the CONDUCTOR sub-commands (page 4-24).

• In menu mode, after the extrusion has been created and all necessary points have been moved, menu item Finish editing moves on to Material Definition Mode, or, when DEFINE is being use for conductor elements, copies the data into the conductor database for modification with the CONDUCTOR sub-commands (page 4-24). Further extrusions can be defined using the EXTEND command (page 4-112).

Full details of the extrusion command line, the point movement cursor commands and the transformation options are given in the following sections.

Extrusion	
Command L	ines.

Extrusion Command Lines			
Two compulsory numeric values:			
first_value	W coordinate (GLOBAL or RELATIVE) of next		
	plane.		
second_value	Number of subdivisions in the layer.		
Optional keywords:			
LINEAR (default) or	W-directed lines LINEAR (no 'Mid-extrusion		
QUADRATIC	Plane'), or QUADRATIC (with 'Mid-extrusion		
	plane').		
GLOBAL (default) or	W coordinate is GLOBAL or RELATIVE to pre-		
RELATIVE	vious plane.		
NOEDIT	No point movements are required on the new		
	plane(s).		
SUBDIVISION	Return to Facet Subdivision Mode (only availa-		
	ble before the first extrusion).		

• Example: to extrude to W=10 with 3 subdivisions and quadratic Wdirected lines:

OP-EXTRUDE > 10 3 global quadratic

• Example: to extrude by adding 5 to the W coordinates of the current plane with 4 subdivisions and linear W-directed lines; no point movements are necessary on the new plane:

OP-EXTRUDE > relative 5 4 linear noedit

Extrusion DialogBox In the DialogBox, the values of coordinate and number of elements must be supplied. **Global** and **Relative** are options.



Point Selection Menu and Cursor Hits

Point Selection Menu and Cursor Hits				
Cursor hit	Menu Item	Function		
<space></space>	Move point	Select point nearest cursor to be moved. It can be repositioned using the Point Repositioning Mode cursor hits (page 4-75).		
H		Display menu help message explaining all the cursor options. (More help available after point selection.)		
K	Select/de-select point	Select point nearest the cursor to be transformed. Repeating K for a selected point de-selects it.		
Q	Finish Editing	Leave this plane. If on a mid- extrusion plane, move on to the top plane of the new layer. Other- wise the program asks about the next extrusion (<i>keyboard mode</i>) or moves on to material defini- tions (<i>menu mode</i>).		

Point Selection Menu and Cursor Hits (continued)			
Cursor hit	Menu Item	Function	
R	Re-draw picture	Reconstruct the display. The pro- gram requests a new size. Valid replies are:	
		4 numeric values	<i>umin, umax, vmin, vmax</i> <i>vmax</i> Default values are the current settings.
		CURSOR	Select diagonally opposite corners of the display area with <space></space> cursor hits.
		RESTORE	Return to previous screen size.
		BOUND	Use bounding rec- tangle of geometry.
		<return></return>	Reconstruct at the same size.
Т	Transform points	Define gener for the points to select poir be transform tion options a "Plane Trans on page 4-77	The formation of the second s
υ	Undo move or transform	Undo the last move or transform operation.	
v	Change view	Change the UVW coordinate sys- tem. This affects the view and the extrusion direction.	
Z	Aspect-ratio search	Switch aspect ratio searching on or off (page 4-52).	
The Point Repositioning Mode Menu and Cursor Hits

Point Repositioning Mode Menu and Cursor Hits			
Cursor hit	Menu item	Function	
<space></space>	At mouse	Reposition the point at the cursor cross hair position.	
C	Give U, V, W	Switch to input from keyboard in cartesian coordinates UVW. Coordinates should be entered in free format. Default values of U, V and W are the values prior to the move. They can be accessed via the variables #1 , #2 and #3 . Type Q to leave point in its present position.	
E	Remove C_line	Erase construction line closest to cursor cross hair.	
н		Display menu help message explaining all the cursor options.	
I	Enter C_lines	Input construction line specifications.	
L	List C_lines	List construction line specifica- tions.	
N	On nearest C_line	Reposition the point on the near- est construction line with mini- mum normal distance. This only affects the U and V coordinates; W remains unchanged.	
P	Give R, Theta, W	Switch to input from keyboard in local polar coordinates $R\theta W$. Coordinates should be entered in free format. Default values of R, θ and W are the values prior to the move. They can be accessed via the variables #1 , #2 and #3 . Type Q to leave point in its present position.	
Q	Return without moving	Leave the point at its previous position.	

Cursor hit	Menu item	Function	
P	Pe-draw picture	Reconstruct the display. The	
ĸ	Re-diaw picture	reconstruct	ule display. The
		Volid replice	
		vand replies are:	
		4 numeric	umin, umax, vmin,
		values	vmax
			Default values are
			the current set-
			tings.
		CURSOR	Select diagonally
			opposite corners
			of the display area
			with <space></space>
			cursor hits.
		RESTORE	Return to previous
			screen size.
		BOUND	Use bounding rec-
			tangle of geome-
			try.
		<return></return>	Reconstruct at the
			same size.
Т	Show coordinates	Type the coo	rdinates of the point
		closest to the	e cross-hairs, and
		select it for re	epositioning instead
		of the point a	already selected.
		The coordina	ates are given in car-
		tesian (UVW	<i>I</i>) and polar (R θ W)
		coordinates.	
Х	At C_line intersection	Reposition th	he point at the clos-
		est construct	ion line intersection
		or end. This	only affects the U
		and V coord	inates; W remains
		unchanged.	
Z	Aspect-ratio search	Switch aspec	t ratio searching on
		or off (page	4-52).

Plane
Transformation
Options

The transformations can be any combination of the following:

	Plane Transformation Commands		
Command	Parameters and Function		
CARTESIAN	<i>exp_u exp_v exp_w</i> The points are moved to new positions defined by expressions for their (<i>u</i> , <i>v</i> and <i>w</i>) coordinates in terms of their old coordinates: U, V, R, TH and W. The copy number, (#COPY), can also be used.		
DISPLACE	du dv dw Displace points in the current viewing local coordinate system by adding (du , dv , dw) to their coordinates (U, V, W).		
POLAR	$exp_r exp_\theta exp_w$ The points are moved to new positions defined by expressions for their (r , θ and w) coordinates in terms of their old coordinates: U, V, R, TH and W. The copy number, (#COPY), can also be used.		
PROJECT	<i>du dv dw ucentre vcentre wcentre uangle vangle wangle</i> Project the points in the direction (<i>du</i> , <i>dv</i> , <i>dw</i>) until they intersect the XY plane of a coordinate system specified by its origin (<i>ucentre</i> , <i>vcentre</i> , <i>wcentre</i>) and axis rotation angles (<i>uangle</i> , <i>vangle</i> , <i>wangle</i>).		
QUIT	End the sequence of transformations.		
ROTATE	<i>ucentre vcentre wcentre uangle vangle wangle</i> Rotate points by angles (<i>uangle</i> , <i>vangle</i> , <i>wangle</i>) around axes parallel to the local coordinate system and passing through the point (<i>ucentre</i> , <i>vcentre</i> , <i>wcentre</i>).		
SCALE	<i>ucentre vcentre factor</i> Scale points by multiplying the distance from local coordinate point (<i>ucentre</i> , <i>vcentre</i>) by <i>factor</i> . The W coordinate of the points is not affected.		

In menu mode the transformation commands are offered as menu items and the parameters are supplied using parameter boxes.

• Example: to shift the selected points sideways by 5 units in the U direction:

OP-TRANS > displace 5 0 0

• Example: to rotate the plane by -30 degrees around a line parallel to the V axis and passing through W=10:

```
OP-TRANS > rotate 0 0 10 0 -30 0
```

• Example: to double the U and halve the V coordinates of the points: OP-TRANS > cartesian u*2 v/2 w

I

Material Definition Mode

The extrusion operations create a discretisation of space. It is now necessary to set the material and mesh properties within each volume. To do this, the program presents the user with each layer in turn and the user sets values which over-ride the default or current settings within each volume.

- In keyboard mode volumes are selected using the cursor. Single volumes are selected using cursor hit <space>. Multiple volumes are selected using K in each volume except the last. The last volume should be selected using <space>. After a <space> cursor hit the program prompts for a one line material property definition. A summary of the volume properties can be displayed in each volume using S. This gives the name, potential code and element type. A full list of all the properties of an individual volume can be obtained using L. Cursor hit Q moves on to the next layer and F finishes definition in all layers. After material definition the program moves on to Boundary Condition Definition Mode.
- In menu mode the menu item Select/de-select volume allows volumes to added or removed from a list. Menu item Select and define adds one last volume to the list and causes the program to display a DialogBox into which the material properties can be entered. Other menu items allow the properties to be summarized in all the volumes (Show) or to be Listed for an individual volume.

Material property definitions consist of two compulsory keywords and several options which add special properties or control setting of several volumes simultaneously.

The first compulsory keyword is the **material name**. This can be (almost) any character string of up to 8 characters, beginning with a letter. There are two pre-defined material names, AIR and NULL. AIR is for any non-conducting volume with a relative permeability and relative permittivity both equal to 1. Volumes with name NULL are omitted from the final mesh, enabling the creation of holes in the mesh to represent e.g. electrodes. (The second keyword must be omitted for material name NULL.) The definition of each material in terms of its permeability and, if necessary, conductivity is supplied by the MATERIALS command (see "The MATERIALS Command" on page 4-127), or during the commands which create the analysis data files.

The second compulsory keyword is the **potential type**. The following rules must be followed, depending on which analysis program will be used:

• TOSCA (magnetostatics)

REDUCED scalar potential *must* be used in a space where source currents are flowing. It is often easiest to make all the AIR use reduced sca-

I

lar potential but a model should not have only **REDUCED** potential volumes which would result in a zero solution.

TOTAL scalar potential *should* be used in non-AIR volumes. It is also possible to use reduced scalar potential in non-AIR volumes but this should only be used as a last resort, if it is not possible to specify a region where the total scalar potential would be single valued.

The total scalar potential becomes multi-valued if there exists a closed path, entirely in total scalar potential volumes, through which a net current flows. TOSCA breaks such a path by the use of automatic cuts, i.e. additional volumes which can have multiple values of potential on the surface. TOSCA gives these volumes additional labels, POTENTIAL_CUT*n*, so that they can be selected and displayed in the post processor.

If automatic cuts are disabled (page 4-167), it is necessary to use reduced scalar potential volumes or two different total scalar potential boundary conditions with one value inside the coil and another outside (see "Boundary Condition Definition Mode" on page 4-86). In this case the potential difference between the boundaries should exactly balance the current enclosed in the mesh (*S.I.* units):

$$\Delta \boldsymbol{\Psi} = - \int \mathbf{H} \cdot dl = \mathbf{I} \tag{4.2}$$

TOTAL scalar potential can also be used in any AIR volumes which do not contain source currents. This reduces the number of nodes at which the coil field has to be calculated and also improves the accuracy of the total field in situations where the fields from the coils and the magnetic materials almost cancel (shielding problems).

To achieve good answers, it is important that coil fields can be represented well by the finite element mesh on the interface between the reduced and total potentials. This implies that the best choice of reduced potential region is a simple shaped region that contains the coils and which has a surface as far as possible from the coil surface. It is not always possible to achieve this, especially when for example the coil is wrapped on an iron surface since in this case the interface must correspond to the coil surface. In such cases the discretisation of the interface should be increased to maintain accuracy and the choice of adaptive integration is essential.

Any VECTOR potential volumes in a TOSCA model will be treated as TOTAL scalar potential.

ELEKTRA

REDUCED vector potential must be used in a space where source currents are flowing. It is often easiest to make all the AIR use reduced vector potential but a model should not have only **REDUCED** potential volumes which would result in a zero solution.

Total vector potential (keywords TOTAL or VECTOR) must be used in conducting or non-conducting non-AIR volumes and can also be used in any AIR volumes which do not contain source currents. This reduces the number of nodes at which the coil field has to be calculated and also improves the accuracy of the total field in situations where the fields from the coils and the magnetic or conducting materials almost cancel.

To achieve good answers, it is important that coil fields can be represented well by the finite element mesh on the interface between the reduced and total vector potentials. This implies that the best choice of reduced potential region is a simple shaped region that contains the coils and which has a surface as far as possible from the coil surface. It is not always possible to achieve this, especially when for example the coil is wrapped on a material surface since in this case the interface must correspond to the coil surface. In such cases the discretisation of the interface should be increased to maintain accuracy and the choice of adaptive integration is essential.

CARMEN

REDUCED scalar potential *must* be used in a space where source currents are flowing. It is often easiest to make all the AIR use reduced scalar potential but a model should not have only **REDUCED** potential volumes.

TOTAL scalar potential *should* be used in non-conducting non-AIR volumes. It is also possible to use reduced scalar potential but this should only be used as a last resort, if it is not possible to specify a region where the total scalar potential would be single valued.

TOTAL scalar potential can also be used in any AIR volumes which do not contain source currents. This reduces the number of nodes at which the coil field has to be calculated and also improves the accuracy of the total field in situations where the fields from the coils and the magnetic materials almost cancel.

Magnetic VECTOR potential *must* be used in non-AIR volumes with non-zero conductivity. In some situations, it must also be used in adjacent AIR or other non-conducting volumes so that the total magnetic scalar potential in surrounding volumes is single valued.

In some models it is convenient to assign a current density to volumes of the mesh. These non-AIR volumes should be modelled with magnetic VECTOR potential and should have zero conductivity. The current density vector is described below.

The total magnetic scalar potential becomes multi-valued if there exists a closed path, entirely in total scalar potential volumes, through which a net current flows. In this context, an interface between vector potential and reduced scalar potential is an infinitely thin total potential volume, which might also include a closed path through which a net current flows. CARMEN breaks such a path by the use of automatic cuts, i.e. additional volumes which can have multiple values of potential on the surface. CARMEN gives these volumes additional labels, **POTENTIAL_CUT***n*, so that they can be selected and displayed in the post processor.

If automatic cuts are disabled (page 4-167), all such paths must be broken by the use of vector or two different total scalar potential boundary conditions with one value inside the coil and another outside (see "Boundary Condition Definition Mode" on page 4-86). In this case the potential difference between the boundaries should exactly balance the current enclosed in the mesh (*S.I.* units):

$$\Delta \Psi = - \int \mathbf{H} \cdot dl = \mathbf{I} \tag{4.3}$$

To achieve good answers, it is important that coil fields can be represented well by the finite element mesh on the interface between the reduced and other potentials. This implies that the best choice of reduced potential region is a simple shaped region that contains the coils and which has a surface as far as possible from the coil surface. It is not always possible to achieve this, especially when for example the coil is wrapped on an iron surface since in this case the interface must correspond to the coil surface. In such cases the discretisation of the interface should be increased to maintain accuracy and the choice of adaptive integration is essential.

• SCALA and TOSCA (current flow and electrostatics)

All volumes are treated as if they use **TOTAL** scalar potential.

• SOPRANO

All volumes must use magnetic VECTOR potential. The program overrides the assignment of other potential types.

More information on the use of the different potentials is given in the user guides for the analysis programs.

Optional keywords defining properties are element types (LINEAR, QUADRATIC), one scalar property (SCALAR), three vector properties (VECTOR, CURRENT, VELOCITY) and a lamination PACKINGFAC-TOR. The vector properties can be defined as expressions in terms of the coordinates (X, Y and Z) to allow spatial variation.

- LINEAR and QUADRATIC define the element types to be used. LIN-EAR elements are 8-noded hexahedra or 4 noded tetrahedra; QUAD-RATIC elements are 20-noded isoparametric hexahedra or 10-noded isoparametric tetrahedra. Both element types can be used in one model. (Only linear elements can be used in ELEKTRA and SOPRANO.)
- In electrostatic problems (SCALA and TOSCA), the volume electric charge density is specified by SCALAR followed by one numeric value.

I

- In problems with permanent magnets the coercive force is specified by the material properties (linear materials) or the BH curve (non-linear materials), but the direction is specified here by VECTOR followed by 3 Euler angles (page 2-34) which define a local coordinate system for the volume. The permanent magnetic field is in the local negative Z direction.
- In problems with anisotropic materials the local coordinate system for the material is defined by VECTOR followed by 3 expressions representing the Euler angles (page 2-34). If the material is laminated, (TOSCA only) the packing factor is specified by PACKFACTOR followed by one numeric value which modifies the permeability in the directions normal and parallel to the laminations (the local XY plane). If the material is to be specified by multiple properties, only the VEC-TOR property is required here to define the local coordinate system for the material. In either case, anisotropy must be switched on in the MATERIAL sub-command of SOLVER (page 4-150).
- In eddy current problems (CARMEN and ELEKTRA), an assigned current density can be specified by CURRENT followed by 3 expressions representing the x, y and z-components of the current density. This only applies to non-conducting VECTOR potential volumes.
- In motion induced eddy current problems (ELEKTRA-VL), the linear velocity is specified by VELOCITY followed by 3 expression representing the x, y and z-components of velocity in units of length sec⁻¹. This can only be defined for VECTOR potential volumes. Rotational motion is given by SCALAR followed by the angular velocity in rpm. The rotation is always around the global Z axis.

The three values associated with CURRENT, VECTOR and VELOCITY can be given as expressions in terms of X, Y and Z so that vectors and properties which vary throughout the volumes can be specified. The computed directions can be viewed using the DISPLAY command (page 4-96).

Optional keywords for setting additional volumes with the same definition are ALL, FROM, TO and KEEP. ALL indicates that all the volumes in the layer(s) should be set the same. Both FROM and TO should be followed by numeric values specifying a layer number to specify a range of layers to be set the same. If either is omitted, the current layer number is assumed. The numeric value after TO can be specified as * to indicate the top layer. KEEP specifies that the current selection of volumes should be kept, following the definition of materials, so that another definition can be given. This can be used when setting corresponding volumes in a non-contiguous set of layers.

The material names, potential types and element types are stored as labels on each volume and can be used to select parts of the model in the DIS-PLAY command (page 4-96).

Full details of the cursor commands and material definitions are given in the following sections.

Material Definition Mode Menu and Cursor Hits

Material Definition Menu and Cursor Hits			
Cursor hit	Menu item	Function	
<space></space>	Select and define	Select the volume containing the cross-hairs to give a mate- rial definition for this volume	
		and any selected with K .	
F	Finish	End the volume material defi- nition mode and move on to Boundary Condition Mode.	
н		Display menu help message explaining all the cursor options. (More help available after volume selection.)	
ĸ	Select/de-select volume	Keep the volume containing the cross-hairs until a space > is used. Repeating K for a selected volume, de- selects the volume.	
L	List properties	List all the properties of the volume containing the cross-hairs.	
Q		Move on to the next extrusion layer or, if at the last layer move on to Boundary Condi- tion Mode.	

Material Definition Menu and Cursor Hits (continued)			
Cursor hit	Menu item	Function	
R	Re-draw picture	Reconstruct t	he display. The
		program requ	lests a new size.
		Valid replies	are:
		4 numeric	umin, umax,
		values	vmin, vmax
			Default values
			are the current
			settings.
		CURSOR	Select diago-
			nally opposite
			corners of the
			display area
			with <space></space>
			cursor hits.
		RESTORE	Return to previ-
			ous screen size.
		BOUND	Use bounding
			rectangle of
			geometry.
		<return></return>	Reconstruct at
			the same size.
S	Show volumes	Display the n	naterial names,
		potential code	es and element
		types.	
v	Change view	Change the U	VW coordinate
		system. This	affects the view
		and the extru	sion direction.
Z	Aspect-ratio search	Switch aspec	t ratio searching
		on or off (pag	ge 4-52).

The single line definitions of material and other volume properties consist of the following items:

Material Definitions			
Two compulsory keywords:			
<i>first_keyword</i> or QUIT Material name. Use AIR for air, NULL to omivolume from mesh or a material name. QUIT abandons the currently selected volumes.			
second_keyword	Potential type: REDUCED, VECTOR or TOTAL. It should not be given for material NULL.		

Material Definition Command Line

I

Material Definitions (continued)			
Optional keywords – element types:			
LINEAR (default) or	LINEAR elements, or QUADRATIC elements.		
QUADRATIC			
Optional keywords – vo	blume properties:		
PACK value	Sets packing factor for volume.		
SCALAR value	Sets charge density (electrostatics only).		
	Sets angular velocity in rotating volumes [rpm] (ELEKTRA-VL).		
CURRENT jx jy jz	Sets assigned current density.		
VECTOR t p s	Sets local coordinate system Euler angles of vol- ume for permanent magnets or anisotropic materials.		
VELOCITY vx vy vz	Sets linear velocity [length unit sec ⁻¹] in moving volumes (ELEKTRA-VL).		
Optional keywords – se	tting multiple volumes:		
ALL	Sets all volumes in layer(s).		
FROM value	Sets all layers from layer number given though		
	to the current layer or the layer number given with TO.		
KEEP	Keeps current selection of volumes for another material definition.		
TO value	Sets all layers from current layer or the layer number given with FROM though to the layer number given. The value may be * to indicate the top layer.		

• Example: to set selected volumes to material name IRON in layers 3, 4,..., 7 and 8:

OP-MATERIALS > iron total from 3 to 8

• Example: to set all volumes in the layer to material name ALCOMAX with vector direction set:

OP-MATERIALS > alcomax total all vect 90 90 0

Material Definition DialogBox In the DialogBox, the default properties correspond to the last volume selected. A material name must be supplied. Unless the material is NULL, one of the potential types must be selected. All the other items are optional and should only be assigned values if needed. The Local XYZ options allow the VECTOR property to be set to appropriate Euler angles to swap the coordinate axes. If another direction is needed, the Euler angles should be typed into the Other vector box.

Material Definition				
Material name AIR				
Potential type:	Element type:			
Total	Linear			
Reduced	Quadratic			
Vector				
Options:				
Jx, Jy, Jz				
Vx, Vy, Vz				
Scalar: Charge Density or H	Rotational Velocity			
Scalar				
Packing factor				
Material orientation:				
🗌 Local XYZ=XYZ 🔲 Local XYZ=YZX 📄 Local XYZ=ZXY				
Other vector				
Other volumes and layers:				
From To All volumes				
Accept Keep	Help Quit			

Boundary Condition Definition Mode

Boundary conditions can be set on any surface of any volume in the mesh. In order to achieve this the program presents the facets in separate groups:

In keyboard mode there are 4 groups of facets: first the facets on the base plane; second the planes between the first and the last; third the final plane of the mesh; and fourth the extrusion facets normal to the planes, one layer at a time. Facets are selected using the cursor. Single facets are selected using **<space>**. Multiple facets are selected using **K** for each facet except the last. The last facet should be selected using **<space>**. After a **<space>** cursor hit the program prompts for a one line boundary condition definition. Alternatively all the free surfaces of the mesh (all planes and layers) can be set to the same boundary condition using **G**. Free surfaces are those which are on the outside of the mesh or adjacent to a volume with material name **NULL**. A full list of all the conditions on an individual facet can be obtained using **L**. Cursor hit **Q** moves on to the next plane or layer and **S** or **F** finishes definition in all planes or layers. After boundary condition definitions the **DEFINE** command is complete and the program expects another top-level command.

In menu mode there are 3 groups of facets: first the facets on the base plane; second the top plane of the mesh; and third the extrusion facets normal to the planes. The menu item **Select/de-select facet** allows facets to be

I

added or removed from a list. Menu item **Select and define** adds one last facet to the list and causes the program to display a DialogBox into which the boundary conditions can be entered. The menu item **List conditions** allows the boundary conditions to be listed for an individual facet.

Boundary conditions can be imposed in seven ways:

• by restricting the MAGNETIC or ELECTRIC field to be NORMAL or TANGENTIAL. This sets combinations of the potentials and derivatives as appropriate for the type of problem and volume concerned:

Normal and Tangential Boundary Conditions			
Normal	Tangential	Normal	Tangential
Magnetic	Magnetic	Electric	Electric
	TOSCA Magne	etic Field Problen	n
$\Psi = 0$	$\frac{\partial \Psi}{\partial n} = 0$	not allowed	not allowed
S	CALA or TOSCA	Electric Field Pro	oblem
not allowed	not allowed	V = 0	$\frac{\partial V}{\partial n} = 0$
	CARMEN Scala	r potential Volun	nes
$\Psi = 0$	$\frac{\partial \Psi}{\partial n} = 0$	$\frac{\partial \Psi}{\partial n} = 0$	$\Psi = 0$
CARMEN/ELEKTRA/SOPRANO Vector potential Volumes			
$\mathbf{A}\cdot\mathbf{n} = 0,$	$\mathbf{A} \times \mathbf{n} = 0$	$\mathbf{A} \times \mathbf{n} = 0$	$\mathbf{A}\cdot\mathbf{n} = 0,$
$\frac{\partial V}{\partial n} = 0$	V = 0	V = 0	$\frac{\partial V}{\partial n} = 0$

In the above table ψ indicates the total or reduced magnetic scalar potential, **A** the magnetic vector potential and *V* the voltage.

If necessary a potential boundary condition can be assigned in addition to the normal or tangential conditions in order to override the zero values of scalar potential.

- by setting values of magnetic scalar POTENTIAL, the VOLTAGE or components of the magnetic vector potential (AX, AY and AZ). For SOPRANO, the incident values of magnetic vector potential and voltage can be specified instead (INAX, INAY, INAZ and INVO).
- by setting the normal derivative of the magnetic scalar potential (DERIVATIVE) or a mixed magnetic scalar potential condition in TOSCA (PMIX).
- by imposing a **RADIATION** condition (SOPRANO only).

- by imposing a PEC (perfect conductor) boundary condition (CAR-MEN, ELEKTRA and SOPRANO only). This is functionally equivalent to NORMAL ELECTRIC but allows boundaries to be distinguished in the post processor.
- by imposing a SLIP condition (CARMEN only) to identify the interface between the stator and rotor.
- by setting a **SYMMETRY** or periodicity condition (TOSCA and SCALA only).

If potential or derivative boundary conditions are used, a facet can have up to 4 potential boundary conditions – one for the scalar potential and one for each component of the vector potential.

Boundary condition definitions consist of a compulsory keyword and up to two values followed by several options which control setting of several facets simultaneously or clearing boundary conditions.

The compulsory keyword is the boundary condition name. This can be one of NORMAL, TANGENTIAL, POTENTIAL, VOLTAGE, AX, AY, AZ, INAX, INAY, INAZ, INVO, DERIVATIVE, PMIX, RADIATION, PEC, SLIP or SYMMETRY. The boundary condition name is assigned to the facet as a label. If normal or tangential boundary conditions are applied, the following labels are assigned to the facet as appropriate: ELECTRIC, MAGNETIC, NORMELEC, NORMMAGN, TANGELEC, TANG-MAGN.

CLEAR followed by a boundary condition name clears that condition from the facet(s).

Boundary conditions have the following effects:

In magnetic scalar potential volumes, setting a constant POTENTIAL condition specifies that the tangential components of the field are to be zero. Non-zero total scalar POTENTIAL conditions can be used to impose an external field (i.e. an m.m.f. across the model) or to balance enclosed currents in multiply connected geometries. Reduced scalar POTENTIAL boundary conditions can only have the value zero.

A constant **DERIVATIVE** specifies the value of the normal field (TOSCA only). The zero **DERIVATIVE** condition is the natural condition of the finite element mesh and need not be assigned explicitly.

 In electric scalar potential volumes (electrostatics or current flow), setting a constant VOLTAGE condition specifies that the tangential components of the field are to be zero. Non-zero total VOLTAGE conditions can be used to impose an external field (i.e. a potential difference across the model).

A constant **DERIVATIVE** specifies the value of the normal field. The zero **DERIVATIVE** condition is the natural condition of the finite element mesh and need not be assigned explicitly.

• In vector potential volumes, setting AX, AY or AZ to zero implies that the electric field in that direction is also zero. Only components tangential to the surface are affected. The constant electric VOLTAGE boundary condition should also be set on surfaces where the tangential components of electric field are zero. This constant should in general be zero, unless the voltage is providing the driving field.

A second set of boundary conditions (INAX, INAY, INAZ) set the characteristics of the incident field for SOPRANO and also apply the RADI-ATION condition

- Periodic or SYMMETRY boundaries (TOSCA and SCALA only) specify that the potentials on one surface are equal to the values on another surface (with or without a change of sign). The facets forming such surfaces are labelled here with the boundary condition SYMME-TRY and are joined together in the PERIODICITY sub-command of the SOLVERS command (page 4-163) by the specification of transformations which map one set of boundary nodes onto the other.
- SLIP boundaries (CARMEN only) identify the interface between the stationary and rotating parts of a rotating machine. The volumes on both sides of a slip surface should be modelled using TOTAL scalar potential. It is often more convenient to apply the SLIP boundary condition using the SLIP command (page 4-149).

In some circumstances it may be necessary to specify the boundary condition values as functions of the node coordinates. The values of conditions POTENTIAL, VOLTAGE, AX, AY and AZ can be specified as expressions in terms of X, Y and Z. User variables and system constants (PI, MUO, EPSILONO and C) can be used in the expressions. The program assigns names for boundary condition expressions and these names can be used as labels in the DISPLAY command (page 4-96).

For analysis with CARMEN, ELEKTRA and SOPRANO/SS, non-zero potential boundary conditions (POTENTIAL, VOLTAGE, AX, AY, AZ, INAX, INAY or INAZ) can be given a drive label so that the associated time-function or phase angle can be assigned by the SOLVER command (page 4-150).

Further information on boundary conditions is given in the User Guides for CARMEN, ELEKTRA, SCALA, SOPRANO and TOSCA and in the OPERA-3d Training Course.

The boundary condition names are stored as labels on each facet and can be used to select parts to the model in the DISPLAY command (page 4-96).

Optional keywords for setting additional facets with the same definition are ALL, FROM and TO. ALL indicates that all the facets in the plane(s) or layer(s) should be set the same. Both FROM and TO should be followed by a numeric value specifying a plane or layer number to specify a range of planes or layers to be set the same. If either is omitted, the current plane or layer number is assumed. The numeric value after TO can be specified as * to indicate the highest numbered plane or layer.

Full details of the menu items and cursor commands and boundary condition definitions are given in the following sections.

Boundary Condition Mode Menu and Cursor Hits			
Cursor hit	Menu item	Function	
<space></space>	Select and define	Select the facet nearest the cross- hairs to give a boundary condi- tion definition for this facet and any selected with \mathbf{K} .	
F		End the boundary condition def- inition mode (extrusion layers only).	
н		Display menu help message explaining all the cursor options. (More help available after facet selection.)	
ĸ	Select/de-select facet	Keep the facet containing the cross-hairs until a <space></space> is used. Repeating K for a selected facet, de-selects the facet.	
L	List conditions	List all the boundary condition on the facet containing the cross- hairs.	
Q	Finish	Move on to the next plane or layer or, if at the last layer, leave Boundary Condition Mode.	

Boundary Condition Definition Mode Menu and Cursor Hits

Boundary Condition Mode Menu and Cursor Hits (continued)				
Cursor hit	Menu item	Function	Function	
R	Re-draw picture	Reconstruct gram request replies are:	the display. The pro- ts a new size. Valid	
		4 numeric values	<i>umin, umax, vmin, vmax</i> Default values are the current set-tings.	
		CURSOR	Select diagonally opposite corners of the display area with <space></space> cursor hits.	
		RESTORE	Return to previous screen size.	
		BOUND	Use bounding rec- tangle of geome- try.	
		<return></return>	Reconstruct at the same size.	
S		Skip to the la only).	Skip to the last plane (planes only).	
v	Change view	Change the U system. This the extrusior	Change the UVW coordinate system. This affects the view and the extrusion direction.	
Z	Aspect-ratio search	Switch aspect ratio searching on or off (page 4-52).		

Boundary Condition Command Line Definitions The single line commands for boundary condition definition consist of the following items:

Boundary Condition Definitions	
One compulsory keyword and values - one of the following:	
AX expression label	The X component of the vector potential.
AY expression label	The Y component of the vector potential.
AZ expression label	The Z component of the vector potential.
CLEAR comp	Clear boundary conditions with <i>comp</i> equal to POTENTIAL, AX, AY or AZ. If <i>comp</i> is omitted all boundary conditions are cleared.
DERIVATIVE value	The normal derivative of scalar potential.

Boundary Conditi	on Definitions (continued)
INAX value label	The incident vector potential, X compo- nent (SOPRANO).
INAY value label	The incident vector potential, Y compo- nent (SOPRANO).
INAZ value label	The incident vector potential, Z compo- nent (SOPRANO).
NORMAL field	Boundary conditions which restrict <i>field</i> to be normal to the facet. <i>Field</i> can be MAGNETIC or ELECTRIC.
PEC	Perfect conductor boundary condition (CARMEN, ELEKTRA and SOPRANO only).
PMIX value1 value2	Mixed scalar potential boundary condi-
	tion in TOSCA (ϕ + <i>value1</i> $\frac{\partial \phi}{\partial n}$ = <i>value2</i>)
POTENTIAL expression label	The magnetic scalar potential.
QUIT	Abandons the current selection of faces.
RADIATION	Radiation boundary condition (SOPRANO).
SLIP	Slip surface periodic boundary condition for a rotating machine.
SYMMETRY	Periodic boundary condition. Facets are connected by rotation and displacement operations defined in the SOLVER com- mand (page 4-150).
TANGENTIAL field	Boundary conditions which restrict <i>field</i> to be tangential to the facet. <i>Field</i> can be MAGNETIC or ELECTRIC.
VOLTAGE expression label	The electric scalar potential.
Optional keywords - setting mul	tiple facets:
ALL	Sets all facets in plane(s) or layer(s).
FROM value	Sets all planes or layers from number given though to the current plane or layer or the number give with TO.
KEEP	Keeps current selection of facets for another boundary condition definition.
TO value	Sets all plane or layers from current plane or layer or the number given with FROM, through to the number given. The value may be * to indicate the top plane or layer.

• Example: to set selected facets to zero scalar potential: OP-B/C > **pote 0**

Boundary Condition DialogBox The default settings correspond to the last facet selected.

The boundary condition can be specified in one of the following ways:

- Potentials: the option should be selected and a value given. For steadystate ac or transient analysis, a label can also be given to enable drive information to be associated with the boundary condition.
- Other boundary condition option buttons: no value is necessary.
- CLEAR: the condition to be cleared can be given in the value box.

Other types of boundary condition can be given by not selecting any of the options but giving a keyboard style boundary condition command line in the value box.

	Boundary Condition	3
Condition name:		
🗌 Magnetic Scalar	🗌 Normal Magnetic	🗌 Tangential Magnetic
🗌 Voltage	🗌 Normal Electric	Tangential Electric
🗌 Total Ax	🗌 Total Ay	🗌 Total Az
🗌 Incident Ax	🗌 Incident Ay	🗌 Incident Az
🗌 Incident Voltage	Perfect Conductor	Radiation
🗌 Normal Derivative	Mixed Derivative	
Symmetry	🗌 Slip surface	Clear
Value	Label/2	ind value
Other planes or layer:	s:	
From	То	All facets
Accept Ke	ep Help	Quit

The **DEVICE** Command

Menu Route:

OPTIONS↓ Graphics output

Command Line Parameters:

Command DEVICE No Parameters

There are two different graphics implementations of the software:

- Windows (available on Windows98 and Windows NT4 SP6, Windows 2000 and Windows XP operating systems)
- X-lib (available on UNIX operating systems).

The **DEVICE** command is only available with X-lib graphics.

Four Graphics Options are available when the program is started and two of them can be reselected using the DEVICE command. The options are:

Option	Start-up	DEVICE command	Meaning
SCREEN	*	*	graphics displayed on the screen
FILE	*		all graphics commands stored in one file
BOTH	*	*	graphics on the screen and in a file
NONE	*		no graphics, except that the DUMP command can still be used to create picture files of specific pages.

If the program is started with no screen graphics (FILE or NONE), the GUI is not available and cannot be made available by the DEVICE command.

The initial specification of the graphics option can be stored in an environment variable (UNIX) called VFGRAPHICS. A valid value of VFGRAPHICS eliminates the initialization prompt.

Variable	Meaning
VFWINDOWW	the initial window width in pixels
VFWINDOWH	the initial window height in pixels
VFINV	If this is set to INVERT , the initial setting of text and background colours will be black on white instead of the default of white on black.

Other environment variables which affect the software on UNIX are:

On Windows systems, similar functionality can be obtained using the Options \rightarrow Graphics Window Preferences menu item on the OPERA Console window.

Picture files can be read by the PICOUT program which is supplied with UNIX versions of the software and is described with the DUMP command.

Picture files can also be created using the DUMP command ("The DUMP Command" on page 4-103).

The **DISPLAY** Command

Menu Route

DISPLAY↓

Display command ... refresh display

Command Line Parameters

Command	DISPLAY			
Parameter	Default	Function		
SIZE	10	Size of coordinate space to be dis-		
		played. The space extends from		
		(XORIGIN, YOR	IGIN, ZORIGIN)	
		by SIZE in each direction.		
XEYE	0	X-coordinate of ey	ve position.	
YEYE	0	Y-coordinate of eye position.		
ZEYE	100	Z-coordinate of eye position.		
PERSPECTIVE	NO	Perspective view s	witch.	
		NO	Orthogonal pro-	
			jection.	
		YES	Perspective pro-	
			jection.	
ROTX	0	Rotation angle abo	out X-axis to	
		change eye positio	on.	
ROTY	0	Rotation angle abo	out Y-axis to	
		change eye positio	on.	
ROTATE	0	Rotation angle abo	out viewing direc-	
		tion.		
ELEMENI	NO	Element display switch.		
		NO	No element sub-	
			division.	
		SURFACE	Subdivision on	
			volume surfaces.	
		VOLUME	Subdivision	
MEOU			within volumes.	
MESH	ALL	Mesh number, ALL or NONE.		
TYPE	VOLUME	Type(s) of entities	to be displayed.	
		ALL or SAME.	I, LINE, POINT,	
LABEL	NOTAIR	Label(s) on entities.		
L1	1	First layer to be displayed.		

Command	DISPLAY (contin	nued)	
Parameter	Default	Function	
L2	*	Last layer to be di top layer.	splayed. * means
COIL	YES	Conductor display	switch.
		NO	Conductors not displayed.
		YES	Conductors dis- played.
C1	1	First conductor to	be displayed.
C2	*	Last conductor to means highest nur	be displayed. * nbered conductor.
XORIGIN	0	X-coordinate at ce	entre of display.
YORIGIN	0	Y-coordinate at ce	entre of display.
ZORIGIN	0	Z-coordinate at centre of display.	
HIDDEN	NO	Hidden surfaces removed sw	
		FULL	Colour-fill dis-
			play of visible
			surfaces (slower
			but more reliable
			algorithm).
		NO	Wire frame dis-
			play of all sur-
			faces.
		YES	Colour-fill dis-
			play of visible
			surfaces
VECTORS	CONDUCTORS	Vector display sw	itch.
		CONDUCTORS	Vectors show
			current direction
			on conductors.
		CURRENT	Vectors show
			current density
			direction.
		MATERIAL	Vectors show
			material orienta-
			tion.
		NO	Vectors not dis-
			played.
		VELOCITY	Vectors show
			velocity direc-
			tion.

Command	DISPLAY (contin	nued)	
Parameter	Default	Function	
ERASE	YES	Picture erase swite	ch.
		NO	Old picture not erased.
		YES	Old picture erased.
THREED	NO	Copy view from 3D Viewer.	
		NO	Use DISPLAY
			command param-
			eters.
		YES	Set DISPLAY
			command param-
			eters to match 3D
			Viewer.
AXES	YES	Draw axes switch:	
		NO	No axes drawn.
		YES	Display coordi-
			nate axes.

The DISPLAY command draws pictures of the three dimensional geometry of the finite element mesh and conductors. Pictures can be line-drawings or coloured surfaces with hidden surfaces obscured. The discretisation can be included and vectors can be drawn to indicate the vector properties of the volumes and the current directions in the conductors. Colours are used to differentiate between volumes with different material names or to show values of assigned potentials. The parameters control the coordinate limits of the volume of three dimensional space included, the direction of the view, the parts of the model included and other options.

Pictures can also be displayed using the 3D Viewer. (See "The THREED Command" on page 4-169.)

The Coordinate Limits and the Viewing Direction

Menu Route	$\mathtt{display} \downarrow$
	Display Command view
	Display Command copy 3d view

The **DISPLAY** command draws a picture including parts of the model which are inside a volume which is a cube of dimension **2*SIZE**. The centre of the cube is at coordinates (XORIGIN, YORIGIN, ZORIGIN).

The view obtained is controlled by the eye position (XEYE, YEYE, ZEYE), the origin (XORIGIN, YORIGIN, ZORIGIN) and whether perspective is selected (+PERSPECTIVE or -PERSPECTIVE). For non-perspective (orthogonal) views the view direction is set by XEYE, YEYE and ZEYE. For perspective views the view direction is set by the eye position and the origin. The distance between the origin and the eye position is important; parts of the model behind the eye will be omitted from the display.

The viewing direction can also be altered by the rotation angle parameters (ROTX and ROTY), which have the effect of altering the values of XEYE, YEYE and ZEYE, the eye position. The values of ROTX and ROTY are not remembered. ROTATE specifies a rotation angle about the viewing direction.

Copying 3D View

The above parameters (SIZE, XEYE, YEYE, ZEYE, PERSPECTIVE, ROTX, ROTY, ROTATE, XORIGIN, YORIGIN and ZORIGIN) can also be set so that the view of the model corresponds to that of the 3D Viewer (+THREED). The THREED parameter is set back to NO after each time it is used so that these parameters can be adjusted for the next DISPLAY.

The view parameters are only effective if the previous picture is erased with **+ERASE**. If the previous picture is not erased (**-ERASE**) then the view remains as before.

The coordinate axes can be optionally displayed to show the scale of the model (**±AXES**)

Selecting Parts of the Finite Element Model

Menu Route

DISPLAY

Display Command ... select parts

I

The DISPLAY command draws all entities, volumes, facets, lines and points, created with the DEFINE and EXTEND commands (page 4-96 and page 4-112). By use of the TYPE and LABEL parameters, sub-sets of the entities can be selected in order to aid visualization of the model. The parameter TYPE can be set to any individual entity type or any combination of entity types separated by + or –. For example, to display lines and points, TYPE=LINE+POINT should be used; for everything except points, TYPE=ALL-POINT. Similarly, LABEL can be set to individual labels or combinations of labels. The additional label, NOTAIR can be used to select all material name labels except AIR. Abbreviated label names can be used where the abbreviation is not ambiguous. Abbreviations can be followed

by * to indicate that all labels which match should be selected. An entity is drawn if it has any of the labels selected. For example, to display all facets

Labels are assigned automatically to parts of the model by the commands DEFINE, EXTEND, MODIFY and SLIP. Automatically assigned labels include ALL, material names, element types potential types and boundary condition types and functions. Additional labels can be given in two ways: the LABEL command (page 4-123) can be used to give any label to any part of the model and the CHECK command (page 4-18) can be used to assign the labels DEBUG to volumes with bad shapes and EXTERNAL to facets which are not shared by two volumes.

with vector potential boundary conditions, **LABE=A*-ALL** could be used.

At the end of the DISPLAY command, LABEL is set to SAME indicating that the same labels will be used for the next picture, unless LABEL is reset. SAME can be used in further combinations of labels to add or remove labels from the previously selected list.

The part of the finite element mesh displayed can also be restricted by the parameters MESH, L1 and L2. The MESH parameter is used to select ALL meshes or one particular mesh (each DEFINE command creates a mesh with a new number). **MESH=NONE** can be used to omit the finite element mesh completely. L1 and L2 are used to select a subset of the layers of the mesh. L2 can be set to * to indicate the top layer.

Selecting Conductors

Menu RouteDISPLAYDisplay command ... select parts \rightarrow Conductors

Display command ... select parts \rightarrow Conductor numbers

The DISPLAY command draws the conductors including any symmetry copies. This can be controlled by the parameters COIL, C1 and C2. +COIL and -COIL switch the display of the conductors on and off, and C1 and C2 select a range of conductors for display. C2 can be set to * to indicate the highest numbered conductor.

Other parameters

Menu Route

DISPLAY↓ Display ... command style

Two types of picture can be produced: wire-frame line drawings or coloured surface pictures, with hidden surfaces obscured. **-HIDDEN** produces a wire-frame picture; **+HIDDEN** uses a fast but unsophisticated algorithm to produce the "hidden surface" pictures. This does not always achieve a perfect picture. It is usually possible to obtain a satisfactory picture with a suitable choice of view point or **HIDD=FULL** which orders the displayed facets by a slower but more reliable algorithm could be used. With **TYPE=VOLUME +HIDDEN**, facets which are shared by volumes with the same labels are omitted.

Hidden surface pictures can only be displayed after the surface mesh has been calculated (see "The MESH Command" on page 4-131).

The discretisation of the surfaces and volumes can also be displayed using the **ELEMENT** parameter. If the volume or surface meshes have not been calculated, the program displays using the nearest options it can.

Hidden surface views of FACETS, with LABEL set to a single potential name (AX, AY, AX, POTENTIAL, VOLTAGE or the incident potentials) are displayed as contour plots to show the values of the potential on the facets.

Arrows to show the direction of the volume vector properties can be drawn inside each volume. By default, the arrows are only drawn inside the conductors to show the current direction (**VECTOR=CONDUCTORS**) but they can also show the MATERIAL orientation, the **CURRENT** density direction or the **VELOCITY** direction. The arrow display can be switched off completely (**VECTOR=NO**).

Examples

The following example DISPLAY commands assume program default values, and that the commands are issued in sequence. The model is assumed to occupy a unit cube in the positive X, Y and Z octant of the coordinate system.

• Example: to display a wire-frame picture without conductors, orthogonal projection from the Z direction (note the use of positional parameters):

OPERA > display 0.5 xorigin=0.5 0.5 0.5 -coil

• Example: to display the conductors, and check that the reduced potential volumes enclose them:

OPERA > xeye=3 4 5 +coil labe=reduced

• Example: to obtain a "hidden-surface" picture of the conductors and one material, checking the finite element discretisation on the surfaces of the volumes, and using perspective (note that the eye position has to be moved further away to reduce the affect of the perspective):

```
OPERA > xeye=9 12 15 +perspective label=iron,
OPERA > +hidden, element=surface
```

• Example: to display the surfaces with the NORMAL MAGNETIC boundary condition:

```
OPERA > type=face label=normmagn
```

The **DUMP** Command

Menu Mouie.

OPTIONS↓ Dump picture

Command Line Parameters:

Command	DUMP		
Parameter	Default	Function	
FILE	none	Name of file to c	ontain the picture.
TYPE	POSTSCRIPT	Graphics language:	
		HPGL	Hewlett-Packard Graphics Language
		PICOUT	Vector Fields PICOUT Graphics Language
		POSTSCRIPT	Adobe PostScript
SIZE	A4	Paper sizes: HPG A0D, A1D, A2D can use A4, A or	L can use A, A3, A4, B, , A3Dor A4D; PostScript USER.
LLX	0	X-coordinate of l (TYPE=POST, S	ower-left corner in mm IZE=USER).
LLY	0	Y-coordinate of l (TYPE=POST, S	ower-left corner in mm IZE=USER).
URX	0	X-coordinate of u (TYPE=POST, S	upper-right corner in mm IZE=USER).
URY	0	Y-coordinate of u (TYPE=POST, S	upper-right corner in mm IZE=USER).
COLOUR	YES	Colour PostScript: NO implies grey-scale.	
FILL	NO	Filled polygons in HPGL: YES or NO.	
ORIENT	LANDSCAPE	PostScript paper orientation: LANDSCAPE or PORTRAIT.	
SWAP	YES	Swap black and white in PostScript:	
		NO	Colours appear as on screen.
		YES	Black and white are swapped.

The DUMP command copies the graphics commands used to create the current display to a FILE in three different formats. If no filename extension is given, extensions *.ps*, *.hgl* or *.pic* are added as appropriate. Note that

the 3D Viewer window cannot be copied to a file using the DUMP command. This window must be saved as a bitmap directly.

- Adobe PostScript (**TYPE=POST**): This has options for paper size, colour and orientation.
 - There are two standard paper sizes: European A4 and American A. Alternatively **SIZE=USER** allows the image to be printed at any size and any position on the page by the specification of the coordinates of the lower-left and upper-right corners in mm (parameter LLX, LLY, URX, URY).
 - If +COLOUR is selected, the screen colours will be matched on the paper as closely as possible, except that black and white can be optionally SWAPPED. (Black is any colour with an intensity of

less than $\frac{1}{256}$ for Red, Green and Blue; white is any colour with an

intensity of more than $\frac{255}{256}$ for Red, Green and Blue – see the COL-

OUR command.)

Grey-scale pictures can be created in several ways: the colours could be changed to grey using the COLOUR command before DUMPING (see the COLOUR command); a colour picture could be sent to a grey-scale printer; or a grey-scale picture could be stored using -COLOUR. In this last case, the grey levels are calcu-

lated using the formula $1 - \frac{r+g+b}{3}$. Thus dark colours on the

screen become light colours on the paper.

- Pictures can be orientated in two ways: **PORTRAIT** pictures have the X horizontal along the bottom of the page and the vertical axis up the left-hand side; LANDSCAPE pictures have the horizontal axis up the right-hand side of the page and the vertical axis from right to left along the bottom of the page.
- HPGL (TYPE=HPGL) pictures are intended for pen-plotters. There is a larger selection of sizes available: the sizes with D appended are for drum plotters, the other sizes are for flat-bed plotters.
 - The only other option allows polygon-fill to be selected (\pm **FILL**). If selected, polygons are filled using shading with parallel horizontal or vertical lines.
- PICOUT (TYPE=PICOUT) pictures can be re-displayed or translated using the PICOUT program supplied by Vector Fields for UNIX systems. It reads graphics files and redisplays them on the screen or converts them to PostScript or HPGL for printing or plotting. The PICOUT program prompts the user for the name of the input file and (if necessary) the name of the output file. It also prompts for options such as paper size, colour or monochrome, etc., although in some implementa-

tions these can be supplied by command line options (-cc) which are shown below.

Instructions for running PICOUT are included in the Implementation Notes supplied with the software.

PICOUT prompts the user for the input it needs. On UNIX systems, some choices can be specified as command line options, for example – **sc** below. Replies to PICOUT's prompts can be abbreviated.

PICOUT always prompts for the name of an input file. There are 3 output options: Screen (–sc), PostScript (–ps) and HPGL (–hp).

- Screen: If output to the screen is chosen, there are no further options. On workstations and X-terminals, after the graphics window has been created, its size can be adjusted using the window manager and the pictures will be scaled to fit the new size. The original aspect ratio of the pictures will be maintained by increasing either the top or right side margin. PICOUT pauses for <*carriage-return>* to continue or *Q* to quit at the end of each picture.
- PostScript: If PostScript output is chosen, PICOUT prompts for the name of the file to contain the PostScript program. If the file already exists it will be overwritten. When PICOUT has finished, this file can be sent directly to a PostScript printer. PostScript output can be written to multiple files with a single page in each file (s) or to one file with many pages in the file (-m). The advantage of multiple files is that a single page can be printed without printing all the others. The filenames for the single page, multiple file option are formed by appending .1, .2, .3 etc. to the name given at the output file prompt. On some operating systems, it is necessary to name the output file with a single part name in order to make use of this facility.

PostScript output can be created for different paper sizes. There are 2 standard sizes: European A4 (-a4) and American A (-a) for which PICOUT knows the paper size. Non-standard user defined paper sizes ($-u \ llx \ lly \ urx \ ury$) can also be given by specifying the coordinates of the lower left and upper right corners of the drawing area in millimetres.

PostScript output can be orientated on the page in landscape format $(-\mathbf{l})$ or portrait format $(-\mathbf{p})$. For paper sizes A and A4, portrait format fits 2 pictures on each page.

PostScript output can be in colour (-c) or greyscale (-g). Colour output assumes white paper, so the black and white of the original screen picture are swapped. Greyscale intensities are inverted so that dark colours appear pale grey and bright colours as dark grey. Some greyscale printer can interpret colour commands and therefore give a choice of using PICOUT's algorithm for colour to grey-scale conversion or the printer's algorithm.

- HPGL: If HPGL output is chosen, PICOUT prompts for the name of the file to contain the HPGL commands. Each page of output is written to a separate file. The filenames are formed by appending .1, .2, .3 etc. to the name given at the output file prompt. If a file already exists it will be overwritten. When PICOUT has finished, the files can be sent directly to a HPGL printer or plotter. On some operating systems, it is necessary to name the output file with a simple name so that the extension can be added.

HPGL output can be created for a variety of paper sizes. For flatbed plotters, sizes A (-a), A3 (-a3), A4 (-a4) and B (-b) are available. For drum plotters, HPGL output can be produced for sizes A0D (-a0d), A1D (-a1d), A2D (-a2d), A3D (-a3d) and A4D (a4d). The difference between the flat-bed and drum sizes is that the coordinate system origin is at the centre of the plot on drum plotters and at the bottom left corner on flat bed plotters. No PS (paper size) command is put into the files.

HPGL output can include extra line drawing commands to shade filled polygons (-f) or polygons can be represented by the outlines alone (-nf).

PICOUT OPTIONS

Output style:

-sc	Screen. PICOUT redraws the pictures onto the
	workstation screen.
-ps	PostScript. PICOUT creates a file or files of Post-
	Script commands.
-hp	HPGL. PICOUT creates files of HPGL commands.
Paper size (ign	nored with –sc):
-size	PostScript output can be produced for standard
	paper sizes A (-a) and A4 (-a4) or for a user choice
	of paper size,(-ullx lly urx ury). The integer num-
	bers given by <i>llx</i> , <i>lly</i> , <i>urx</i> and <i>ury</i> specify the posi-
	tion of the lower left and upper right corners of the
	picture in millimetres.
<i>_size</i> [d]	For flat-bed plotters, HPGL output can be produced for sizes a , a4 , a3 and b ; for drum plotters, HPGL
	output can be produced for sizes a0d, a1d, a2d, a3d
	and a4d (d for 'drum'.)
Polygon fill (H	IPGL only):
-f	Fill polygons (solid shading).

Do not fill polygons (outlines drawn). -nf

Colour or greyscale (PostScript only):

PICOUT OPTIONS (continued)

- -c Colour. White paper is assumed, so black and white are swapped.
- -g Greyscale. Intensities are inverted, so that dark colours appear as pale grey and bright colours as dark grey.

Picture orientation (PostScript only):

- -l Landscape (one picture on each page).
- -p Portrait (two half-size pictures on each page).

File output (PostScript only):

- -s Single page per file. Each picture is written to a separate file, with filenames generated from the output file name given by the user.
- -m All pages in one file.

Help:

-h For help (all other options ignored).

The **EDIT** Command

Not available from Menus

Command Line Parameters

Command	EDIT	
Parameter	Default	Function
FILE	none	Name of OPERA-3d pre processor data file.

The EDIT command reads an OPERA-3d pre processor data file into the program in edit mode. This allows major changes to be made to the data, including changes to the topology of the base plane. The data already stored in the pre processor can also be changed using the EXTEND command (page 4-112), the MODIFY command (page 4-133) and the REDEFINE command (page 4-148).

The EDIT command has one parameter which defines the name of the FILE. If no file name extension is given, the extension *oppre* is assumed.

As the file is being read a certain amount of editing can be done. Each toplevel command can be EXECUTED, IGNORED or replaced; and at breakpoints within the DEFINE command the user can add additional information or change the data which has been read. At any point it is possible to FINISH editing and read the rest of the file or SKIP over the rest of the file and return immediately to normal control.

All the commands which the EDITOR executes are decoded, so that the parameter values are stored but only certain commands are obeyed. These are the commands which define finite element data (DEFINE, EXTEND and MODIFY), conductor data (CONDUCTOR) and built-in commands (\$ OS and \$ CD are not executed).

More details of reading OPERA-3d pre processor data files are given with the READ command. When the editing is complete, the WRITE command can be used to store the edited data in a file.

The EDIT command should not be used in a \$ COMINPUT file.

Editing Top-level Commands

As the program reads each top-level command from the file, the command is displayed on the terminal with the message Your next input was:

and the user has the opportunity to EXECUTE, IGNORE or replace it, to SKIP over all of the remaining commands in the file or to FINISH reading and executing until the bottom of the file is reached.

Replacement commands should not introduce extra interactions with the program, since it is not possible to insert new command lines.

Top-level Edits		
Keyword	Meaning	
FINISH	Continue reading to end of file.	
IGNORE	Ignore the displayed command.	
SKIP	Ignore the rest of the file.	
XECUTE	Execute the displayed command.	
replacement command	Execute the replacement command.	

Editing the DEFINE Command

As the sub-commands and cursor-hits within the DEFINE command are being read, some of the keyboard command lines can be edited in the same way as top-level commands above. It is not usually sensible to use the IGNORE option except to remove an invalid command line from the file otherwise the sequence of commands would be destroyed. Similarly, a replacement command should not invoke extra interaction with the program, since it is impossible to add extra command lines.

Edits to DEFINE sub-commands		
Keyword	Meaning	
FINISH	Continue reading to end of file.	
IGNORE	Ignore the displayed command.	
SKIP	Ignore the rest of the file.	
XECUTE	Execute the displayed command.	
replacement command	Execute the replacement command.	

Additional break points are included at the ends of each of the modes of the DEFINE command. At these break points a question requiring the answer YES or NO is given enabling the user to define more points or facets, redefine facet subdivisions, move points on subsequent planes or redefine materials and boundary conditions.

Note that there is only a break point at the ends of Point Definition Mode and Facet Definition Mode when those modes are left for the first time. There is no break point when Point Definition Mode is re-entered from Facet Definition Mode or when Facet Definition Mode is re-entered from Subdivision Mode. However it is possible to re-enter Point Definition Mode from the break point at the end of Facet Definition Mode and to reenter Facet Definition Mode from the break point at the end of Subdivision Mode.

• Example - at the end of Base Plane Facet Definition Mode, the message given is:

End of facet definition mode. Do you need to define more facets? (Y or N)

In response to a YES reply, the program enters the appropriate DEFINE command mode as described on page 4-49. It is usually necessary to use the R cursor hit first so that the current state of the data can be displayed. When a Q cursor hit is used to leave the mode, the program resumes reading the file, until the next break point is reached. In the case of the above example, a YES response makes the Facet Definition Mode cursor commands available, including N for returning to Point Definition Mode (page 4-55).

FINISH and SKIP can be used as replies instead of YES or NO to end the editing by reading and executing the rest of the file or returning immediately to normal input.

Edits at DEFINE break-points		
Keyword	Meaning	
FINISH	Continue reading to end of file.	
NO	Continue reading to the next break point.	
SKIP	Ignore the rest of the file.	
YES	Make cursor input available to edit the data.	
The END Command

Menu Route

FILE↓ End OPERA-3d/Pre

Command Line Parameters

Command END No Parameters

The END command stops the OPERA-3d pre processor. All data files are closed.

Note that it is important to WRITE a pre processor data file (page 4-177) before ending the program so that all the commands and data are recorded.

If the program is ended without a WRITE command having been issued to store the pre processor data, then the file *Opera3d_Pre_n.backup* can be renamed to have a file name extension *oppre*. This file is equivalent to a pre processor data file.

The **EXTEND** Command

Menu Route

 $\mathtt{DEFINE} \downarrow$

Extend existing mesh

Command Line Parameters

Command	EXTEND			
Parameter	Default	Function		
MESH	1	Number of finite element mesh to be extended.		
EDIT	YES	Material and boundary condition editing switch:		
		NO Materials left as AIR TOTAL LINEAR, no		
		boundary conditions set.		
		YES Material properties and boundary conditions can be set after extensions.		

The EXTEND command puts the pre processor into its finite element mesh creation mode but starts from the top plane of an existing mesh. The user input is tightly structured by the program. The top mesh plane surface is extruded or swept through space thus creating new layers of volumes. The topology is maintained during the extrusion operations, but the point coordinates can be changed in the new surface created by each extrusion or sweep. If editing has been selected (+EDIT), once all the new layers have been created, the volumes in the new layers of the mesh can be assigned material attributes and boundary conditions can be assigned to the faces of the volumes.

In menu mode the **Extend with editing** option with editing adds one additional extrusion at a time and allows coordinates, material properties and boundary conditions to be edited. **Extend without editing** can be used to add several extrusions, but any variation in coordinate positions, material properties and boundary conditions must be applied later using **MODIFY**.

The MESH parameter specifies the number of the finite element mesh to be extended.

Extrusions Mode

The EXTEND command applies extrusion operations to the set of facets of the top mesh plane of previously created mesh. There must be at least one extrusion operation, but many others may be needed to define the complete problem. It is also possible to add more extrusions to a completed mesh I

using the EXTEND command again. Point movements and transformations can be performed in the same way as for DEFINE. The details of the command lines to define the extrusions are given with the DEFINE command (page 4-49).

After the extrusion has been created and all necessary points have been moved, the program moves on to the next extrusion. The user is given the option of creating another extrusion or not (valid replies are YES or NO). After a NO response, the program moves on to Material Definition Mode for the new layers, if EDITing is selected.

Material Definition Mode

It is now necessary to set the material and mesh properties within each new volume. This is done by presenting the user with each layer in turn and the user setting values which over-ride the default or current settings within each volume. Only the new layers are presented to the user, but lower numbered layers can be modified by using the FROM keyword on the material definition command line.

Volumes are selected and materials and properties defined in the same ways as for the DEFINE command (page 4-49). When all necessary changes have been made the program moves on to Boundary Condition Definition Mode for the new facets, starting with the mesh plane which was previously the top plane.

Boundary Condition Definition Mode

Boundary conditions can be set on any surface of any of the new volumes in the mesh. In order to achieve this the program presents the facets in 3 separate groups: first the facets on the planes from the previous top plane to the one below the new top plane, second the final plane of the mesh, and last the facets normal to the planes, for each of the new layers, one layer at a time. Facets of lower numbered planes or layers can be modified by using the FROM keyword on the boundary condition definition command line.

Facets are selected and boundary conditions defined in the same ways as for the DEFINE command (page 4-49). When all necessary changes have been made the program ends the EXTEND command and waits for another top-level command.

4-114

The **FILL** Command

Menu Routes	$\texttt{MESH}{\downarrow}$
	Volume mesh
	options
	Mesh

Command Line Parameters

Command	FILL	
Parameter	Default	Function
TOLERANCE	1.E-5	Geometric Tolerance
PRINT	0	Diagnostic printing level (0 or 1)

The FILL command generates the volume finite element mesh in all the pre processor volumes. The type of mesh (tetrahedra or hexahedra) is determined by which type of surface mesh has been created by the MESH command (page 4-131).

Tetrahedral meshes

Each region of the model is meshed independently, given the required triangular element mesh on the surfaces that has been created by the MESH command. If the volume has only 3 or 4 sided facets with regular subdivisions, the internal nodes are regularly positioned. Otherwise, the meshing is based on Delaunay point insertion, followed by maximising the minimum angle of the elements.

The **TOLERANCE** parameter may need to be adjusted if the software fails to create a mesh successfully. Problems can occur if the element size varies too much between the surfaces of a region; the tolerance should be increased to overcome this. In other cases, for example, a complicated region with a wide range of feature sizes, the tolerance may need to be reduced.

If the mesh generation process fails, and the **TOLERANCE** parameter does not correct the problem, then it is likely that the element size is changing too much between adjoining surfaces. Regions that cannot be meshed are flagged using the **DEBUG** label and can be selected for **DISPLAY**. Adjust the edge subdivisions to reduce the element size variations in such regions.

Hexahedral meshes

Hexahedral elements can only be generated if the model uses regions that are hexahedra or degenerate hexahedra. A regular finite element mesh is generated in all regions, by subdivision of the region to similar shaped elements. The **TOLERANCE** parameter is used to test for coincident points in degenerate shapes.

The **HELP** Command

Menu Route

HELP↓ Help

Command Line Parameters

Command	HELP
No Paramet	ters

The HELP command gives help to remind users of several of the features of the program. The topics are:

- **System Overview**: this gives a flow-chart of the OPERA-3d pre processor top level commands indicating the sequence in which they should be used to prepare a data set for analysis.
- **Command Interpreter**: this summarizes the syntax and built-in help features of the command decoder, including details of sub-commands and cursor commands. More information is given in chapter 2
- Euler Angles: the definitions of the Euler Angles used within OPERA-3d is given, and information on how to use the escape function \$EULER to specify them. More information is given on page 2-34
- **New Features**: this summarize the features of the program which have been added since the previous version.
- Units: this lists the units used in the systems allowed by the analysis programs.

The program prompts for the name of a topic.

Keyword	Meaning	
COMMAND	Command interpreter.	
EULER	Euler angles.	
NEW	New features.	
SYSTEM	System overview.	
UNITS	Unit systems.	
QUIT	Leave the HELP COMMAND.	

When accessed from the menus, the HELP command has different topics explaining the use of the program and the menu interface.

The **IDEAS** Command

Menu Route

 $\mathtt{FILE} \downarrow$

Read I-DEAS universal file

Command Line Parameters

Command IDEAS
No Parameters

The IDEAS command introduces a set of sub-commands which can be used to read a finite element mesh from an I–DEAS Universal File and associate OPERA-3d material properties and boundary conditions with the elements and nodes it contains.

The READ sub-command is used to read the data. Additional information about materials and boundary conditions can be supplied using the subcommands BOUNDARY and MATERIAL after the file has been read. Conductors can be added and the data can be DISPLAYED (page 4-96) and written to an OPERA-3d analysis database (see "The SOLVERS Command" on page 4-150), but the finite element data cannot be modified.

Universal Data File for Input to OPERA-3d

The Universal Data File created by I–DEAS must contain the following Datasets (Level 6 or Master Series):

Dataset Name	Dataset Number
Header	151
Nodes	15, 781 or 2411
Material Properties	91, 747, 773, 1710 or 1714
Elements	71, 780 or 2412
Restraints or Loads	755, 756, 782, 791 or 792

The order of the datasets is not important except that any data which is referenced by other datasets must be defined before it is referenced; e.g.nodes must be defined before elements. However, if an element refers to an undefined material number, the program will invent a material name. Other datasets may also be included in the input file, but will be ignored.

The I-DEAS Universal Data File Level 6 and Master Series formats are defined in the I-DEAS documentation published by SDRC, and are not

repeated here. The following information describes how the data is used in interface to OPERA-3d.

Material
PropertiesUp to 100 materials may be used in the universal file. The properties in the
universal file are ignored; only the material number and name are used by
the interface. A different material should be used for each different combi-
nation of OPERA-3d material and volume properties. For example, in the
universal file data, the volume representing free space might be divided
into two parts with different material names, so that in OPERA-3d part can
use REDUCED potential and the other part TOTAL potential. Each mate-
rial defined in the universal file will initially be given the properties of AIR
with TOTAL potential and LINEAR elements. The OPERA-3d material
name, potential type and element type can be changed and other volume
properties added using the MATERIAL sub-command (page 4-121).

Material names are invented for any material number which is not defined before is it is referenced by an element.

Element Topologies

I

I

The interface is able to process elements of the following types:

111	Linear tetrahedron
115	Linear hexahedron
116	Quadratic hexahedron
118 or 119	Quadratic tetrahedron

Mixed element types are allowed, but is this case tetrahedra will be represented by degenerate hexahedra.

All quadratic elements in the universal file are converted to linear elements by the interface. Quadratic elements can be selected either by specifying that a particular universal file material should use quadratic elements (MATERIAL sub-command, page 4-121) or that all elements should be quadratic when the analysis database is created (SOLVERS command, page 4-150).

Restraints and
LoadsRestraint and Load
the same boundary
ature but the values
the node numbers a

Restraint and Load datasets identify the sets of nodes which should have the same boundary condition. The types of restraint must be nodal temperature but the values of temperature in the universal file are ignored. Only the node numbers are used. The **BOUNDARY** sub-command can be used to associate one or more OPERA-3d boundary conditions with element faces which have all their nodes in the a particular restraint set.

Displaying Universal File Data

Universal File data can be displayed using the DISPLAY command (page 4-96). Line drawings are not available, but displays without hidden surfaces can show surface or volume elements.

As the universal file data is being read, the elements are allocated to volumes, one for each universal file material. A label MATEn (where *n* is the universal file material number) is added to each volume so that they can be selected for display. After OPERA-3d properties have been added, the material names and other property labels can be used as well.

Similarly, the element facets identified by the nodes in each boundary condition set are grouped and given facet labels BCSETn (where *n* is a counter of how many boundary condition sets have been read). The boundary condition facets can be displayed using these labels. After the OPERA-3d boundary conditions have been applied using the BOUNDARY, the OPERA-3d boundary condition names can be used as well.

The IDEAS Sub-command BOUNDARY

Menu Route

FILE↓

I-DEAS universal file \rightarrow Define Boundary Conditions

Command Line
Parameters

Sub-command	BOUNDARY			
Parameter	Default	Function		
NUMBER	none	Boundary condition set number		
OPTION	ENQUIRE	Option:		
		ENQUIRE	List boundary conditions defined for this set.	
		SET	Define or clear boundary conditions for this set.	

Sub-command	BOUNDARY (continued)			
Parameter	Default	Function		
CONDITION	none	Boundary condition name or NONE:		
		AX	AX boundary condition.	
		AY	AY boundary condition.	
		AZ	AZ boundary condition.	
		DAX	DAX boundary condition.	
		DAY	DAY boundary condition.	
		DAZ	DAZ boundary condition.	
		DERIVATIVE	DERIVATIVE boundary	
			condition.	
		INAX	INAX boundary condition.	
		INAY	INAY boundary condition.	
		INAZ	INAZ boundary condition.	
		NONE	Clear all boundary condi-	
			tions.	
		NORMELEC	NORMAL ELECTRIC	
			boundary condition.	
		NORMMAGN	NORMAL MAGNETIC	
			boundary condition.	
		PEC	PEC boundary condition.	
		POTENTIAL	POTENTIAL boundary con-	
		DADIATION	dition.	
		RADIATION	RADIATION boundary con-	
		SLIP	SLIP boundary condition.	
		SYMMETRY	SYMMETRY boundary con-	
		TANGELEC	houndary condition	
		TANGMAGN	TANGENTIAL MAG-	
			NETIC boundary condition.	
		VOLTAGE	VOLTAGE boundary condi-	
			tion.	
VALUE	none	Potential value.		
LABEL	none	Drive label.		

The **BOUNDARY** sub-command can be used to list existing boundary conditions on the set of nodes given by **NUMBER**, (**OPTION=ENQUIRE**) or add new ones (**OPTION=SET**, **CONDITION=***name*). **OPTION=SET**, **CONDITION=NONE** clears all existing conditions on the set of nodes. The OPERA-3d boundary conditions are described on page 4-86.

I

The IDEAS Sub-command MATERIAL

Menu Route

 $\mathtt{file} \downarrow$

I-DEAS universal file \rightarrow Define Materials

Command Line Parameters

Sub-command	MATERIAL			
Parameter	Default	Function		
NUMBER	none	Universal file material number.		
OPTION	ENQUIRE	Option:		
		ENQUIRE	List material properties.	
		SET	Define material properties.	
NAME	none	OPERA-3d ma	terial name.	
POTENTIAL	none	Potential type:		
		REDUCED	Reduced potential.	
		TOTAL	Total potential.	
		VECTOR	Magnetic vector potential.	
ELEMENT	none	Element type:		
		LINEAR	First order elements.	
		QUADRATIC	Second order elements.	
PACK	none	Packing factor.		
SCALAR	none	Scalar property: charge density or rotational		
		velocity.		
CURX	none	X-component of current density.		
CURY	none	Y-component of current density.		
CURZ	none	Z-component of current density.		
VELX	none	X-component of velocity.		
VELY	none	Y-component of velocity.		
VELZ	none	Z-component of velocity.		
THETA	none	Material orientation Euler Angle θ .		
PHI	none	Material orientation Euler Angle ϕ .		
PSI	none	Material orientation Euler Angle ψ .		

The MATERIAL sub-command can be used to list existing material properties (OPTION=ENQUIRE) or define new ones (OPTION=SET). OPTION=ENQUIRE also sets the default values of the other parameters to the current values for the material.

The current density, velocity and material orientation vectors can be expressed as functions of X, Y and Z.

The OPERA-3d material properties are described in the section "Material Definition Mode" on page 4-78.

The IDEAS Sub-command QUIT

Menu Route

 $\mathtt{file} \downarrow$

I-DEAS universal file \rightarrow Return

Command Line Parameters

Sub-command	QUIT
No Parameters	

The QUIT sub-command leaves the IDEAS command and returns to the top-level commands.

The IDEAS Sub-command READ

Menu Route

file↓

I-DEAS universal file \rightarrow Read File

Command Line	
Parameters	

Sub-command	READ	
Parameter	Default	Function
FILE	none	Name of file.

The READ sub-command reads a Universal File to input materials, nodes, elements and boundary conditions. If no file name extension is given, the extension *unv* is assumed.

The LABEL Command

Menu Route

MODIFY Labels

Command Line Parameters

Command	LABEL	
Parameter	Default	Function
MESH	1	Number of finite element mesh to be labelled.
START	1	Number of the first plane or layer. Planes are num- bered: 1,2, 3, etc.; mid-extrusion planes and layers are numbered: 1.5, 2.5, 3.5, etc.

The LABEL command allows labels to be added or removed from the entities (points, lines, facets and volumes) of the finite element mesh. Some labels are added automatically by the DEFINE command (page 4-49): every entity has label ALL, facets have the boundary condition types, and volumes have the material names, potential types and element types. Such labels can only be changed by the MODIFY command. Other labels are added to parts of the model by the CHECK command (page 4-18), FILL command (page 4-114), MESH command (page 4-131) and SLIP command (page 4-149). Additional labelling by the LABEL command allows entities to be grouped for DISPLAY and points to grouped for the TRANS-FORM command (page 4-176). To enable multiple points to be labelled more easily, any labelling operation on volumes, facets or lines is also applied to the points which define them.

Facet and volume labels are stored in analysis databases and can be used in the post processor.

The data of the finite element mesh is separated into 4 sections, corresponding to the modes of the DEFINE command. The user is asked first to choose between points, lines, facets and volumes.

LABEL Modes		
Keyword	Meaning	
FACETS	Add or remove labels on facets.	
LINES	Add or remove labels on lines.	
POINTS	Add or remove labels on points.	
QUIT	Leave the LABEL command.	
VOLUMES	Add or remove labels on volumes.	

Valid replies are: FACETS, LINES, POINTS, QUIT or VOLUMES.

In menu mode labels can be set on one plane of layer at a time. The program prompts for a value of **Plane number** or **Layer number**. The menu item **Select/de-select entity** allows entities to be added or removed from a list. Menu item **Select and define** adds one last entity to the list and causes the program to display a DialogBox into which the labels can be entered. The menu item **List conditions** allows the labels to be listed for an individual entity.

In keyboard mode, each mesh plane or extrusion layer of the mesh specified by the MESH parameter, starting with the plane or layer given by the START parameter, is presented to the user. Mesh planes are numbered 1, 2, 3, etc., and mid-extrusion planes and layers 1.5, 2.5, 3.5, etc. Layer 1.5 is between plane 1 and plane 2. The facets are presented in two sections: first those lying in the mesh planes, then those connecting the extruded planes. Entities can be selected using the cursor, singly with a **<space>**, or in groups with κ for all but the last and **<space>** for the last.

A new label can be ADDED to or an existing label REMOVED from the selected entities and corresponding entities in other planes or layers. For each set of entities the selection cursor hits and labelling sub-commands are the same.

Entity Sele	Entity Selection Menu and Cursor Hits			
Cursor hit	Menu item	Function		
<space></space>	Select and define	Select entity nearest cursor and prompt for a labelling com- mand.		
F	Finish	Finish labelling this type of entity.		
н		Display menu help message explaining all the cursor options. (More help is available after entity selection.)		
ĸ	Select/de-select entity	Select the entity nearest the cur- sor. Repeating K for a selected entity, de-selects it. A labelling command for all the selected entities can be given after a <space> cursor hit.</space>		

Entity Selection

Entity Selection Menu and Cursor Hits (continued)			
Cursor hit	Menu item	Function	
L	List labels	List all the latentity.	bels on the nearest
Q		Leave this pla	ane or layer.
R	Re-draw picture	Reconstruct t program requ Valid replies	he display. The lests a new size. are:
		4 numeric	umin, umax,
		values	<i>vmin, vmax</i> Default values are the current settings.
		CURSOR	Select diagonally opposite corners of the display area with < space >cursor hits.
		RESTORE	Return to previ- ous screen size.
		BOUND	Use bounding rectangle of geometry.
		<return></return>	Reconstruct at the same size.
v	Change view	Change the U system. This and the extru	VW coordinate affects the view sion direction.
Z	Aspect-ratio search	Switch aspect or off (page 4	tratio searching on I-52).

Labelling Sub-commands in Keyboard Mode

Following a **<space>** cursor hit, one of the following sub-commands should be given. The commands apply to the entity selected with the **<space>** and to any previously selected with κ .

The labelling sub-commands consist of the following items:

Labelling Sub-commands		
Command and parar	neter:	
ADD label	Add a label to the selected entities.	
HELP	Obtain help on all sub-commands and options.	

Labelling Sub-commands (continued)			
QUIT	Abandon the currently selected entities.		
REMOVE <i>label</i>	Remove a label to the selected entities.		
Optional keywords -	setting multiple entities:		
ALL	Sets all entities in layer(s) or plane(s).		
FROM value	Sets all layers/planes from layer/plane number given through to the current layer/plane or the layer/plane number given with TO.		
KEEP	Keeps current selection of entities for another ADD or REMOVE command.		
TO value	Sets all layers/planes from current layer/plane or the layer/plane number given with FROM through to the layer/plane number given. The value may be * to indicate the top layer/plane.		

Labelling DialogBox in Menu Mode

In the DialogBox, ADD or REMOVE must be selected and a label name given.



The MATERIALS Command

Menu Route

 $\texttt{define}^{\downarrow}$

Material Properties

Command Line Parameters

Command	MATEF	MATERIALS		
Parameter	Default	Function		
OPTION		PICK	Adds a material label to a list to be set.	
		UNPICK	Clears the list of material labels to be set.	
		RESET	Sets picked materials to have the properties of air.	
		MODIFY	Sets the data for the picked materials.	
		METRE	Work in SI units.	
		CGS	Work in CGS units.	
		INCH	Work in SI units with inches.	
		MM	Work in SI units with MM.	
		MICRON	Work in SI units with microns.	
		LIST	Lists the material proper- ties of the picked materi- als.	
		DELETE	Deletes the picked mate- rials.	
MATERIALLABEL		Material label use	ed with OPTION=PICK	
ANISOTROPY		Sets the anisotrop mittivity and con-	by of all permeability, per- ductivity	
		ISOTROPIC	Set to isotropic.	
		PACKED	Set to packed.	
		MULTIPLE	Set to anisotropic.	
LINEARITY		Sets the propertie	s to be linear or non-linear.	
		LINEAR	Set to linear.	
		NONLINEAR	Set to non-linear.	

Command	MATEF	RIALS (continued)	
Parameter	Default	Function	
MUANISOTROPY		Sets the anisotropy of permeability.	
		ISOTROPIC	Set to isotropic.
		PACKED	Set to packed.
		MULTIPLE	Set to anisotropic.
MULINEARITY		LINEAR	Set to linear.
		NONLINEAR	Set to non-linear.
MU		Isotropic linear p	ermeability.
HC		Isotropic coercivity	ity.
BH		Isotropic non-line	ear BH curve.
MPHASE		Phase lag for isot	ropic permeability.
MUXX		Anisotropic com	ponents of linear permea-
MUYY		bility.	
MUZZ			
HCX		Anisotropic com	ponents of linear coerciv-
HCY		ity.	
HCZ			
BHX		File names for an	isotropic BH characteris-
BHY		tics.	
BHZ			
MAPHASE		Complex phase lag for anisotropic permeability.	
SIGANISOTROPY		Sets the anisotropy of conductivity.	
		ISOTROPIC	Set to isotropic.
		MULTIPLE	Set to anisotropic.
SIGMA		Isotropic conductivity.	
SPHASE		Phase lag for isotropic conductivity.	
SIGXX		Anisotropic com	ponents of conductivity.
SIGYY			
SIGZZ			
SAPHASE		Complex phase lag for anisotropic permit- tivity.	
EPSANISOTROPY		Sets the anisotropy of permittivity.	
		ISOTROPIC	Set to isotropic.
		MULTIPLE	Set to anisotropic.
EPSILON		Phase lag for isot	ropic permittivity.
EPHASE		Isotropic permittivity.	

Notes

Command	MATERIALS (continued)	
Parameter	Default	Function
EPSXX		Anisotropic components of permittivity.
EPSYY		
EPSZZ		
EAPHASE		Complex phase lag for anisotropic permit- tivity.

This command defines the material characteristics for use by the analysis programs.

A set of material labels is picked using the command repeatedly, with **OPTION=PICK** and a **MATERIALLABEL** specified. A material label can be removed from the set using **OPTION=UNPICK**. If no **MATERIALLA-BEL** is given, the set is emptied.

Issuing the command with **OPTION=MODIFY** will modify the properties of the picked material labels to the new values given in the parameters. The value of properties associated with the material labels are unchanged if the parameter is unset.

The default value of each of the parameters is updated to be the common value of all the picked material labels. If the data of one of these parameters is unset, or the picked material labels do not share the same value, then the parameter value is left clear.

New material labels are assumed to be linear and isotropic. **OPTION=RESET** will clear the properties associated with all of the picked material labels.

The working material unit set can be changed using **OPTION=METRE**, **OPTION=CGS** or one of the other sets. If the units are changed, the values in the parameters are converted to the new unit set, and values entered will be interpreted in this unit set (note however that in **CGS** units, the coercive force \mathbf{H}_c is in units Oersted, and for all other options, \mathbf{H}_c is in units A/m).

'The properties of all material labels can be listed using **OPTION=LIST**.

Material labels that are not used, i.e. have no cell referencing them can be deleted using **OPTION=DELETE**. Deleting a material label that is in use will reset its properties to those of air.

All parameters can be specified. Which of these values will used by the analysis will depend upon the analysis module being used, e.g. the phase lag properties are not used by **TOSCA**.

This command defines the MATERIAL properties for use by the analysis programs. All parameters can be specified for any model. Each analysis program will use the appropriate subset of properties which it needs. The values can be adjusted during creation of the database using the SOLVER MATERIAL sub-command.

All materials start with properties of air.

- **option=pick:** A set of material labels is selected using the command repeatedly, with **option=pick** and a material label.
- **option=modify:** After a set of material labels has been picked, the default values of each parameter will be set to the data that is common to the picked material labels. This data can be modified using **option=modify**, with the new values supplied for each of the parameters. If a value is not supplied for a parameter value, that data stored with each material label is unaffected. There are two sets of parameters, one for **anisotropy=isotropic** or **packed** and the other for **anisotropy=multiple**. Only isotropic properties can be set using the GUI.
- **option=reset** will reset the values of all the data of all picked material labels to the properties of air.
- option=METRE, option=CGS: The values can be specified in CGS units or SI with various length units by using Material option=CGS, option=METRE etc. If the units are changed, the values in the parameters are converted to the new unit set, and values entered will be interpreted in this unit set.
- **option=list:** The properties of all materials can be listed using **option=list**.
- **option=delete:** Material labels that are not used, i.e. have no volume referencing them can be deleted using **option=delete**. Deleting a material that is in use will reset its properties to those of air.

The **MESH** Command

Menu	Routes
------	--------

 $\texttt{MESH} \downarrow$

- Surface mesh options ... triangles ... quadrilaterals
- **Command Line**

r arameters

Command	MESH			
Parameter	Default	Function		
NORMAL	0.1	Curved surface fitting tolerance.		
PARAME- TER	NO	Triangulate in surface parameter space: YES or NO.		
UPDATE	YES	Check that the trianguthe Delaunay criterion point is added: YES	llation matches n after each or NO.	
ELEMENT	QUADRILATERAL	Surface element type:		
		QUADRILATERAL	4-sided sur-	
			face facets leading to a	
			hexahedral volume mesh.	
		TRIANGLE	3-sided sur-	
			leading to a	
			tetrahedral	
TOI	1 OF 5	Coincident point tole	volume mesn.	
IUL	1.0L-J	Conclucin point tole	ance.	

The finite element mesh is created by first meshing the region surfaces using the MESH command. The FILL command (page 4-114) can then be used to generate the volume mesh.

The MESH command generates the surface mesh on all the pre processor surface facets from the subdivision the user has specified for the edges. The two types of elements (TRIANGLES and QUADRILATERALS) that may be selected, determine which type of volume element will be created by the FILL command (tetrahedra or hexahedra) (page 4-114).

If the MESH command fails on any surface facet because the variation of element size is too extreme or the discretisation does not obey the rules for

I

quadrilateral meshes, that facet is given the label DEBUG so that it can be identified using the DISPLAY command (page 4-96) or the THREED command (page 4-169).

Triangular meshes

Triangular meshes can be generated on any polygonal surface. The only restriction is that surfaces with more than 4 sides should be planar.

Points are automatically added to the surface to give a smooth variation of element size, given the edge subdivision. A regular triangular mesh will be created on 3 and 4 sided surfaces, if the edge subdivisions are equal on opposite sides. Additional points may be added to the surface mesh if the surface is curved. The difference between the real surface shape and the finite element approximation is checked against the NORMAL tolerance and points may be added to improve the finite element model.

In general the surface mesh should be created in the real coordinate system used for the model. There may be exceptional cases where the surface mesh is required to be created in the **PARAMETER** space of the surface, for example, to achieve a rapid variation in element size.

Quadrilateral meshes

Quadrilateral meshes can only be generated if the model consists of regions which are all hexahedra, or degenerate hexahedra. The following conditions must be fulfilled in order to generate a mesh of quadrilateral surface elements:

- there are no facets with more than 4 sides
- all 4 sided facets have equal subdivisions on opposite sides
- all 3 sided facets have two sides with the same subdivision

The **MODIFY** Command

Menu Route

MODIFY↓

Mesh number Point coordinates Subdivisions Material properties Boundary conditions

Command Line Parameters

Command	MODIF	Y		
Parameter	Default	Function		
MESH	1	Number of finite element mesh to be modified.		
START	1	Number of the first plane or layer. Planes are num- bered: 1,2, 3, etc.; mid-extrusion planes and layers are numbered: 1.5, 2.5, 3.5, etc.		
THREED	NO	Use 3D Viewer.		
		NO	No 3D Viewer.	
		YES	Use 3D Viewer to show the model during MODIFY.	

The MODIFY command allows much of the finite element mesh data created by the DEFINE (page 4-49) and EXTEND (page 4-112) commands to be changed. The interaction is very similar to that used in DEFINE and EXTEND, and therefore only the differences will be highlighted.

Only the topology of the base plane cannot be modified. The EDIT and REDEFINE commands (page 4-108 and page 4-148) can be used to change this topology.

The data of the finite element mesh is separated into 4 sections, corresponding to the modes of the DEFINE command. The user is asked first to choose between points, subdivisions of base plane facets and extrusion layers, materials and volume properties or boundary conditions.

MODIFY Modes			
Keyword	Meaning		
BOUNDARY	Boundary conditions can be changed.		
MATERIALS	Material names, potential types, element types and properties can be changed.		
POINTS	Points can be moved and mesh planes transformed.		

I

MODIFY Modes (continued)			
QUIT	Leave the MODIFY command.		
SUBDIVISIONS	Subdivision of the planes and the layers can be changed. Layers can be changed between LINEAR and QUADRATIC extrusions.		

Valid replies are: BOUNDARY, MATERIAL, POINTS, QUIT or SUBDI-VISIONS.

In menu mode modifications can be made to one plane or layer at a time. The program prompts for a value of PLANE NUMBER or LAYER NUMBER.

In keyboard mode, for each section, each mesh plane or extrusion layer of the mesh specified by the MESH parameter, starting with the plane or layer given by the START parameter, is presented to the user. Mesh planes are numbered 1, 2, 3, etc., and mid-extrusion planes and layers 1.5, 2.5, 3.5, etc. Layer 1.5 is between plane 1 and plane 2.

Point Modification Mode

Menu Route

MODIFY↓

Point coordinates

In menu mode, the plane selected is presented to the user. Points can be moved individually or in groups. Individual points are selected using menu item Move Point and can be repositioned graphically including the use of grids and construction lines (see page 4-53) or numerically. Points can be grouped using menu item Select/de-select point and transformed using Transform points. If no points have been grouped, all the points of the plane are transformed. If mistakes are made, the point or points can be returned to their original locations using Undo. This must be done before leaving the MODIFY command.

In keyboard mode each plane, starting from the plane number given by the **START** parameter and ending with the top plane is presented to the user. Points can be moved individually or in groups. Individual points are selected using the **<space>** cursor hit and can be repositioned using the cursor including the use of grids and construction lines (see page 4-53) or the keyboard. Points can be grouped using **K** and transformed using **T**. If no points have been grouped, then **T** will transform all the points of the plane. If mistakes are made, the point or points can be returned to their original locations using **U** cursor hit. This must be done before finishing modifications to the plane.

Full details of the cursor commands and the transformation options are given in the following sections.

Point Selection Menu and Cursor Hits

Point Selection Menu and Cursor Hits					
Cursor hit	Menu Item	Function			
<space></space>	Move point	Select point nearest cursor to be moved. It can be repositioned using the Point Repositioning			
н		Display men explaining al (More help a selection.)	u help message Il the cursor options. available after point		
к	Select/de-select point	Select point a be transform selected point	nearest the cursor to ed. Repeating K for a nt de-selects it.		
Q	Finish Editing	Leave this plane. If on a mid- extrusion plane, move on to the top plane of the new layer. Other- wise the program asks about the next extrusion (<i>keyboard mode</i>) or moves on to material defini- tions (<i>menu mode</i>).			
R	Re-draw picture	Reconstruct the display. The pro- gram requests a new size. Valid replies are:			
		4 numeric values	<i>umin, umax, vmin, vmax</i> Default values are the current settings.		
		CURSOR	Select diagonally opposite corners of the display area with <space></space> cursor hits.		
		RESTORE	Return to previous screen size.		
		BOUND	Use bounding rec- tangle of geometry.		
		<return></return>	Reconstruct at the same size.		

Point Selection Menu and Cursor Hits (continued)				
Cursor hit	Menu Item	Function		
Т	Transform points	Define general transformations for the points on the plane. (Use κ to select points, or all points will be transformed.) The transforma- tion options are given on page 4- 77.		
υ	Undo move or transform	Undo the last move or transform operation on this plane or layer.		
v	Change view	Change the UVW coordinate sys- tem. This affects the view and the extrusion direction.		
Z	Aspect-ratio search	Switch aspect ratio searching on or off (page 4-52).		

The Point Repositioning Mode Menu and Cursor Hits

Point Repositioning Mode Menu and Cursor Hits				
Cursor hit	Menu item	Function		
<space></space>	At mouse	Reposition the point at the cur- sor cross hair position or the nearest grid point.		
С	Give U, V, W	Switch to input from keyboard in cartesian coordinates UVW. Coordinates should be entered in free format. Default values of U, V and W are the values prior to the move. They can be accessed via the variables #1 , #2 and #3 . Type Q to leave point in its present position.		
Е	Remove C_line	Erase construction line closest to cursor cross hair.		
G	Grid	Define or remove a grid.		
Н		Display menu help message explaining all the cursor options.		
I	Enter C_lines	Input construction line specifications.		
L	List C_lines	List construction line specifica- tions.		

Point Repositioning Mode Menu and Cursor Hits (continued)					
Cursor hit	Menu item	Function			
N	On nearest C_line	Reposition the point on the near-			
		est constructi	ion line with mini-		
		mum normal	distance. This only		
		affects the U	and V coordinates;		
		W remains unchanged.			
Р	Give R, Theta, W	Switch to input from keyboard			
		in local polar	coordinates RθW.		
		Coordinates	should be entered		
		in free formation	t. Default values of		
		R, ϑ and W a	the values prior		
		to the move.	to the move. They can be		
		accessed via the variables #1 ,			
		#2 and #3 . Type Q to leave			
-		point in its present position.			
Q	Return without moving	Leave the point at its previous			
		position.			
R	Re-draw picture	Reconstruct the display. The program requests a new size.			
		Valid replies	are:		
		4 numeric	umin, umax,		
		values	vmin, vmax		
			Default values		
			are the current		
			settings.		
		CURSOR	Select diago-		
			nally opposite		
			corners of the		
			display area with		
			<space> cul-</space>		
		DESTORE	Boturn to provi		
		RESTORE	Return to previ-		
			Us screen size.		
		BOUND	Use bounding		
			rectangle of		
			geometry.		
		<return></return>	Reconstruct at		
			the same size.		

Point Repositioning Mode Menu and Cursor Hits (continued)				
Cursor hit	Menu item	Function		
Т	Show coordinates	Type the coordinates of the point closest to the cross-hairs, and select it for repositioning instead of the point already selected. The coordinates are given in cartesian (UVW) and polar ($R\theta W$) coordinates.		
x	At C_line intersection	Reposition the point at the clos- est construction line intersection or end. This only affects the U and V coordinates; W remains unchanged.		
Z	Aspect-ratio search	Switch aspect ratio searching on or off (page 4-52).		

Plane Transformation Options The transformations can be any combination of the following:

	Plane Transformation Commands				
Command	Parameters and Function				
CARTESIAN	<i>exp_u exp_v exp_w</i> The points are moved to new positions defined by expressions for their (<i>u</i> , <i>v</i> and <i>w</i>) coordinates in terms of their old coordinates: U, V, R, TH and W. The copy number, (#COPY), can also be used.				
DISPLACE	<i>du dv dw</i> Displace points in the current viewing local coordinate system by adding (<i>du</i> , <i>dv</i> , <i>dw</i>) to their coordinates (U, V, W).				
POLAR	$exp_r exp_{\theta} exp_w$ The points are moved to new positions defined by expressions for their (<i>r</i> , θ and <i>w</i>) coordinates in terms of their old coordinates: U, V, R, TH and W. The copy number, (#COPY), can also be used.				
PROJECT	du dv dw ucentre vcentre wcentre uangle vangle wangle Project the points in the direction (du, dv, dw) until they intersect the XY plane of a coordinate system specified by its origin (ucentre, vcentre, wcentre) and axis rotation angles (uangle, vangle, wangle).				
QUIT	End the sequence of transformations.				

4-139

Plane Transformation Commands (continued)		
Command	Parameters and Function	
ROTATE	<i>ucentre vcentre wcentre uangle vangle wangle</i> Rotate points by angles (<i>uangle, vangle, wangle</i>) around axes parallel to the local coordinate system and passing through the point (<i>ucentre, vcentre, wcentre</i>).	
SCALE	<i>ucentre vcentre factor</i> Scale points by multiplying the distance from local coor- dinate point (<i>ucentre</i> , <i>vcentre</i>) by <i>factor</i> . The W coordi- nate of the points is not affected.	

In menu mode the transformation commands are offered as menu items and the parameters are supplied using parameter boxes.

Subdivision Modification Mode

The two mesh generators (for tetrahedra and hexahedra) have different requirements for the subdivision of mesh volumes. Hexahedral elements can only be generated in hexahedral volumes or degenerate hexahedral volumes with regular subdivision (i.e. opposite lines of each facet must have the same numbers of elements). Tetrahedral elements can be generated in any volume with any subdivision.

To support the hexahedral mesh generator, the subdivision of the facets in the mesh plane should be modified use the method described in "Subdivision Modification Mode" on page 4-139, accessed by using **START=1**. The subdivision of extrusion layers should be modified as described in "Modifying Subdivisions of the Extrusion Layers" on page 4-141, avoiding the EDIT option.

The EDIT option ("Modifying Subdivisions for Tetrahedral Meshes" on page 4-142) provides facilities for modifying the subdivision of lines in a more general way to support the flexibility of the tetrahedral mesh generator.

Modifying the	MODIFY↓
Subdivisions of	Subdivision $ ightarrow$ Uniform subdivision $ ightarrow$ In-plane
All the Planes	In keyboard mode a facet is selected for subdivision by position

In keyboard mode a facet is selected for subdivision by positioning the cross-hairs just inside a facet, close to the edge to be divided and pressing the <**SPACE**> bar. The program prompts for the number of subdivisions. Subdivisions can also be set globally, i.e. all edges set to the same subdivision number, using the **G** cursor hit.

In menu mode the subdivision can be set first using menu item **Set subdi-vision** and can then be applied to a single edge or globally to all the inplane lines in the entire mesh.

If the subdivision is given for a single edge and that edge is one side of a quadrilateral facet, then the subdivision of the opposite edge will be set to the same value. The subdivision will be carried through the mesh until a triangular facet, the edge of the mesh or a facet already set in this operation is reached. (This does not happen if a polygon has been defined on the base plane.)

When the subdivision of an edge is set, the corresponding edges in all other layers are also set to the same subdivision.

Subdivisions of edges can be changed any number of times, until the required pattern is achieved.

Facet Subdivision Menu and Cursor Hits

Facet Subdivision Menu and Cursor Hits		
Cursor hit	Menu item	Function
<space></space>	Apply to line	Select closest edge for its subdivision to be set.
A	Finish valid hex mesh?	Leave Facet Subdivision Modifica- tion, performing checks.
G	Apply globally	Select all edges for their subdivi- sions to be set to the same value.
н		Display menu help message explaining all the cursor options.
Q		Leave Facet Subdivision Modifica- tion and proceed to Extrusion Layer Modification.

Facet Subdivision Menu and Cursor Hits (continued)				
Cursor hit	Menu item	Function		
R	Re-draw picture	Reconstruct	Reconstruct the display. The pro-	
		gram request	s a new size. Valid	
		replies are:		
		4 numeric	umin, umax, vmin,	
		values	vmax	
			Default values are	
			the current settings.	
		CURSOR	Select diagonally	
			opposite corners of	
			the display area with	
			<space> cursor</space>	
			hits.	
		RESTORE	Return to previous	
			screen size.	
		BOUND	Use bounding rec-	
			tangle of geometry.	
		<return></return>	Reconstruct at the	
			same size.	
х	Finish no check-	Leave Facet Subdivision Modifica-		
	ing	tion, without	checking.	
v	Change view	Change the U	JVW coordinate sys-	
		tem. This aff	ects the view and the	
		extrusion dir	ection.	
Z	Aspect-ratio search	Switch aspec	t ratio searching on or	
		off (page 4-5	2)	

Modifying Subdivisions of the Extrusion Layers

MODIFY↓

Subdivision ightarrow Uniform subdivision ightarrow Extrusions

The program presents a list of the current settings of the extrusion subdivisions and requests that any necessary changes are made. The list gives the layer number, the number of extrusions and the layer type. The layer type is either LINEAR or QUADRATIC; a quadratic extrusion layer has a midextrusion plane and the W directed lines are quadratic.

The subdivision and layer type of each layer can be changed. Changing a linear layer to a quadratic layer causes a new mid-extrusion plane of points to be added. The positions of these points is initially at the geometric mid-point of the layer, but can be changed in Point Modification Mode. Changing a quadratic extrusion to linear removes the mid-extrusion plane.

Single line commands specify the changes to be made. These consist of the layer number, the number of subdivisions and the keyword LINEAR or QUADRATIC. * can be used for the layer number to indicate that all layers should be set the same. Three keywords are available:

- LIST to list the current settings
- EDIT PLANE *n* to switch to variable subdivision editing on plane *n*.
- EDIT LAYER *n* to switch to variable subdivision editing in layer *n*.
- **QUIT** to leave Extrusion Layer Modification Mode.
- Example to change layer 3 to have 5 subdivisions and be quadratic, to list the settings and then leave Extrusion Layer Modification Mode:

OP-SUBDIV > 3 5 q OP-SUBDIV > list OP-SUBDIV > quit

MODIFY↓

Subdivision ightarrow Variable subdivision

To support the flexibility of the tetrahedral mesh generator, the subdivision of the lines in mesh planes or extrusion layers can be modified using the Variable Subdivision option.

In keyboard mode this is selected using the **EDIT LAYER** n or **EDIT PLANE** n options in the extrusion subdivision modification mode.

The selected plane or layer is displayed so that one or more lines can be selected. Note that, for layers, the lines to be selected appear as the corners of the facets. A new number of subdivisions can be applied to the lines and also to the corresponding lines in other planes or layers using the FROM and TO keywords. The number of subdivisions can be given as an expression in terms of N, the existing number of subdivisions and SIDE, the length of the line.

Modifying Subdivisions for

Tetrahedral

Meshes

Line Selection Menu and Cursor Hits

Line Selection Menu and Cursor Hits			
Cursor hit	Menu item	Function	-
<space></space>	Select and define	Select line nea	arest cursor and
		prompt for su	bdivision.
н		Display menu	help message
		explaining all	the cursor options.
		(More help is	available after
72	Colort/do colort lino	Soloot the line	nearest the surger
r	Select/de-select line	Repeating K f	for a selected line
		de-selects it	The subdivision for
		all the selected	d lines can be given
		after a <spa< b=""></spa<>	CE> cursor hit.
Q	Finish	Leave this pla	ine or layer.
R	Re-draw picture	Reconstruct th	he display. The pro-
	-	gram requests	a new size. Valid
		replies are:	
		4 numeric	umin, umax, vmin,
		values	vmax
			Default values are
			the current set-
			tings.
		CURSOR	Select diagonally
			opposite corners of
			the display area
			cursor hits
		RESTORE	Return to previous
			screen size.
		BOUND	Use bounding rec-
			tangle of geome-
			try.
		<return></return>	Reconstruct at the
			same size.
v	Change view	Change the U	VW coordinate sys-
		tem. This affe	cts the view and the
		extrusion dire	ction.
Z	Aspect-ratio search	Switch aspect	ratio searching on
		or off (page 4	-52).

Subdivision Subcommands in Keyboard Mode

Following a **<space>** cursor hit, one of the following sub-commands should be given. The commands apply to the line selected with the **<space>** and to any previously selected with **K**.

The subdivision sub-commands consist of the following items:

Subdivision Sub-commands			
Command and	Command and parameter:		
number	Divide the line into <i>number</i> elements. <i>number</i> can be an expression in terms of N, the existing subdivision and SIDE, the length of the line.		
HELP	Obtain help on all sub-commands and options.		
QUIT	Abandon the currently selected lines.		
Optional keywo	ords - setting multiple entities:		
ALL	Sets all lines in layer(s) or plane(s).		
FROM value	Sets all layers/planes from layer/plane number given through to the current layer/plane or the layer/plane number given with TO.		
KEEP	Keeps current selection of lines for another subdivision.		
TO value	Sets all layers/planes from current layer/plane or the layer/ plane number given with FROM through to the layer/ plane number given. The value may be *to indicate the top layer/plane.		

Subdivision DialogBox in Menu Mode

Following **Select and define**, a value for the subdivision must be supplied in the DialogBox.



The value can be a number or an expression in terms N, the existing number of subdivisions or SIDE, the length of the line. Other inputs and buttons give access to other layers and to setting all lines in the layer or plane.

Material Modification Mode

Menu Route	MODIFY↓ Material properties		
	The material and mesh properties within each volume from the layer spec- ified with the START parameter can be changed.		
	• <i>In keyboard mode:</i> Each layer is presented to the user in turn and values may be set which over-ride the current settings within each volume.		
	• <i>In menu mode:</i> Only the layer selected by START is presented. Other layers can be modified by setting values for From and To .		
	The method for selecting volumes and defining materials is exactly the same as in the DEFINE command (page 4-78).		
	Boundary Condition Modification Mode		
Menu Route	MODIFY↓ Boundary conditions		
	Boundary conditions can be set on any surface of any volume in the mesh. In order to achieve this the program presents the facets in 4 separate groups: first the facets on the base plane, second the planes between the first and the last, third the final plane of the mesh, and last the facets normal to the planes, one layer at a time. However, if the START parameter is greater than 1, then some of the planes are omitted. If START points to an extru- sion layer then only the extrusion facets are presented.		

Facets are selected and boundary conditions defined in exactly the same ways as in the DEFINE command (page 4-86).

The **READ** Command

Menu Routes

 $\texttt{FILE} \downarrow$

Read pre-processor file

 $\mathtt{define} \downarrow$

Read conductor data

Command Line Parameters

I

Command	READ	
Parameter	Default	Function
FILE	none	Name of OPERA-3d pre processor data file.

The **READ** command reads a pre processor data file into the program. There is one parameter which defines the name of the **FILE**. If no file name extension is given, the extension *oppre* is assumed.

OPERA-3d pre processor data files consist of all the commands issued to the program including cursor commands. Thus reading a file recreates the data in the same way as when commands are typed at the terminal except that fewer text messages appear on the screen and only the picture of the base plane is shown. All messages are written to the dialogue file *Opera3d_Pre_nn.lp*.

All the commands are decoded, so that the parameter values are stored but only certain commands are executed. These are the commands which define finite element data (DEFINE, EXTEND and MODIFY), the conductor data (CONDUCTOR) and built-in commands (\$ OS and \$ CD are not executed).

If any errors are encountered while reading the file, the number of the line last read is given. This may enable correction of the file. The file has comments in it to aid the user to follow the sequence of commands. These occur in place of the cursor commands which are introduced by the character string curs. Comments have the character string **** followed by the number of following comment lines.

If the file does not contain all the commands necessary to complete a DEFINE command, normal terminal input is resumed when the end of the file is reached. It is advisable to use the **R** cursor hit first to obtain a picture of the current position within the DEFINE sequence.
Pre processor data files can be edited to contain **\$ COMINPUT** commands and a command input file can also contain a **READ** command. However, if a command input file contains a **READ** command, the file to be read must not contain a **\$ COMINPUT** command.

The **READ** command in menu mode should only be used to read complete data files, i.e. files which contain complete **DEFINE** commands. Incomplete data files can only be read in keyboard mode.

The **REDEFINE** Command

Menu Route

 $\text{DEFINE} \downarrow$ Redefine a mesh

Command line Parameters

Command	REDEFINE		
Parameter	Default Function		
MESH	1	Number of finite element mesh to be redefined.	

The REDEFINE command removes all the volumes from a mesh, leaving only the baseplane points, lines and facets. It then enters the same command or menu sequence as the DEFINE command (page 4-49) to add, remove or edit point, facets and subdivisions in the baseplane and then extrude. The only way this differs from the DEFINE command is that the baseplane facets from the mesh are already there when Point Definition Mode is entered (page 4-55).

This implies that all the points on the baseplane will retain their existing \mathbf{u} , \mathbf{v} and \mathbf{w} coordinates. Any new points added will use the specified \mathbf{w} coordinate.

The parameter, MESH, allows any mesh in a multiple mesh model to be redefined.

The **SLIP** Command

Menu	Routes

 $\begin{array}{c} \texttt{MODIFY} \\ \texttt{Add slip surface} \end{array}$

Remove slip surface

Command Line Parameters

Command	SLIP			
Parameter	Default	Function		
OPTION	ADD	Command option:		
		ADD Add slip surface.		
		REMOVE Remove all SLIP boundary condi-		
			tions.	
RADIUS	1	Radius of slip surface.		
RTOLERANCE	0.001	Tolerance on RADIUS.		

The SLIP command is the most convenient way to apply the SLIP boundary condition to the interface between the stator and rotor of CARMEN rotating machine analysis models. The slip surface should be cylindrical, extend the full length of the model, and only touch volumes modelled with AIR and TOTAL scalar potential.

- The SLIP OPTION=ADD command adds a slip surface by applying the SLIP boundary condition to all facets whose points are at the given RADIUS±RTOLERANCE. The label SLIP is also added to the points for use in the TRANSFORM command (page 4-176).
- The **SLIP OPTION=REMOVE** command removes the **SLIP** boundary condition from all facets which have it and also removes the label **SLIP** from their points.

The **SOLVERS** Command

Menu Route

FILE↓ Analysis

Command Line Parameters

Command	SOLVE	RS			
Parameter	Default	Function			
FILE	none	Database file name.			
PROGRAM	none	Analysis program:			
		CARMEN	Rotating machines.		
		ELEKTRA	Low frequency eddy cur-		
			rents.		
		SCALA	Space charge beam.		
		SOPRANO	High frequency.		
		TOSCA	Statics.		
TYPE	none	Analysis type:			
		CURRENT	Current flow (TOSCA).		
		ELECTROSTATICS	Electrostatics (TOSCA).		
		EV	Eigenvalues		
			(SOPRANO).		
		MAGNETOSTATICS	Magnetostatics (TOSCA).		
		ROTATIONAL	Rotational velocity		
			(ELEKTRA).		
		SSAC	Steady-state AC		
			(ELEKTRA or		
			SUPRANU).		
			Transient (ELEKTRA).		
		VELOCITY	Linear velocity		
			(ELEKTRA).		

I

I

4-151

Command	SOLVE	RS (continued)			
Parameter	Default	Function			
OPTION	NEW	Database option:			
		ADD	Add a new simulation to		
			an existing database.		
		COPY	Copy a simulation within an existing database.		
		DEFAULT	Equivalent to COPY or		
			there is a volume mesh or not.		
		EDIT	Edit a pending simulation in an existing database.		
		NEW	Create a new database.		
		RESTART	Restart a completed anal-		
		ysis in an existing da			
			base.		
CASE	1	Simulation number to b RESTARTED.	e COPIED, EDITED or		
CASE ELEMENT	1 MIXED	Simulation number to b RESTARTED. Element-type over-ride	e COPIED, EDITED or		
CASE	1 MIXED	Simulation number to b RESTARTED. Element-type over-ride LINEAR	e COPIED, EDITED or All elements linear.		
CASE ELEMENT	1 MIXED	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED	e COPIED, EDITED or All elements linear. Elements as previously		
CASE	1 MIXED	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED	e COPIED, EDITED or All elements linear. Elements as previously set.		
CASE	1 MIXED	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED QUADRATIC	All elements linear. Elements as previously set. All elements quadratic.		
CASE ELEMENT UNITS	1 MIXED CGS	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED QUADRATIC Units of the data.	All elements linear. Elements as previously set. All elements quadratic.		
CASE ELEMENT UNITS	1 MIXED CGS	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED QUADRATIC Units of the data. CGS	All elements linear. Elements as previously set. All elements quadratic. Practical CGS units.		
CASE ELEMENT UNITS	1 MIXED CGS	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED QUADRATIC Units of the data. CGS INCH	All elements linear. Elements as previously set. All elements quadratic. Practical CGS units. SI units with lengths in		
CASE ELEMENT UNITS	1 MIXED CGS	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED QUADRATIC Units of the data. CGS INCH	All elements linear. Elements as previously set. All elements quadratic. Practical CGS units. SI units with lengths in inches.		
CASE ELEMENT UNITS	1 MIXED CGS	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED QUADRATIC Units of the data. CGS INCH METRE	All elements linear. Elements as previously set. All elements quadratic. Practical CGS units. SI units with lengths in inches. SI units.		
CASE ELEMENT UNITS	1 MIXED CGS	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED QUADRATIC Units of the data. CGS INCH METRE MICRON	All elements linear. Elements as previously set. All elements quadratic. Practical CGS units. SI units with lengths in inches. SI units. SI units. SI units with lengths in		
CASE ELEMENT UNITS	1 MIXED CGS	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED QUADRATIC Units of the data. CGS INCH METRE MICRON	All elements linear. Elements as previously set. All elements quadratic. Practical CGS units. SI units with lengths in inches. SI units. SI units. SI units with lengths in μm.		
CASE ELEMENT UNITS	1 MIXED CGS	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED QUADRATIC Units of the data. CGS INCH METRE MICRON MM	e COPIED, EDITED or All elements linear. Elements as previously set. All elements quadratic. Practical CGS units. SI units with lengths in inches. SI units. SI units. SI units with lengths in μm. SI units with lengths in		
CASE ELEMENT UNITS	1 MIXED CGS	Simulation number to b RESTARTED. Element-type over-ride LINEAR MIXED QUADRATIC Units of the data. CGS INCH METRE MICRON MM	All elements linear. Elements as previously set. All elements quadratic. Practical CGS units. SI units with lengths in inches. SI units. SI units. SI units with lengths in μm. SI units with lengths in mm.		

This command creates or modifies a database file for analysis by one or more of the OPERA-3d analysis programs. The command prompts for some additional information and has a set of sub-commands which should be used to set other options and material properties. The SOLVERS command has been designed to be used in menu mode and the options displayed in the menu boxes are restricted to those appropriate for the type of analysis selected.

Analysis types

The parameters **PROGRAM** and **TYPE** together select the analysis type. The following are valid combinations.

OPERA-3d Analysis Types				
PROGRAM	TYPE	Menu item	Meaning	
CARMEN		Rotating Machine	Eddy currents induced by rotational motion in a transient field including non-linear materials and perma- nent magnets.	
		Low Frequency		
ELEKTRA	SSAC	Steady-state AC	Eddy currents induced by low-frequency AC fields, including quasi- non-linear materials.	
ELEKTRA	TRANSIENT	Transient	Eddy currents induced by transient fields, including non-linear materials and perma- nent magnets.	
ELEKTRA	VELOCITY	Linear velocity	Eddy currents induced by linear motion of smooth components in a DC field, including non-linear materials and permanent mag- nets.	
ELEKTRA	ROTATION	Rotation velocity	Eddy currents induced by rotational motion of smooth rotors in a DC field, including non- linear materials and permanent magnets.	
SCALA		Space charge beam	Electrostatics, includ- ing particle emitters and tracking and space charge effects.	

OPERA-3d Analysis Types (continued)				
PROGRAM	TYPE	Menu item	Meaning	
		High Frequency		
SOPRANO	SSAC	Steady-state AC	High-frequency analy-	
			sis data defined fre-	
			quency.	
SOPRANO	EV	Eigenvalues	Eigenvalue analysis of	
			cavities.	
		Statics		
TOSCA	MAGNETIC	Magnetostatics	Non-linear magneto-	
			statics.	
TOSCA	ELECTRIC	Electrostatics	Non-linear electroto-	
			statics.	
TOSCA	CURRENT	Current Flow	Non-linear current	
			flow.	

Database files and file names

OPERA-3d database files are binary files. The pre processor adds a file name extension *op3*.

A database file contains one finite element mesh and one or more simulations. The finite element mesh consists of:

- the nodes
- the elements, including material names and potential types and references to pre processor facets and volume numbers
- the unit set

A simulation consists of:

- the analysis type
- the analysis options
- the boundary conditions
- the material properties
- the coils and drive specifications
- an optional title

When a database is created, the finite element data is stored in it, together with a simulation, which is initially marked as 'pending'. Additional simulations can be added to a database. During analysis, the OPERA-3d analysis programs will solve any pending simulations of the appropriate type found in the database.

Creating a new database

Menu Route FILE↓ Analysis ... create new database

Command Line SOLVE OPTION=NEW,

Before a new database can be created, the surface and volume mesh must be generated (see "The MESH Command" on page 4-131 and "The FILL Command" on page 4-114). The program writes the database with the finite element data and creates a new simulation of the type selected.

Two sets of options are available when creating a new database: the units being used and the element type.

Units

Parameters

The OPERA-3d databases and analysis programs have one set of units. The pre processor data is converted to the internal units when the database is created. The pre processor data is interpreted in any one of five unit systems. These are:

Units					
Menu Item	CGS	SI (metres)	SI (mm)	SI (Microns)	SI (Inches)
Keyword	CGS	SI	MM	MICRON	INCH
length	cm	m	mm	μm	inch
magnetic flux density	gauss	tesla	tesla	tesla	tesla
magnetic field strength	oersted	amp m ⁻¹	amp m ⁻¹	amp m ⁻¹	amp m ⁻¹
magnetic scalar pot.	oersted cm	amp	amp	amp	amp
magnetic vector pot.	gauss cm	weber m ⁻¹	weber m ⁻¹	weber m ⁻¹	weber m ⁻¹
electric flux density	coulomb cm ⁻²	coulomb m ⁻²	coulomb m ⁻²	coulomb m ⁻²	coulomb m ⁻²
electric field strength	volt cm ⁻¹	volt m ⁻¹	volt m ⁻¹	volt m ⁻¹	volt m ⁻¹
electric potential	volt	volt	volt	volt	volt
conductivity	siemen cm ⁻¹	siemen m ⁻¹	siemen mm ⁻¹	siemen μm^{-1}	siemen inch ⁻¹
current density	amp cm ⁻²	amp m ⁻²	amp mm ⁻²	amp µm⁻²	amp inch ⁻²

The unit set is stored in the database and loaded into the post processor.

ELEMENT override

The element types (LINEAR or QUADRATIC) are set during the Materials Definition Mode of the DEFINE, EXTEND and MODIFY commands (page 4-78, page 4-112 and page 4-133). It is possible to over-ride these settings to set all elements to one type (LINEAR or QUADRATIC), or to leave them as previously set (MIXED).

Element Types			
Keyword	Meaning		
LINEAR	All elements linear (8-noded hexahedra or 4-noded tetrahedra).		
MIXED	Element types as set in DEFINE, MODIFY or EXTEND commands.		
QUADRATIC	All elements quadratic (20-noded hexahedra or 10-noded tetrahedra).		

Editing an existing database

Menu Route	FILE↓ Analysis use existing database
Command Line Parameters	SOLVE OPTION=ADD, SOLVE OPTION=COPY,CASE=n, SOLVE OPTION=EDIT,CASE=n, SOLVE OPTION=RESTART,CASE=n,
Adding a new simulation	To add a new simulation to an existing database, it is necessary that the pre processor contains the corresponding model and that the surface and vol- ume mesh are generated. Before meshing, the MODIFY command (page 4-133) can be used to change the boundary conditions. Material names, potential types, point coordinates and subdivisions should not be changed. The coils can also be changed using the CONDUCTOR sub-command, MODIFY ("The CON- DUCTOR Command" on page 4-24) and the external field using the CONDUCTOR sub-command EXTERNAL (page 4-43).
	As the simulation is added, the analysis options and material properties can be set.
	NB : Potential types which are not part of the formulation of an analysis

program are treated as a valid type. TOSCA will treat VECTOR potential

volumes as TOTAL scalar potential; SOPRANO will treat all volumes as VECTOR potential.

Copying aCopying a simulation does not need the pre processor model or finite ele-
ment mesh. A new simulation of the same analysis type is created. Only the
material properties and analysis options can be changed.

Editing aEditing a simulation does not create a new simulation. It allows the materialsimulationproperties and analysis options of a pending simulation to be changed.

Restarting a simulation Restarting a simulation does the same as copying, allowing the same changes, except that within the analysis program, the old solution is used as an initial solution.

SOLVERS Command Prompts

The **SOLVERS** command prompts the user for some additional information:

- a title (all analysis types)
- drive information (CARMEN, ELEKTRA, SOPRANO-SS and TOSCA)
- eigenvalues (SOPRANO-EV)

Title Simulations can be annotated with a title. This can have as many lines of up to 80 characters as required. It is delimited by a * as the first character of the last line. Titles can be displayed by the post processor SHOW command (page 5-92).

Each conductor in an OPERA-3d model has a drive label; non-zero potential boundary conditions and external fields can also have drive labels. The SOLVERS command prompts the user for drive information for each drive label.

- Statics (ELEKTRA-VL and TOSCA): the drive information is a scaling factor. If the analysis is the second or subsequent case with the conductor fields and integrals already in the database, the fields and integrals will be scaled from a previous case to reduce the analysis time.
- **Steady-State AC:** the drive information required is the phase angle in degrees for each drive (zero corresponds to a cosine drive).

Drive

Information

٠

Transient Drives	~ *	
Keyword	Meaning	Data
DC	F = 1 for all time.	none
STEP	t < 0: F = 0	none
	$t \ge 0: F = 1$	
RAMP	t < 0: F = 0	t _c
	$t \ge 0, t \le t_c: F = \frac{t}{t_c}$	
	$t > t_c: F = 1.$	
SINE	t < 0: F = F(0)	f and ϕ
	t ≥ 0 : F = sin $2\pi ft + \phi$	
COSINE	t < 0: F = F(0)	f and ϕ
	t ≥ 0 : F = cos $2\pi ft + \phi$	
PEAK	t < 0: F = 0	t _c
	$t \ge 0: F = at \exp(-t^2/b).$	
	a and b are chosen such that $F = 1$ at	
	$t = t_c$.	
RISE	t < 0: F = 0	t _c
	$t \ge 0: F = 1 - exp(-t/t_c).$	
TABLE (switch on)	$t \le 0: F = 0$	filename
	t > 0: $F =$ cubic splines from tabulated	
	data in a time-table file.	film
IUFF (lable switch off)	t < 0: $F = F(0)t > 0$: $F = cubic splines from tabulated$	juename
	data in a time-table file	
	F(0) is the value of the function in the	
	table file at time t=0.	

Transient: the drive information specifies the shape in time of each drive using the following options:

The drive function is then used to scale the corresponding 'driving' field.

The program prompts for the additional information needed for each drive type. In addition, the keyword **ROTATING** should be specified for any drives which are in the rotating part of a CARMEN model.

Time-table files should be created by an editor. They consist of 2 columns of numbers in free-format, the first giving values of time and the second of function value. Consecutive lines with the same value of time introduce a discontinuity of function value or time-derivative into the cubic-spline fitting. Time-table files can be displayed using the GRAPH command of the post processor (page 5-53).

Eigenvalues The data for the SOPRANO Eigenvalue analysis program includes the number of eigenvalues required. They can be found in a specified range of frequencies or close to a specified frequency.

SOLVERS Subcommands ed

Seven sub-commands allow material properties and analysis options to set, edited and checked.

- MATERIAL: to define material properties
- **SETTINGS**: to set the analysis options
- **PERIODICITY**: to define periodicity data (ELEKTRA, SCALA and TOSCA)
- **DRIVE**: to add drive fields from a table file
- CASE: to specify drive frequencies (ELEKTRA-SS and SOPRANO-SS), output times (ELEKTRA-TR) or output angles (CARMEN)
- CHECK: to check the database and display a summary of the materials and options selected
- **QUIT**: to leave the **SOLVERS** command.

CASE sub-command

Menu Route	file \downarrow	
	Analysis	
	create new database	ightarrow Drive frequencies
	use existing database	ightarrow Output times
		ightarrow Output angles

Command Line
Parameters

Sub-Command	CASE		
Parameter	Default	Function	
COMMAND		Command o	ptions:
		ADD	Add a new VALUE to the end of
			the list.
		DELETE	Delete the item given by POSI-
			TION from the list.
		INSERT	Insert a new value after the item
			given by POSITION .
		LIST	List all the values.
		REPLACE	Replace the VALUE given by POSITION .
POSITION		Position in t REPLACE.	he list for DELETE, INSERT or
VALUE		Value for A	DD, INSERT or REPLACE.

The CASE sub-command should be used to create and edit a list of multiple case data:

- ELEKTRA-SS and SOPRANO-SS: multiple frequencies. At least one frequency must be supplied.
- ELEKTRA-TR: output times. The program automatically inserts zero in the first position in the list.
- CARMEN: output angles, unless the rotation speed is zero, in which case output times must be given.

CHECK sub-command

Menu Route	$\mathtt{FILE}_{\downarrow}$
	Analysis
	\dots create new database $ o$ Check
	\dots use existing database $ o$ Check

Command Line	
Parameters	

Sub-Command	CHECK
No Parameters	

The CHECK sub-command should be used to ensure that all the relevant analysis options have been specified and that all materials have been given properties.

DRIVE sub-command

Menu Route	file↓				
	Analysis				
	create new database	\rightarrow	Add	drive	fields
	use existing database	\rightarrow	Add	drive	fields

Command Line			
Parameters	Sub-Command	DRIVE	
	Parameter	Default	Function
	FILE	none	Name of table file containing fields.
	LABEL	none	Label for drive.

The DRIVE sub-command adds drive fields to the analysis database from a table file. The fields replace the fields from coils for a TOSCA magneto-statics analysis or form an additional drive for a CARMEN or ELEKTRA analysis.

The format of the table file is described on page 5-19. The first three columns should hold the node coordinates and the second three columns the magnetic fields.

If this command is used, the adaptive RHS integrals must not be used, since only nodal values of the drive field are available (see "SETTINGS subcommand" on page 4-165). I

The program prompts the user to provide the scaling factor (statics), phase angle (steady-state ac) or the drive shape (transient) which should be used with the additional drive. The prompts and replies take the same format as for coil and boundary condition drives (section "Drive Information" on page 4-156).

MATERIAL sub-command

Menu Route	$\texttt{FILE} \downarrow$	
	Analysis	
	create new database	ightarrow Materials
	use existing database	ightarrow Materials

Command Line Parameters

Sub-Command	MATERIAL		
Parameter	Default	Function	
NAME	none	Material name.	! for list.
TYPE	LINEAR	Material type:	
		LINEAR	Constant properties.
		NONLINEAR	Properties depend on field
			strength.
ANISOTROPY	ISOTROPIC	Material struct	ure:
		ISOTROPIC	Same properties in all directions.
		PACKED	Laminated material
			(TOSCA only).
		MULTIPLE	Tensor properties.
CANCEL	NO	Cancel the mat	erial command to clear
		defaults. CAN	CEL always reverts to NO
		after use.	
MUX	1	Relative perme	ability (isotropic or μ_{xx}).
MUY		Relative perme	eability μ_{yy} .
MUZ		Relative perme	eability μ_{zz} .
MPHA	0	Complex phase	e lag for permeability.
HCX	0	X component c	of coercive force, H_c .
HCY	0	Y component c	of coercive force, H_c .
HCZ	0	Z component o	of coercive force, H_c .
BHX		BH file (isotrop	pic or X-direction).
BHY		BH file (Y-dire	ection).

Sub-Command	MATERIAL	(continued)
Parameter	Default	Function
BHZ		BH file (Z-direction).
SIGX	0	Conductivity (isotropic or σ_{xx}).
SIGY		Conductivity σ_{yy} .
SIGZ		Conductivity σ_{ZZ} .
SPHA	0	Complex phase lag for conductivity.
JEX		JE file (isotropic or X-direction).
JEY		JE file (Y-direction).
JEZ		JE file (Z-direction).
EPSX	1	Relative permittivity (isotropic or ε_{xx}).
EPSY		Relative permittivity ε_{yy} .
EPSZ		Relative permittivity ε_{zz} .
EPHA	0	Complex phase lag for permittivity.
DEX		DE file (isotropic or X-direction).
DEY		DE file (Y-direction).
DEZ		DE file (Z-direction).

When a database is created, all materials have the properties of air or the values given by the top-level MATERIAL command (page 4-127). The MATERIAL sub-command can be used as many times as necessary to edit the material properties to the ones required.

In the current version, only the BH characteristic can be non-linear; the parameters to specify JE or DE curves are ignored. See also page 4-166 for a description of how the material TYPE and properties are interpreted in linear and non-linear analyses.

The BH curves can be specified as **\$default** to request the default BH curve which is displayed by the first use of the BHDATA command (page 4-15).

If only the material name is given, then the default values for all the parameters are set as appropriate for that material. The **MATE** +CANC command clears the default values.

In menu mode only those properties relevant to the analysis type of the current simulation are presented for editing.

PERIODICITY sub-command

Menu Route

${\tt file} \downarrow$

Analysis

... create new database \rightarrow Periodicity conditions ... use existing database \rightarrow Periodicity conditions

Command Line

Parameters

Sub-Command	PERIO	DICITY	
Parameter	Default	Function	
OPTION	none	Option:	
		ADD	Add a new periodicity condi-
			tion.
		DELETE	Delete a periodicity condition.
		EDIT	Edit a periodicity condition.
		LIST	List all periodicity conditions.
NUMBER	none	Condition nu	mber for DELETE or EDIT .
XBC	none	Translation in	n x-direction.
YBC	none	Translation in	n y-direction.
ZBC	none	Translation in	n z-direction.
TBC	none	Euler angle θ).
PBC	none	Euler angle	Ф.
SBC	none	Euler angle 4	Υ.
TYPE	none	Connection t	ype:
		NEGATIVE	Potential changes sign
		POSITIVE	Potential has same sign

The **PERIODICITY** sub-command connects together nodes on facets which have the **SYMMETRY** boundary condition (see page 4-89). The connection is made by use of a coordinate transformation. The points are first off-set by (**XBC**, **YBC**, **ZBC**) and are then rotated using the Euler angles (**TBC**, **PBC**, **SBC**). In most cases either the translations or the rotations will be zero. If a transformed (slave) node is coincident with an untransformed (master) node, then a periodicity condition is established between them, such that the potential of the slave is the same as (**TYPE=POSITIVE**) or the negative of (**TYPE=NEGATIVE**) the potential of the master.

Coordinate Transformation	θ	Φ	Ψ
rotation around X-axis	θ	90	-90
rotation around Y-axis	θ	0	0
rotation around Z-axis	0	Φ	0

Euler angles are described in detail on page 2-34, but here are some examples:

QUIT sub-command

Menu Route	$\texttt{FILE} \downarrow$	
	Analysis	

- \dots create new database \rightarrow Return
- ... use existing database \rightarrow Return

Command Line
Parameters

Sub-Command	QUIT
No Parameters	

The QUIT sub-command closes the database and leaves the SOLVERS command. QUIT also runs CHECK to display the final properties of the simulation.

It is good practice to follow the QUIT sub-command with a WRITE command, so that all the SOLVERS sub-commands are recorded in the pre processor data file (page 4-177).

SETTINGS sub-command

Menu Route

file \downarrow

Analysis

- \dots create new database \rightarrow Linear solution
- ... use existing database \rightarrow Non-linear solution
- - \rightarrow Simple RHS Integrals
 - \rightarrow Adaptive RHS Integrals
 - \rightarrow External fields
 - \rightarrow Scala iteration data
 - \rightarrow Timestep options
 - \rightarrow Rotation speed
 - ightarrow Use Upwinding
 - ightarrow No Upwinding
 - \rightarrow Automatic potential cuts
 - \rightarrow No potential cuts

Command Line Parameters

Sub-Command	SETTINGS		
Parameter	Default	Function	
LINEAR	NO	Linear soluti	on: YES or NO.
TYPE	NEWTON	Iteration type	e (TOSCA):
		NEWTON	Newton-Raphson itera-
			tions.
		SIMPLE	Simple updates.
NITERATIONS	21	Number of n	on-linear iterations.
TOLERANCE	0.001	Non-linear convergence tolerance.	
RELAXATION	0.1	Under-relaxation factor (SCALA	
		only).	
RHS	ADAPTIVE	RHS integral type:	
		SIMPLE	Trapezium rule on each
			element edge.
		ADAPTIVE	Up to 9 gauss points on
			each edge.
HXEXT	0	X component of the external field.	
HYEXT	0	Y component of the external field.	
HZEXT	0	Z component of the external field.	
UPWINDING	NO	Use upwinding for ELEKTRA-VL	
		analysis: YES or NO.	
ITPTSTEP	21	Non-linear iterations per time step	
		(ELEKTRA-	TR).

Sub-Command	SETTINGS	(continued)	
Parameter	Default	Function	
UPDATE	SIMPLE Time stepping method (ELEK' TR).		ng method (ELEKTRA-
		AD2RK	Adaptive 2 nd order Runge-Kutta.
		AD4RK	Adaptive 4 th order Runge-Kutta.
		F2RK	Fixed time step 2 nd order Runge-Kutta.
		F4RK	Fixed time step 4 th order Runge-Kutta.
		SIMPLE	Fixed time step Crank- Nicholson
MAXADERR	1	Maximum % error in adaptive time stepping.	
DELTAT	0.001	Fixed or initial time step.	
RPM	3000	Rotation speed (CARMEN)	
POTENTIALCUT	YES	Use automatic cuts in TOSCA: YES or NO.	

The **SETTINGS** command sets various analysis options.

In menu mode only those options relevant to the current simulation are available.

- Linear or Non-linear: Linear analyses use constant material properties; non-linear analyses update the material properties, depending on the solution and re-solve. Non-linear analyses can include some linear materials and linear analyses can include non-linear materials. In this latter case, the material property characteristic (e.g. BH curve) is used to supply the value of the material property (permeability). The slope of the curve at zero flux-density is used.
- **RHS Integrals**: The line and surface integrals of coil fields which are part of TOSCA magnetostatics, CARMEN and ELEKTRA analyses can be done using simple integrals, which only use the values of coil fields at the nodes or using adaptive integrals which use additional field points in order to reduce the errors.
- External Fields can be added to TOSCA (magnetostatics), SCALA, ELEKTRA and CARMEN analyses. For ELEKTRA-SS, ELEKTRA-TR and CARMEN, the external field and associated drive label should be first defined using the CONDUCTOR EXTERNAL sub-command ("The CONDUCTOR Sub-command EXTERNAL" on page 4-43). The values of the field vector can be modified using this command.

Also see the OPERA-3d User Guide for more information on External Fields.

- Automatic Potential cuts can be used in TOSCA magnetostatic analysis to automatically insert potential cuts to avoid having multiply connected volumes, where a loop of total potential volume encloses a nonzero net current.
- SCALA Iterations converge to a consistent set of particle trajectories and fields. The number of iterations and under-relaxation factor can be set.
- **Upwinding** is a technique to improve analysis of moving systems (ELEKTRA-VL). The analysis program reports whether upwinding is required or not.
- **Time stepping** in ELEKTRA-TR can use fixed time steps or can adjust the time step to achieve a given accuracy. If several similar models are to be analysed, the most efficient approach might be to use adaptive 4th order Runge-Kutta for one model to ascertain the appropriate time step to use. Subsequent models can then use a fixed time step method.
- **Rotation speed** in CARMEN.

The TABLE Command

Menu Route

 $\mathtt{file} \downarrow$

Write node table file

Command Line Parameters

Command	TABLE	
Parameter	Default	Function
FILE	none	Name of table file.
UNIT	CGS	Name of length unit.

The TABLE command is provided to facilitate an interface to other programs, primarily so that coil fields from other sources can be added to an OPERA-3d database (see "DRIVE sub-command" on page 4-160).

The TABLE command can only be used after the volume mesh has been generated using the FILL command (page 4-114).

The coordinates of the points in the table are always stored as centimetres, and are scaled as necessary depending on the value of UNIT.

The format of table files is described in "TABLE Files" on page 5-19.

The THREED Command

Menu Route

 $\mathtt{display} \downarrow$

3d Viewer ... refresh display

Command Line Parameters

Command	THREED			
Parameter	Default	Function		
ELEMENT	NO	Element displa	Element display switch.	
		NO	No element sub-	
			division.	
		SURFACE	Subdivision on	
			volume surfaces.	
		VOLUME	Subdivision	
			within volumes.	
MESH	ALL	Mesh number, ALL or NONE.		
TYPE	VOLUME	Type(s) of enti	Type(s) of entities to be displayed.	
		VOLUME, FA	CET, LINE, POINT,	
		ALL or SAME		
LABEL	NOTAIR	Label(s) on en	tities.	
L1	1	First layer to be displayed.		
L2	*	Last layer to be	e displayed. * means	
		top layer.		
COIL	YES	Conductor display switch.		
		NO	Conductors not	
			displayed.	
		YES	Conductors dis-	
			played.	
C1	1	First conductor to be displayed.		
C2	*	Last conductor	to be displayed. *	
		means highest numbered condu		

Command	THREED (continued)		
Parameter	Default	Function	
VECTORS	ORS CONDUCTORS Vector displ		itch.
		CONDUCTORS	Vectors show current direction on conductors.
		CURRENT	Vectors show current density direction.
		MATERIAL	Vectors show material orienta- tion.
		NO	Vectors not dis- played.
		VELOCITY	Vectors show velocity direc- tion.

The THREED command starts or updates the picture in the 3D Viewer. Pictures show the three dimensional geometry of the finite element mesh and conductors using line-drawings or coloured surfaces with hidden surfaces obscured. The discretisation can be included and vectors can be drawn to indicate the vector properties of the volumes and the current directions in the conductors. Colours are used to differentiate between volumes with different material names or to show values of assigned potentials. The parameters control the parts of the model included and other options.

The orientation, position and size of the model in the 3D Viewer and the contents of the picture can be adjusted dynamically using the mouse.

Selecting Parts of the Finite Element Model

Menu Route

display \downarrow

3d Viewer ... select parts

The THREED command draws volumes and facets created with the DEFINE command (page 4-96). By use of the TYPE and LABEL parameters, sub-sets of the entities can be selected in order to aid visualization of the model. The parameter TYPE can be set to FACET or VOLUME; the parameter LABEL can be set to individual labels or combinations of labels. The additional label, NOTAIR can be used to select all material name labels except AIR. Abbreviated label names can be used where the abbreviation is not ambiguous. Abbreviations can be followed by * to indicate

that all labels which match should be selected. An entity is drawn if it has any of the labels selected. For example, to display all facets with vector potential boundary conditions, LABE=A*-ALL could be used.

Labels are assigned automatically to parts of the model by the commands DEFINE, EXTEND, MODIFY and SLIP. Automatically assigned labels include ALL, material names, element types potential types and boundary condition types and functions. Additional labels can be given in two ways: the LABEL command (page 4-123) can be used to give any label to any part of the model and the CHECK command (page 4-18) can be used to assign the labels DEBUG to volumes with bad shapes and EXTERNAL to facets which are not shared by two volumes.

The part of the finite element mesh displayed can also be restricted by the parameters MESH, L1 and L2. The MESH parameter is used to select ALL meshes or one particular mesh (each DEFINE command creates a mesh with a new number). **MESH=NONE** can be used to omit the finite element mesh completely. L1 and L2 are used to select a subset of the layers of the mesh. L2 can be set to * to indicate the top layer.

Selecting Conductors

Menu Route	$\mathtt{DISPLAY} \downarrow$
	3d Viewer select parts $ ightarrow$ Conductors
	ightarrow Conductor numbers
	The THREED command draws the conductors including any symmetry copies. This can be controlled by the parameters COIL , C1 and C2 . +COII and -COIL switch the display of the conductors on and off, and C1 and C2 select a range of conductors for display. C2 can be set to * to indicate the highest numbered conductor.

Other parameters

Menu Route DISPLAY

3d Viewer ... command style

Three different types of picture can be displayed:

- **ELEMENT=NO** gives a wire-frame view without discretisation. This is always available.
- **ELEMENT=SURFACE** gives a solid colour view showing the selected surfaces (by facet label or volume label) and the surface discretisation.

I

This option can only be used after the MESH command (page 4-131) has been run successfully.

Views of facets, with LABEL set to a single potential name (AX, AY, AX, POTENTIAL, VOLTAGE or the incident potentials) are displayed as contour plots to show the values of the potential on the facets.

• ELEMENT=VOLUME, TYPE=VOLUME gives a solid colour view of each individual element. The elements are shown shrunk in size so that internal elements are also visible. This option can only be used after the FILL command (page 4-114) has been run successfully.

Arrows to show the direction of the volume vector properties can be drawn inside each volume. By default, the arrows are only drawn inside the conductors to show the current direction (**VECTOR=CONDUCTORS**) but they can also show the MATERIAL orientation, the **CURRENT** density direction or the **VELOCITY** direction. The arrow display can be switched off completely (**VECTOR=NO**).

The 3D Viewer Menus

The picture in the 3D Viewer can be manipulated using the mouse. The main control is the left mouse button. Moving the mouse with the left button pressed changes the view of the model. The way in which the view is changed and other options can be selected from a menu which pops up when the right mouse button is clicked. The menu options can be selected using the left mouse button.

3D Viewer Menu			
Translate	The model can be moved in any direction by dragging with the		
	left mouse button pressed. The cursor shape is \clubsuit .		
Rotate	The model can be rotated by dragging with the left mouse but- ton pressed. Movement up or down rotates the model around the horizontal axis. Movement from side to side rotates around the vertical axis. This is the default state when the 3D Viewer is started. The cursor shape is \bigcirc .		
Zoom	The model can be moved closer or further away by dragging with the left mouse button pressed. Movement up the window brings the model closer; movement down the window pushes the model further away. The cursor shape is \clubsuit .		
Views	The options in the Views sub-menu can be used to re-initialise the view or to rotate the model so it is being viewed from either the positive or negative ends of the major coordinate axes.		

4-173

3D Viewer Menu (continued)		
Options	The Options sub-menu can be used to switch on or off components of the display, e.g. the axes.	
Quit	The Quit option closes the 3D Viewer. It can be restarted using the THREED command.	

The TITLE Command

Menu Route

View↓ Title

Command Line Parameters

TITLE			
Default	Function		
none	A grap	phics window title.	
TL	Graph	ics window title position:	
	BC	Bottom centre.	
	BL	Bottom left.	
	BR	Bottom right.	
	тс	Top centre.	
	TL	Top left.	
	TR	Top right.	
YES	Title preservation switch:		
	NO	Only display title once.	
	YES	Display title on subsequent pictures as well.	
YES	First appearance switch:		
	NO	Display after next graphics window clear.	
	YES	Display immediately.	
NO	Date, time and page number switch:		
	NO	Date, time and page number not displayed.	
	YES	Date, time and page number displayed.	
YES	Display units and problem data:		
	NO	Units and Problem Data not displayed.	
	YES	Units and Problem Data displayed.	
	TITLE Default none TL YES YES	TITLE Default Functi none A grap TL Graph BC BL BR TC TL TR YES Title p NO YES YES First a NO YES NO Date, 7 NO YES YES Displa	

Notes

The TITLE command controls the display of titles and other information on the graphics window. The items which can be controlled are an additional title, and the default labelling of the graphics window with date, time and page number.

An additional title, STRING, can be POSITIONed at any one of 6 places on the graphics window. It can be displayed immediately (+NOW) or after the next graphics window clear (-NOW). It can be used once (-

KEEP) or kept for subsequent pictures (**+KEEP**). **STRING**s which contain spaces or commas or start with a non-alphabetic character must be enclosed in quotation marks (**'**).

- The display of the DATE, time and page number can be switched on or off on subsequent pictures with +DATE and -DATE.
- The display of unit settings and information about the model can be switched on or off on subsequent pictures with **+EXTRA** and **-EXTRA**.

The **TRANSFORM** Command

Menu Route

MODIFY↓

Transform labelled points

Command Line Parameters

Command	TRANSFORM		
Parameter	Default	Function	
LABEL	ALL	Point label.	
NEWX	Х	Expression for new X-coordinates.	
NEWY	Y	Expression for new Y-coordinates.	
NEWZ	Z	Expression for new Z-coordinates.	

The TRANSFORM command can be used to move a labelled set of points to new coordinates. The points must first be labelled using the LABEL command (page 4-123). Then the new coordinates for the points (NEWX, NEWY, NEWZ) can be given as values or as expressions in terms of their existing cartesian (X, Y, Z) or cylindrical polar (R, TH, Z) coordinates

The following should be noted:

- Whenever the LABEL command is used to add a label to or remove a label from a volume, facet or line, the same operation is also applied to the points which define the volume, facet or line. This enables groups of points to be labelled together.
- The SLIP command (page 4-149) adds the label SLIP to all the points on the slip surface.
- The **TRANSFORM** command cannot be undone unless the inverse transformation exists.

The WRITE Command

Menu Route

 $\mathtt{FILE} \downarrow$

Write pre-processor file

Command Line Parameters

Command	WRITE	
Parameter	Default	Function
FILE	none	Name of OPERA-3d pre processor data file.

The WRITE command writes an OPERA-3d pre processor data file. There is one parameter which defines the name of the FILE. If no file name extension is given, and extension *oppre* is assumed.

Pre processor data files consist of all the commands issued to the program including cursor commands, except that some erroneous cursor hits are omitted. Comments are included among the cursor commands to aid the user to follow the sequence of commands. Cursor commands are introduced by the character string curs. Comments have the character string ***** followed by the number of following comment lines.

• Example - the following is the beginning of a OPERA-3d pre processor data file:

```
DEFT
XY
0 100 0 100
* * * *
         1
 Start of point definition
CURS I 11 0.2461406E+03 0.1195938E+03 0.9931790E+02 0.3576628E+02
A 0 0 10 0 10 60
A 0 0 55 0 55 60
A 0 0 100 0 100 60
T, 0 0 100 0 0
L 0 0 100 0 60
0
       15 0.2475313E+03 0.5979688E+02 0.1001073E+03 0.1821565E+01
CURS X
        15 0.1654844E+03 0.2058125E+03 0.5353201E+02 0.8470982E+02
CURS X
CURS X 15 0.1293281E+03 0.1348906E+03 0.3300729E+02 0.4444981E+02
CURS X 15 0.8343750E+02 0.7787500E+02 0.6956694E+01 0.1208392E+02
CURS X
       15 0.6953125E+02 0.5423438E+02-0.9374268E+00-0.1336082E+01
CURS X
        15 0.1001250E+03 0.5423438E+02 0.1642964E+02-0.1336082E+01
CURS X
        15 0.1682656E+03 0.5423438E+02 0.5511083E+02-0.1336082E+01
        11 0.1682656E+03 0.5423438E+02 0.5511083E+02-0.1336082E+01
CURS I
L 0 0 10 0 30
L 0 0 100 0 30
CURS X 15 0.8900000E+02 0.6675000E+02 0.1011434E+02 0.5768625E+01
CURS X 15 0.1724375E+03 0.9734375E+02 0.5747906E+02 0.2313569E+02
CURS X 15 0.2308438E+03 0.1279375E+03 0.9063436E+02 0.4050275E+02
```

CURS	Q	6	0.2308438E+03	0.1279375E+03	0.9063436E+02 0.4050275E+02					
* * * *		1								
Stai	rt	of face definition								
CURS	С	9	0.2308438E+03	0.5006250E+02	0.9063436E+02-0.3704319E+01					
CURS	М	10	0.2308438E+03	0.1251563E+03	0.9063436E+02 0.3892393E+02					
CURS	С	9	0.1613125E+03	0.1835625E+03	0.5116376E+02 0.7207923E+02					
CURS	С	9	0.1265469E+03	0.1585313E+03	0.3142846E+02 0.5786982E+02					
CURS	М	10	0.1501875E+03	0.1029063E+03	0.4484847E+02 0.2629334E+02					
CURS	С	9	0.1613125E+03	0.5006250E+02	0.5116376E+02-0.3704319E+01					
CURS	F	11	0.1515781E+03	0.5006250E+02	0.4563788E+02-0.3704319E+01					
* * * *		1								
End	of	face	9							
CURS	С	9	0.1515781E+03	0.5006250E+02	0.4563788E+02-0.3704319E+01					
CURS	М	10	0.1515781E+03	0.1112500E+03	0.4563788E+02 0.3102981E+02					
CURS	С	9	0.1223750E+03	0.1404531E+03	0.2906023E+02 0.4760746E+02					
CURS	С	9	0.7648438E+02	0.7787500E+02	0.3009633E+01 0.1208392E+02					
CURS	М	10	0.8621875E+02	0.6814063E+02	0.8535518E+01 0.6558037E+01					
CURS	С	9	0.8621875E+02	0.5284375E+02	0.8535518E+01-0.2125494E+01					
CURS	F	11	0.8621875E+02	0.5284375E+02	0.8535518E+01-0.2125494E+01					
* * * *		1								
End	of	face	e							
CURS	С	9	0.8621875E+02	0.5284375E+02	0.8535518E+01-0.2125494E+01					
CURS	М	10	0.8621875E+02	0.6396875E+02	0.8535518E+01 0.4189801E+01					
CURS	С	9	0.7370313E+02	0.7092188E+02	0.1430809E+01 0.8136862E+01					
CURS	С	9	0.7370313E+02	0.5423438E+02	0.1430809E+01-0.1336082E+01					
CURS	F	11	0.7370313E+02	0.5423438E+02	0.1430809E+01-0.1336082E+01					
* * * *		1								
End	of	face	face							

VF-02-04-B2

OPERA-3D REFERENCE MANUAL PART 2

Vector Fields Limited 24 Bankside Kidlington Oxford OX5 1JE England

Copyright © 1999-2003 by Vector Fields Limited, England

This document was prepared using Adobe[®] FrameMaker[®].

UNIX is a trademark of X/Open Company Ltd. HPGL is a trademark of Hewlett-Packard Incorporated PostScript is a trademark of Adobe Systems Incorporated Windows 98/NT/2000/XP are trademarks of Microsoft Corporation X Window System is a trademark of Massachusetts Institute of Technology I-DEAS is a trademark of Structural Dynamics Reseach Corporation OpenGL is a trademark of Silicon Graphics Corporation

All other brand or product names are trademarks or registered trademarks of their respective companies or organisations.

Chapter 5 OPERA-3d Post Processor

Introduction

The OPERA-3d post processor displays and performs further calculations on results from electromagnetic field analysis programs including CAR-MEN, ELEKTRA, SCALA, SOPRANO, TEMPO and TOSCA. The analysis programs use finite elements to model three dimensional electromagnetic devices. The post processor provides facilities to view the finite element data, with superimposed contours of results and to process and display the results calculated along lines or on two dimensional areas, in addition to specific functions such as body forces and trajectory calculations.

The OPERA-3d post processor can be used interactively on PCs, UNIX workstations, or X-terminals having GLX support.

The program is used to display three dimensional finite element models from direct access database files created by the analysis programs. Alternatively, conductor only problems can be evaluated by entering the conductor data into the post processor directly using the CONDUCTOR command.

Many results databases can be available to the program (activated) at any one time. It is possible for the program to look for a field point in all the active files until it is found or to restrict its searching to one resident (loaded) file. This enables the calculation of particle trajectories which extend through a system of magnets which have been analysed separately. When activating a database a local coordinate system can be specified to enable the individual magnets to be orientated correctly with respect to each other. Reflection codes and symmetries can also be given to replicate the finite element mesh so that the complete model is available to OPERA-3d even if only a small section of it was analysed and the rest was implied by boundary conditions. When the OPERA-3d post processor is started or restarted with the CLEAR command, the program looks for a file called *opera.comi*, first in the current file directory and then in the user's home directory. If such a file exists it is read into the program as a \$ COMINPUT file (see section "Command Input Files" on page 2-25). This allows the user to reset the default values of certain commands, e.g. COLOUR and UNITS or define frequently used CONSTANTS and PARAMETERS each time the program is started.

Two methods of command and data entry are available:-

- 1. Menu system or GUI (Graphical User Interface) command selection and data specification are carried out under mouse control. Refer to the User Guide for information on the GUI.
- 2. Command line input command selection and data specification are carried out from the keyboard.
Post Processor Quick Reference Guide

The following is a complete list of the commands which can be entered in response to the console prompt 'OPERA-3d >'. Following sections contain complete descriptions of all the commands and sub-commands of the OPERA-3d post processor.

Commands for Keyboard Entry – Command Line

Command line entry is carried out in the console window.

Commands to make database mes available to Of EKA-5d.	
ACTIVATE	Activate a result database file, specifying reflections,
	symmetries and local coordinate system.
LOAD	Make an active database file resident or close a loaded
	database.
SHOW	List the names and some details of the active database
	files.
SYSVARIABLE	Add, delete or list system variables (solution vectors).
BHDATA	List the BH tables used in the loaded file.

• Commands to make database files available to OPERA-3d:

• Conductor definition and editing command:

CONDUCTOR	Define, modify, erase, list, import and export conductor
	data.

• Units selection command:

UNITS	Select units for display and evaluation

• Display commands:

SELECT	Select objects to be displayed.
THREED	Displays SELECT ion of mesh and conductors with surface contours.
WINDOW	Show or hide parts of the display.

• Field evaluation and display commands:

SET	Set field calculation options and local coordinate sys-
	tem for definition of field points.
POINT	Calculation of functions of the field at a point.
LINE	Calculation of functions of the field along a line.
ARC	Calculation of functions of the field along a circular arc
	defined by end points and centre.
CIRCLE	Calculation of functions of the field along a circular arc
	defined by radius, angles and local coordinate system.

Display the results from the ARC, CIRCLE or LINE commands. PLOT also calculates the line integral.
Fit Fourier series to ARC, CIRCLE or LINE results or evaluate fields on a sphere and fit Legendre polynomi- als.
Evaluate fields over a quadrilateral patch defined in cartesian (XYZ) coordinates.
Evaluate fields over a quadrilateral patch defined in cylindrical polar ($R\theta Z$) coordinates.
Display a contour maps or histograms of fields or field vectors evaluated with CARTESIAN or POLAR com- mands. MAP also calculates the surface integral.

• Field integration commands:

INTEGRATE	Integrate Maxwell Stress on SELECTed surface to give forces.
SURFACE	Integrate any field component on the SELECT ed surface.
BODY	Integrate body forces and torques on conductors: $\mathbf{J} \times \mathbf{B}$.
ENERGY	Calculate the stored energy, Lorentz force and power loss.
VOLUME	Integrate any field component over the volume of a

• Particle tracking commands:

TRACK	Calculate charged particle trajectories.
COMBINE	Combine magnetostatic and electrostatic fields for par- ticle tracking.
VIEW	Display particle trajectory intercepts.

• Interface and utility commands:

GRID	Evaluate fields over a 3D grid and put answers in a file.
ARITHMETIC	Combine values from two table files.
TABLE	Input tables of coordinates, etc., and output tables of coordinates, fields, etc.
IDEAS	Write or append mesh and results to an I-DEAS Universal File
GRAPH	Draw graphs of data in text files.

• Program management commands:

· · ·	
COLOUR	Enquire and set colours for the display.
TITLE	Control screen titles.
CLEAR	Clear program data and re-initialize all commands.

I

PICTURE	Copy the picture to the clipboard or a bitmap file.
PRINT	Send the picture to a printer.
MOUSE	Swap functionality of middle and right mouse buttons.
GUIOPTIONS	Set window preferences

• Ending the program:

- Ending the program.	
END	End OPERA-3d.

Menu System

The following is an overview of the first level menu structure. A brief description of the functionality of each item is also given.

<u>File View Options Fields Integrals Trajectories Tables Conductors Windows Help</u>

File

2	Open (activate+load)	Ctrl+O	activate and load an OPERA-3d database	page 5-22
	<u>R</u> eload active database		re-load in a previously loaded file	page 5-70
	List active databases		list details of active databases	page 5-92
	Clo <u>s</u> e loaded database		closes the currently loaded database	page 5-70
	Deac <u>t</u> ivate loaded database		closes and removes from the active list the currently loaded database	page 5-70
	Graph <u>B</u> H data		plot graphs of BH curves used in analysis	page 5-27
9	Print	Ctrl+P	sends the picture to a printer	page 5-83
Þ	<u>C</u> opy to clipboard Copy to <u>f</u> ile	Ctrl+C	copies the picture to the clipboard or a bit- map file	page 5-76
	Commands <u>i</u> n		read command input file	page 2-25
	Change <u>d</u> irectory		change the current directory	page 2-30
	Graph data in text file		plot graph of external data	page 5-53
	Clear <u>a</u> ll data		re-initialise the program	page 5-35
	R <u>e</u> cent files		activate and load recently used databases	page 5-22
	E <u>x</u> it		end post processing session	page 5-46

View

Select	select parts of model to be displayed	page 5-84
Default select and refresh	select according to simulation type and refresh display	page 5-84
e Repeat select and refresh	select without changing list of labels and refresh display	page 5-84

🔞 <u>3</u> d display	control the style and view of the display	page 5-104
<u>R</u> efresh	refresh the display	page 5-104
Parts of the display	show or hide parts of the display	page 5-124
Views	refresh with predefined views	page 5-104
<u> Ch</u> ange colours	change the colours used for the display	page 5-36
<u>T</u> itle	set a title for the display	page 5-104

Options

	User variable calculator	set and list user defined parameter and constants	page 2-21
	User defined string variable	set and list user defined string variables	page 2-23
≫f	Field calculation method	options which affect the way fields are	page 5-88
*	Field <u>p</u> oint options	calculated, the locations of field points and the angle around the ac cycle	
Θ	AC time		
麗	Add system variables	make available or delete additional sys-	page 5-96
	Delete system variables	tem variables and list the available variables	
	List variables in database		
	List variables in pro <u>g</u> ram		
	Include Jc in coils	include current density in coils as a sys- tem variable	page 5-91
SI CGS	Units	set the units to be used for all physical quantities	page 5-112
Ś	Toggle right mouse button	swap use of middle and right buttons	page 5-75

Fields

• Fields at a point	evaluate field at 3D coordinate	page 5-80
Fields on an <u>a</u> rc	evaluate field along an arc defined by ends and centre	page 5-24
• Fields on a <u>c</u> ircle	evaluate field along an arc defined by centre and radius	page 5-33
Fields on a straight line	evaluate field along a straight line	page 5-68
Plot graph of field values	display fields as a graph	page 5-77
Fit <u>F</u> ourier series to values	find Fourier coefficients to match values calculated	page 5-51

□ Fields on a	a ca <u>r</u> tesian patch	evaluate fields over a 2D surface defined in xyz	page 5-31
O Fields on a	a p <u>o</u> lar patch	evaluate fields over a 2D surface defined in r, θ, z	page 5-81
🕕 Contour or	vector <u>map</u>	display fields as a contour, vector map, or histogram	page 5-72
Fit L <u>e</u> gendr	re polynomials to values	evaluate fields on the surface of a sphere and fit Legendre polyno- mials to the values	page 5-51
Fields on t	the <u>s</u> urface of the model	display field contours on the sur- face of the model	page 5-104

Integrals

Energy, power and force	calculate stored energy, power loss	page 5-47
	and Lorentz forces by volume inte-	
	gration	
∭ Other volume integrals	integrate any quantity over vol-	page 5-121
	umes	
Maxwell stress on selected surface	integrate Maxwell stress on	page 5-65
	selected surface	
∬ Other surface integrals	integrate any quantity on selected	page 5-94
	surface	
E Lorentz forces in conductors	calculate Lorentz forces in con-	page 5-28
	ductors	

Trajectories

<u>C</u> alculate particle trajectories	calculate the trajectory of particles through the magnetic or electric fields	page 5-107
Display Particle trajectories Graph trajectories Intersect trajectories with patch Current density <u>map</u> Create <u>f</u> lux tubes	display and process calculated tra- jectories	page 5-115
Use magnetic and electric fields	use both magnetic and electric fields when calculating trajectories	page 5-38

Tables

Read and write <u>t</u> able files	create or process an OPERA-3d table file	page 5-101
Table of field values on a grid	calculate field values at a 3-dimen- sional grid of points and store them in a table file	page 5-56
Arithmetic on tables files	combine the values in 2 table files, creating a third	page 5-26
SDRC I-DEAS <u>U</u> niversal file	create or append to a Universal file	page 5-60

Conductors

■ New solenoid	define a new conductor	page 5-40
💁 New <u>r</u> acetrack		
🛌 New <u>b</u> edstead		
💊 New <u>h</u> elical end		
∑ New <u>c</u> onstant perimeter end		
New s <u>t</u> raight bar		
✓ New <u>a</u> rc		
🗑 New <u>8</u> -node brick		
<u>₩</u> New <u>2</u> 0-node brick		
Toggle conductor picking	allow graphical selection of conductors	page 5-40
Se <u>l</u> ect conductors	form a list of conductors for modify, list, erase, export or body forces	page 5-40
Modify selected conductors	modify list of conductors	page 5-40
<u>E</u> rase selected conductors	erase list of conductors	page 5-40
📄 List conductor <u>d</u> ata	list data of conductors in list or all conductors if list is empty	page 5-40
Import conductors from file	read a file of conductor data	page 5-40
Export conductors to file	store in a file conductors in list or all conduc- tors if list is empty	page 5-40

Windows

<u>C</u> lose window	close the window	
C <u>a</u> scade	arrange 2D and 3D windows in a cascade	
<u>T</u> ile	arrange 2D and 3D windows side by side.	
3D graphics	window for 3D views of the model	
2D graphics	window for graphs	
Tool <u>b</u> ars	choose which toolbars are visible	
Preferences	set console visibility and other window options	page 5-58

Help

Reference Manual	F1	Reference Manual	
About		show version number and support information	

Field Component Evaluation

The command decoder used with OPERA-3d is described fully in chapter 2. More details are given here concerning the way in which the post processor makes use of the expression analyser to evaluate output field components.

Several commands, THREED, FIT, MAP, POINT, PLOT, etc., have parameters COMPONENT, VX, VY, and VZ to define scalar and vector field components to be displayed. Expressions can be used to define these output field quantities with the variables being System Variables, user constants, user parameters and the parameters of the commands.

The expression for COMPONENT, VX, VY and VZ used with any command becomes the default value for all the commands which use those parameters. The initial value for COMPONENT is X and VX, VY and VZ are initially set to X, Y and Z. (These are used as defaults because they are always available.)

The **\$ CONSTANT** and **\$ PARAMETER** commands (page 2-21) can be used to perform further calculations on the results of the commands.

System Variables

System Variables hold the data which can be used for point values, line graphs, contours and further calculations in the OPERA-3d post processor. Some system variables, such as field point coordinates, are available for all analysis types but others only have values if they can be read from an analysis database. System variables of this second type are the results of analysis. For example, a TOSCA magnetostatics analysis returns the magnetic scalar potential, field strength, flux density and source field strength (a scalar and three vectors). Commands such as THREED, MAP and PLOT accessed via the GUI allow the user to select the commonly used system variables through pop-up menus. These are activated by the button on the right side of the COMPONENT entry box.

Most loading of system variables happens automatically within the program, variables being loaded when they are required. For example, for TOSCA magnetostatics, the potential, field strength and flux density are loaded when a database is activated and loaded. The source field strength is loaded automatically if integral coil fields are selected in the SET command (page 5-88).

However, the user might sometimes need a variable which is available in the database but has not yet been loaded, or, for reasons of efficiency, might want to delete some of the system variables which have been loaded. The SYSVARIABLE command (page 5-96) provides this functionality as well as listing the variables in the database and program. The calculation of the values of the source current density vector for field points inside conductors is selected using the SET command (page 5-88).

In post processing results from ELEKTRA-SS or SOPRANO-SS, the system variables represent the real and imaginary parts of complex quantities and additional expressions are defined to give the instantaneous values. For other types of analysis only the real parts are available.

When system variables are loaded the program follows this procedure:

- Scalars, for example the magnetic scalar potential, POT: The program reads the real part (RPOT). If the imaginary part (IPOT) is also available it will be read as well. The following expressions will be defined:
 - Real scalars:

POT=RPOT

- Complex scalars:

```
POT=RPOT*COST+IPOT*SINT
POT0=SQRT(RPOT**2+IPOT**2)
POTP=ATAN2D(IPOT;RPOT)
```

where **COST** and **SINT** are calculated from the value of AC time defined by the **SET** command (page 5-88).

The following variables can be used by the user: POT, RPOT and (for steady state ac analyses) IPOT, POT0 and POTP.

- Vectors, for example the magnetic field strength, **H**: The program reads each component (real and imaginary parts if available) and defines the following expressions.
 - Real vectors:

HX=RHX

- Complex vectors:

```
HX=RHX*COST+IHX*SINT
HX0=SQRT(RHX**2+IHX**2)
HXP=ATAN2D(IHX;RHX)
```

and similarly for the Y and Z components. It also defines the magnitude and the radial and azimuthal components:

```
HMOD=SQRT(HX**2+HY**2+HZ**2)
HR=(HX*X+HY*Y)/R
HT=(HX*Y-HY*X)/R
```

The following variables can be used by the user: HX, RHX, HY, RHY, HZ, RHZ, HMOD, HR, HT and (for steady-state ac analyses) IHX, IHY, IHZ, HX0, HY0, HZ0, HXP, HYP and HZP.

 Edge or Face Vectors, A_ and E_: The program interpolates the edge or face values to provide vector quantities, e.g E_X, E_Y and E_Z and E_MOD, at each field point.

The following variables can be used by the user: E_X, RE_X, E_Y, RE_Y, E_Z, RE_Z, E_MOD.

In the lists of variables below, only the stem names are given (e.g. H); it should be understood that all the components are also implied.

When system variables are removed, they are marked as no longer being available within the expression analyser software.

Unit conversion is performed on all system variables.

System Variables Defined in the Software

Field point geometry		
X, Y, Z	Field point coordinates	
TX, TY, TZ	Tangential unit vector to lines	
NX, NY, NZ	Normal unit vector to surfaces	
NODE	Number of the node nearest to the field point.	
ELEMENT	Element containing the field point	

Source Quantities		
JC	Source current density	vector
HC	Coercive field	vector
VEL	Velocity	vector
CHARGE	Charge density	scalar

System Variables in Analysis Databases

System Variables in Analysis Databases				
CARME	CARMEN			
POT	Magnetic scalar potential	scalar		
V	Electric scalar potential	scalar		
Α	Magnetic vector potential	vector		
Н	Magnetic field strength	vector		
HS	Source magnetic field strength	vector		
В	Magnetic flux density	vector		
E	Electric field strength	vector		
J	Eddy current density	vector		

I

System Variables in Analysis Databases (continued)			
ELEKTI	RA		
V	Electric scalar potential	scalar	
A_	Magnetic vector potential	vector	
DADT_	Time derivative of magnetic vector potential	vector	
н	Magnetic field strength	vector	
HS	Source magnetic field strength	vector	
В	Magnetic flux density	vector	
E	Electric field strength	vector	
J	Eddy current density	vector	
ERRB	Error in flux density	vector	
ERRJ	Error in current density	vector	
SCALA :	and TOSCA electrostatics		
V	Electric scalar potential	scalar	
E	Electric field strength	vector	
D	Electric flux density	vector	
ERRD	Error in flux density	vector	
SOPRAN	NO-EV		
E_	Edge values of electric field strength	vector	
В	Nodal values of magnetic flux density	vector	
D	Electric flux density	vector	
E	Nodal values of electric field strength	vector	
н	Magnetic field strength	vector	
SOPRAN	NO-SS		
E_	Edge values of electric field strength	vector	
В	Magnetic flux density	vector	
D	Electric flux density	vector	
E	Electric field strength	vector	
н	Magnetic field strength	vector	
J	Current density	vector	
TEMPO			
Т	Temperature	scalar	
DT	Temperature gradient	vector	
Q	Heat flux	vector	
ERRQ	Error in heat flux	vector	
TOSCA	current flow		
V	Electric scalar potential	scalar	
E	Electric field strength	vector	
J	Current density	vector	
ERRJ	Error in current density	vector	

	System Variables in Analysis Databases (continued)			
	TOSCA magnetostatics			
	POT	Magnetic scalar pot	ential	scalar
	Н	Magnetic field stren	gth	vector
	HS	Source magnetic fie	ld strength	vector
	B	Magnetic flux densi	ty	vector
	ERRB	Error in flux density	1	vector
	In TOSCA cui integral fields	rrent flow, the magne are selected ("The S	etic field strength, H is al ET Command" on page 3	lso available if 5-88)
Material properties at field point	Material prope calculated from permeability in	erties, permeability, permeability, permeability, permeability, permeability, permeability, permeability, perme mathematical structure of the second s	ermittivity and conductiv d quantities. For example	ity can only be e, the isotropic
	BMOD/HMOD			
Solution values	These are set when eigenvalue, steady-state ac or transient results are act vated.			esults are acti-
	Se	olution values		
	A	NGLE Rotor angle		
	FI	REQ Frequency or	rotational frequency (rpi	m)
	Т	TIME Transient time	3	
	The variable (base.	CASES holds the nu	mber of simulations in th	he active data-
Local coordinate system	The field vectors are defined with respect to the Global Coordinate Syst The rotation matrix defined by the Euler angles of the SET comm (page 5-88) can also be used in the expressions. The local X componer the flux density is given by		dinate System. ET command component of	
	COMP=BX*R	OTL11+BY*ROTL2	1+BZ*ROTL31, etc.	
	Local coordi	nate system		
	ROTL11, RC)TL12,, ROTL33	Rotation matrix to co	onvert global
			field vectors to local co	oordinate sys-
			tem.	

Results of commands

After the commands BODY, ENERGY, INTEGRATE, MAP, PLOT, THREED and VIEW system variables are updated with the results of the calculations.

Results of comm	Commands		
MINIMUM, MAXIMUM	Extreme values	MAP, PLOT, THREED	
FX, FY, FZ	Forces	BODY ENERGY	
FMOD	Magnitude of force: SQRT(FX**2+FY**2+FZ**2)	INTEGRATE	
TORQX, TORQY, TORQZ	Torques	BODY,	
TORQMOD	SQRT(TORQX**2+TORQY** +TORQZ**2)	INTEGRATE	
ENERGY, POWER, VOLUME COENERGY, NLENERGY, ELECENER, HYSENERGY, HYSPOWER	Energy, power, etc. integrals including non-linear, high-fre- quency and complex permeability analyses effects	ENERGY	
INTEGRAL	Field integral	MAP, PLOT, SURFACE, VOLUME	
AREA	Surface area	CONDUCTOR, SURFACE	
VOLUME	Volume	VOLUME	
J	Charged beam current density	VIEW	
PJ	Charged beam power density	VIEW	
TOF	Time of flight of particle	VIEW	
TXBEAM, TYBEAM, TZBEAM	Tangential unit vector to a trajec- tory	VIEW	
VELX, VELY, VELZ	Partcle velocity	VIEW	
X0BEAM, Y0BEAM, Z0BEAM	Coordinates at centre of a beam	VIEW	
XSTART, YSTART, ZSTART	Coordinates at start of a trajectory	VIEW	

I

Constants

Constants		
PI	π	
MU0	μ_0 , permeability of free space (SI)	
EPSILON0	ε_0 , permittivity of free space (SI)	
С	c, speed of light (SI)	

User constants

User constants are defined and examined using the **\$ CONSTANT** command. They allow the current value of system variables or expressions to be stored for use in subsequent calculations (see page 2-21)

User parameters

User parameters are defined and examined using the **\$ PARAMETER** command. They allow expressions to be stored. The value of user parameters is recalculated from the expression each time it is referenced using the current values of any other parameters or variables (see page 2-21)

Expressions

Expressions cannot exceed 250 characters, since they cannot be continued on subsequent lines. Full details of the operators and functions allowed in expressions are described on page 2-8. The **\$ PARAMETER** command should be used to 'program' OPERA-3d if more complicated expressions are needed.

Examples

Two examples are given here. The first example gives expressions to calculate the Maxwell Stress on a surface using the integral given by the MAP command. The force densities are given by #FX, #FY and #FZ. The parameters #X0, #Y0 and #Z0 should be previously set to the action point for the torques, #TQX, #TQY and #TQZ. The expressions given assume that the units are set to SI (see "The UNITS Command" on page 5-112).

- Example to calculate forces and torques (output from the program is not shown):
- \$ para #hn hx*nx+hy*ny+hz*nz
- \$ para #bdoth bx*hx+by*hy+bz*hz
- \$ para #fx bx*#hn-nx*#bdoth/2

```
$ para #fy by*#hn-ny*#bdoth/2
$ para #fz bz*#hn-nz*#bdoth/2
$ para #tqx (y-#y0)*#fz-(z-#z0)*#fy
$ para #tqy (z-#z0)*#fx-(x-#x0)*#fz
$ para #tqz (x-#x0)*#fy-(y-#y0)*#fx
```

The above commands are only valid if the field points used are all in air. If the integration surface passes through ferromagnetic material, the following set of commands should be used instead. These calculate the forces assuming an infinitely thin gap at the integration surface.

```
$ para #han (nx*bx+ny*by+nz*bz)/mu0
$ para #hanx nx*#han
$ para #hany ny*#han
$ para #hanz nz*#han
$ para #hax hx-nx*(nx*hx+ny*hy+nz*hz)+#hanx
$ para #hay hy-ny*(nx*hx+ny*hy+nz*hz)+#hany
$ para #haz hz-nz*(nx*hx+ny*hy+nz*hz)+#hanz
$ para #haz hz-nz*(nx*hx+ny*hy+nz*hz)+#haz
$ para #hn #hax*nx+#hay*ny+#haz*nz
$ para #hm2 #hax*#hax+#hay*#hay+#haz*#haz
$ para #fx (2*#hax*#hn-nx*#hm2)*mu0/2.0
$ para #fz (2*#haz*#hn-nz*#hm2)*mu0/2.0
```

It is sometimes necessary to store the current value of a system variable or expression. The **\$ CONSTANT** command copies the current value of a variable or expression into a user defined name. Thus to plot the homogeneity of the flux density along a line the following set of commands could be used. First, the field at the reference point (10,0,0) is calculated and stored in a constant.

Example - to calculate the homogeneity of flux density poin 10 0 0 by
\$ cons #byrf by

The value of **BY** assigned to the constant #**BYRF** corresponds to value at the point (10, 0, 0). Next the homogeneity can be calculated.

line 0 0 0 20 0 0 100
plot comp=(by-#byrf)/#byrf

Post Processor Data Files

The OPERA-3d post processor reads data from the direct access database files created by the Modeller or pre processor and updated by the analysis programs. Other files can be created and read by the post processor for use by the results display commands. See

- "The CONDUCTOR Command" on page 5-40 for details of conductor data files,
- "The ENERGY Command" on page 5-47 for multipole files,
- "The GRAPH Command" on page 5-53 for graph files,
- "The IDEAS Command" on page 5-60 for Universal files,

and below for databases, grid files, table files and track files.

OPERA-3d Database Files

OPERA-3d database files contain all the information used and calculated by the analysis programs. This includes the finite element mesh including boundary conditions, the material properties and BH curves, the conductors and solution options. These files can only be read by OPERA-3d. The SHOW command can be used to list the contents of database (page 5-92).

GRID files

The **GRID** command writes a limited number of field values to a file to enable users to interface to their own post processing software. The files can be text or binary.

The following items are output: X, Y, Z and up to 9 other items which can be specified by the user.

In *text* files the table file format is used (see page 5-19).

Binary files contain one record per field point with up to 12 DOUBLE PRECISION (8 byte) values.

TABLE Files

Table file format is used by the following commands:

ARITHMETIC	reads values from two table files and combines them, writing the answers to a third table file.
FIT	reads coordinates and field values from a file and fits a Fourier series.
GRID	writes coordinates and field values at a grid of points to a table file.
MAP	reads coordinates and field values from a table file and draws a contour or vector map.
PLOT	reads coordinates and field values from a table file and plots a graph.
TABLE	read and writes table files.

Table files are text files with some formatted and some free-format data.

Record type 1: The first line of a table file contains the number of data points in the file and a version number. The number of points is specified as 3 numbers, so that 1, 2 or three dimensional arrays of points can be stored. For example, a total of 1000 points could be stored as a line with record 1 containing

```
1000 1 1 1
```

or as a 2 dimensional patch, with 40 points on side 1 and 25 points on side 2, indicated by

```
40 25 1 1
```

or as a 3 dimensional grid of points:

10 10 10 1

The 4th number on record type 1 holds the version number of the table file. Zero is assumed when the version number is missing. Version 0 files were written by OPERA-3d versions less than 9 and can be read by all versions. Version 1 files are read/written by OPERA-3d version 9 and above.

The record is free-format.

Record type 2: The second record is repeated up to 12 times, once for each column in the file.

• Version 0: The record holds an integer (the column number), followed by a space, and then the column description of up to 80 characters. In most contexts, the string is compared to X, Y or Z to identify the coordinates, but otherwise is not decoded.

The equivalent FORTRAN format is (12, 1X, A).

Version 1: The record holds an integer (the column number), followed by a space, and then the column description of up to 255 characters. The column description consists of two parts: the column name followed by the unit expression in square brackets. In most contexts, the name is compared to X, Y or Z to identify the coordinates, but otherwise is not decoded. The unit expression should be in terms of the unit conversion factors (page 5-114).

The equivalent FORTRAN format is (12, 1X, A).

Record type 3:

• Version 0: The third record has the same format as record 2 but the integer value must be zero. It indicates the end of the list of column headings and holds in the character string the unit name associated with the file. The units names which are recognised are: [CGS], [METRE], [MM], [MICRON] and [INCH].

The equivalent FORTRAN format is (12, 1X, A).

• Version 1: The third record contains a single zero to indicate that there are no more column headings.

The equivalent FORTRAN format is (12).

Record type 4: The final record contains the data and is repeated for as many points as indicated by the product of the first 3 numbers in record 1. There should be up to 12 values on each line, as many as the column headings give by record 2. In version 1 table files, the values are stored in OPERA internal units and the values are scaled by the unit expressions when they are used by the FIT, MAP and PLOT commands.

The following is an example of a typical data file:

```
12 1 1 1
1 X [LENGU]
2 Y [LENGU]
3 Z [LENGU]
4 TY*BZ-TZ*BY [FLUXU]
5 TZ*BX-TX*BZ [FLUXU]
6 TX*BY-TY*BX [FLUXU]
0
0.10000 0.26000 0.12625 0.49232E-03 -0.12105E-03 0.13192E-03
0.20000 0.26000 0.12625 0.49232E-03 0.12105E-03 0.13192E-03
0.30000 0.26000 0.12625 0.16246E-02 -0.10484E-03 0.16246E-02
0.40000 0.26000 0.12625 0.16246E-02 0.10484E-03 0.16246E-02
0.50000 0.26000 0.12625 0.13180E-03 -0.11982E-03 0.49189E-03
0.60000 0.26000 0.12625 0.13180E-03 0.11982E-03
                                                           0.49189E-03
0.70000 0.26000 0.12625 -0.13180E-03 -0.11982E-03
                                                           0.49189E-03
0.80000 0.26000 0.12625 -0.13180E-03 0.11982E-03 0.49189E-03
0.90000 0.26000 0.12625 -0.16246E-02 -0.10484E-03 0.16246E-02
1.00000 0.26000 0.12625 -0.16246E-02 0.10484E-03 0.16246E-02
1.10000 0.26000 0.12625 -0.49232E-03 -0.12105E-03 0.13192E-03
1.20000 0.26000 0.12625 -0.49232E-03 0.12105E-03 0.13192E-03
```

TRACK files

The post processor **TRACK** command and the SCALA program store the trajectory coordinates in a binary unformatted file. Coordinates are in cm and velocities in cm s⁻¹. There is no indication of the number of tracks in the file; the data must be read with end and error control to detect when all data has been read. The contents of the file for each track is as follows:

Record 1: 20 INTEGER values, ITRAK. Most ITRAK values can be ignored; the following entries are important. ITRAK(1) is the number of steps. ITRAK(3) is the emitter number. ITRAK(7) is 0 (local coordinates) or 1 (global coordinates) Record 2: 20 DOUBLE PRECISION values, RTRAK.

RTRAK(1) is the current associated with the track.

RTRAK(2) is the mass. RTRAK(3) is the charge. RTRAK(4) is the step-length RTRAK(5-20) can be ignored.

Record 3: NSTEP DOUBLE PRECISION values, x coordinates of the track points. Record 4: NSTEP DOUBLE PRECISION values, y coordinates of the track points. Record 5: NSTEP DOUBLE PRECISION values, z coordinates of the track points. Record 6: NSTEP DOUBLE PRECISION values, x-component of the velocity. Record 7: NSTEP DOUBLE PRECISION values, y-component of the velocity. Record 8: NSTEP DOUBLE PRECISION values, z-component of the velocity. On UNIX systems, if NSTEP is greater than 500, records 3 to 8 are blocked into sub-records which contain 500 DOUBLE PRECISION words.

The **ACTIVATE** Command

Icon



Menu Routes

File ↓ Open Recent files

Command Line Parameters

Command	ACTIVATE			
Parameter	Default	t Function		
FILE	none	Name of dat	abase file.	
XORIGIN	0	X-coordinat	e of the coordinate system origin.	
YORIGIN	0	Y-coordinate	e of the coordinate system origin.	
ZORIGIN	0	Z-coordinate	e of the coordinate system origin.	
THETA	0	Euler angle	defining the coordinate system.	
PHI	0	Euler angle	defining the coordinate system.	
PSI	0	Euler angle	defining the coordinate system.	
SYMMETRY	1	Rotational s	ymmetry around local Z axis.	
RXY		Reflection in local XY plane.		
		NO	No reflection.	
		YES	Reflection with zero Z field.	
		INVERSE	Reflection with zero X and Y fields.	
RYZ	NO	Reflection in local YZ plane.		
		NO	No reflection.	
		YES	Reflection with zero X field.	
		INVERSE	Reflection with zero Y and Z fields.	
RZX	NO	Reflection in	n local ZX plane.	
		NO	No reflection.	
		YES	Reflection with zero Y field.	
		INVERSE	Reflection with zero Z and X fields.	
CASE	1	Simulation number or * for the last simulation.		

Notes The ACTIVATE command opens and checks a database file created by one of the analysis programs. The parameter FILE sets the file name. If no file name extension is given an extension of *op3* is assumed.

The ACTIVATE command also allows a local coordinate system and symmetry codes to be set. The local coordinate system is defined by its origin (XORIGIN, YORIGIN, ZORIGIN) and Euler angles (THETA, PHI, PSI). This repositions the whole model in space as if it had been defined in this position.

The symmetry codes can be used to create the complete model from the section which was analysed by rotation in the local Z axis and reflection in the local coordinate planes. Three types of symmetry are available.

- **SYMMETRY=***n* creates *n* copies by rotating through 360/*n* degrees around the local Z axis. The sign of the parameter **SYMMETRY** determines the direction of the field in the copies of the model. If **SYMME**-**TRY** is negative the field direction is reversed in alternate copies.
- **R***uv***=YES** should be used when the field normal to the local *uv* plane is zero. The program reflects the geometry in the *uv* plane of the local system and inverts the sign of the normal field in the reflected copy.
- **R***uv***=INVERSE** should be used when the tangential field in the local *uv* plane is zero. The program reflects the geometry in the *uv* planes of the local system and inverts the sign of the tangential field in the reflected copy.

The field reflections apply to the principal field of the simulation, i.e. **H** for magnetic field problems (CARMEN, ELEKTRA, SOPRANO and TOSCA), **E** for electric field problems (SCALA and TOSCA) and ∇T for thermal problems (TEMPO).

For databases with more than one simulation, the CASE parameter should be set to choose which simulation should be activated. CASE=* indicates the last simulation in the database. After activating a database, the system variable CASES is updated with the number of simulations in the database.

Up to 100 files can be active at a time, each with its own coordinate system. In this way devices which form a complicated system, but which can be analysed separately can be linked together for post processing. The SEARCH parameter on the SET command controls the way in which multiple active database files are searched for field points (see page 5-88).

In menu mode, the ACTIVATE command is automatically followed by the LOAD command.

The **ARC** Command

Summary	Calculate fields	along a	circular are	defined by	centre and end i	noints
Summury	Calculate fields	along a	circular are	Lucificu by	centre and end p	joints.

Icon



Menu Route

Fields \downarrow Fields on an arc

Command Line Parameters

Command	ARC	
Parameter	Default	Function
X1	none	X-coordinate of the first point on the arc.
Y1	none	Y-coordinate of the first point on the arc.
Z1	none	Z-coordinate of the first point on the arc.
X2	none	X-coordinate of the last point on the arc.
Y2	none	Y-coordinate of the last point on the arc.
Z2	none	Z-coordinate of the last point on the arc.
XC	0	X-coordinate of the centre of curvature.
YC	0	Y-coordinate of the centre of curvature.
ZC	0	Z-coordinate of the centre of curvature.
NP	10	Number of steps between the first and last points, i.e NP+1 points.

Notes

The ARC command evaluates field quantities along a circular arc (Figure 5.1, on page 5-69) for use by the FIT and PLOT commands (see page 5-51 and page 5-77). Its function is similar to that of the CIRCLE command (see page 5-33). For each field point all the currently available system variables are calculated and stored ("System Variables" on page 5-10).

The arc is specified by its end points (X1,Y1,Z1 and X2,Y2,Z2) and centre of curvature (XC,YC,ZC). The end points and centre of curvature must not be colinear, otherwise the plane of the arc cannot be determined. The minor arc is always chosen. Major arcs can be specified with the CIRCLE command.

The positions of the end and centre points are affected by any local coordinate system defined with the SET command (page 5-88). The vector field quantities are evaluated with respect to the Global Coordinate System for the active file.

The field quantities are evaluated at NP+1 points along the arc.

It is possible to store the evaluated field quantities in a file using the TABLE command (page 5-101).

The **ARITHMETIC** Command

Summary	Combine data in two table files and creates a third.
Menu Route	Tables \downarrow

Arithmetic on table files

Command Line Parameters

Command	ARITHMETIC	
Parameter	Default	Function
FILE1	none	Name of first input data file.
FILE2	none	Name of second input data
		file.
FILE3	none	Name of output data file.
VALUE	VAL1+VAL2	Expression for values in out-
		put data file.
NAME1,, NAME12	names from FILE1	Names for the columns in
		output data file.
UNIT1,, UNIT12	units from FILE1	Units for the columns in out-
		put data file.

Notes

The **ARITHMETIC** command combines the values in two *table* files and creates a new file of the same type. The format of the files is described in section "TABLE Files" on page 5-19.

The two input files, FILE1 and FILE2, must contain the same number of records in the same configuration. For example, if they both represent points on a 2-dimensional patch, they must have the same numbers of points in the two directions given on the first record. They must also contain the same number of columns.

By default the output file, FILE3, is created using the numbers of fields points, the column names and the units from FILE1. The column names and units can be replaced using parameters NAME1 to NAME12 and UNIT1 to UNIT12. For each column in each line of numerical data, the value from FILE1 is assigned to variable VAL1 and the value from FILE2 to VAL2 and the result of evaluating the expression given by VALUE is written to FILE3. Any columns with names X, Y or Z are not processed. The value from FILE1 is used.

The **BHDATA** Command

Summary	Plot graphs of BH characteristic	s used in analysis
Summury	i lot graphs of bit characteristic	s used in analysis.

 Menu Route
 File ↓

 Graph BH data

Command Line Parameters

Command	BHDAT	A	
Parameter	Default	Functior	n
MATERIAL	1	Material	l name or number.
DIRECTION	Х	Direction	on for anisotropic materials (X, Y or Z).
OPTION	VIEW	Options:	:
		VIEW V	View graphs of characteristic and its derivatives.
		LIST I	List values of B and H.

The BHDATA command displays the material characteristics used by the analysis programs. The parameter, MATERIAL, specifies the material name or number. A list of material names can be obtained with MATE-RIAL=!.

The two **OPTIONS** allow:

- VIEW: display graphs of **B** values, **B** interpolated, **M**, μ , and $\frac{\partial \mu}{\partial \mathbf{H}}$ against **H**. The data can be also exported to a BH file using the Export option from the BH Viewer's File menu or the **G** icon on the toolbar.
- LIST: list values of **B** and **H**.

For isotropic or laminated materials, one curve is displayed; for anisotropic materials, the curve for the specified local coordinate **DIRECTION** is displayed.

The **BODY** Command

Summary	Calculate Lorentz forces in conductors.
	Culculate Bolente forces in conductors.

Icon



Menu Route	Integrals \downarrow			
	Lorentz	forces	in	conductors

Command Line Parameters

Command	BODY			
Parameter	Default	Function		
N1	2	Number of gau	uss points in direction 1.	
N2	2	Number of gau	uss points in direction 2.	
N3	2	Number of gauss points in direction 3.		
X0	0	X coordinate of point of action for torque.		
Y0	0	Y coordinate of point of action for torque.		
Z0	0	Z coordinate of point of action for torque.		
TAVERAGE	YES	Time-average switch.		
		NO	Calculate integrals at time of SET command.	
		YES	Calculate time-average integrals.	

Notes

The BODY command integrates the body forces and torques on the source conductors using $\mathbf{J} \times \mathbf{B}$. Each section of each conductor is represented by an 8 or 20 node finite element. Gaussian quadrature integration with a selectable number of Gauss points (in the range 1 to 10, or values 16 or 32) is used to calculate the forces.

The command operates as part of a 2-stage process:

- 1. form a list of conductors using the CONDUCTOR command (page 5-40). An empty list implies all conductors.
- 2. integrate forces using the **BODY** command.

The parameters N1, N2 and N3 specify the numbers of Gauss points in each direction. N3 is the number of Gauss points in the direction of current flow.

The torque calculation uses the parameters X0, Y0 and Z0 to define the fixed point.

Unless **BODY** is used for conductor only problems it is essential that the conductors are completely enclosed in the finite element mesh. If the mesh does not completely contain the conductor the **SYMMETRY** and reflection parameters of the **ACTIVATE** command must be used to recreate the complete model (page 5-22).

The system variables FX, FY and FZ and TORQX, TORQY and TORQZ are updated with the values of force and torque.

The program calculates the force acting at the centroid of each section of each conductor. The coordinates and force on each section are listed in the file *Opera3d_Post_nn.lp*. The total force on each conductor and the total force on all conductors are displayed and listed in the file. If a conductor has reflections or symmetries, all copies are treated as one conductor.

Users should critically examine the forces to ensure that the net force on the conductor set has not arisen from field cancellation errors. The accuracy of force calculations can be enhanced by use of the integral conductor field recovery option (**SET COIL=INTEGRATE**, page 5-88). This option uses the values of **CURD** and **TOLERANCE** in the conductor definition which can be reset using the **CONDUCTOR** command (page 5-40). Changing **CURD** allows conductors to be switched on or off during the force calculations in order to find the self and mutual components of the force. Changing **TOLERANCE** further affects the accuracy of the conductor field calculations.

In steady-state alternating current problems, the force is a function of time with the form

$$F_{r} = A + B\cos 2\omega t + C\sin 2\omega t \tag{5.1}$$

The values *B* and *C* have little meaning on their own. The time-average value, *A* is the value commonly required. This can be calculated directly using the +TAVERAGE option.

The values of *B* and *C* can be found by setting the times to 0, 45 and 90 (see "The SET Command" on page 5-88), to give values of F_x at each time: F_0 , F_{45} and F_{90} with the **-TAVERAGE** option.

$$A = \frac{F_0 + F_{90}}{2}$$

$$B = \frac{F_0 - F_{90}}{2}$$

$$C = F_{45} - A$$
(5.2)

The following commands can be used to achieve this:

• Example - time-average F_x :

```
set time=0
body -tave
$ cons #fx0 fx
set time=45
body
$ cons #fx45 fx
set time=90
body
$ cons #fx90 fx
$ para #fxa 0.5*(#fx0+#fx90)
$ para #fxb 0.5*(#fx0-#fx90)
$ para #fxc #fx45-#fxa
```

The values F_0 , F_{45} and F_{90} have little meaning on their own.

The **CARTESIAN** Command

|--|

Icon



Menu Route

Fields \downarrow Fields on a cartesian a patch

Command Line Parameters

Command	CARTE	SIAN
Parameter	Default	Function
X1	0	X-coordinate of the first corner of the surface.
Y1	0	Y-coordinate of the first corner of the surface.
Z1	0	Z-coordinate of the first corner of the surface.
X2	none	X-coordinate of the second corner of the surface.
Y2	none	Y-coordinate of the second corner of the surface.
Z2	none	Z-coordinate of the second corner of the surface.
X3	none	X-coordinate of the third corner of the surface.
Y3	none	Y-coordinate of the third corner of the surface.
Z3	none	Z-coordinate of the third corner of the surface.
X4	none	X-coordinate of the fourth corner of the surface.
Y4	none	Y-coordinate of the fourth corner of the surface.
Z4	none	Z-coordinate of the fourth corner of the surface.
X5	none	X-coordinate of the mid-point on side 1.
Y5	none	Y-coordinate of the mid-point on side 1.
Z5	none	Z-coordinate of the mid-point on side 1.
X6	none	X-coordinate of the mid-point on side 2.
Y6	none	Y-coordinate of the mid-point on side 2.
Z6	none	Z-coordinate of the mid-point on side 2.
X7	none	X-coordinate of the mid-point on side 3.
Y7	none	Y-coordinate of the mid-point on side 3.
Z7	none	Z-coordinate of the mid-point on side 3.
X8	none	X-coordinate of the mid-point on side 4.
Y8	none	Y-coordinate of the mid-point on side 4.

Command	CARTE	CARTESIAN (continued)		
Parameter	Default	Function		
Z8	none	Z-coordinate of the mid-point on side 4.		
CORNERS	4	Number of points defining surface: 4 or 8.		
N1	10	Number of points on sides 1 and 3.		
N2	10	Number of points on sides 2 and 4.		

Notes

The CARTESIAN command evaluates field quantities on 4 or 8-noded surface patches. The results can be displayed by the MAP command (page 5-72). For each field point all the currently available system variables are calculated and stored (see "System Variables" on page 5-10).

The patch is specified by its corner points (X1,Y1,Z1, X2,Y2,Z2, X3,Y3,Z3 and X4,Y4,Z4) and optional mid-side points (X5,Y5,Z5, X6,Y6,Z6, X7,Y7,Z7 and X8,Y8,Z8). The mid-side points are ignored if CORNERS=4, and included if CORNERS=8. Point 5 is on side 1 (point 1 to point 2); point 6 is on side 2 (point 2 to point 3); point 7 is on side 3 (point 3 to point 4); point 8 is on side 4 (point 4 to point 1).

The positions of the points are defined in the local coordinate system defined with the SET command (page 5-88). The field vectors are evaluated in the Global Coordinate System.

The field quantities are evaluated at N1*N2 points. The coordinates of points other than the corner and mid-side points are found by interpolation using 4 or 8-noded isoparametric two-dimensional finite element shape functions.

It is possible to store the evaluated field quantities in a file using the TABLE command (page 5-101).

The **CIRCLE** Command

Carrier and	Coloulata	fields along a	airoular ara	manified by	radius and	malac
Summary	Calculate	neius along a	circular are s	specified by	autus attu a	ingles.

Icon



Menu Route Fields ↓ Fields on circle

Command Line Parameters

Command	CIRCLE	
Parameter	Default	Function
RADIUS	none	Radius of circular arc.
TH1	0	Azimuthal-coordinate of the first point on the arc.
TH2	360	Azimuthal-coordinate of the last point on the arc.
ZC	0	Axial coordinate of arc.
NP	10	Number of steps between the first and last points, i.e NP+1 points.

Notes

The CIRCLE command evaluates field quantities along a circular arc (Figure 5.1, on page 5-69) for use by the FIT and PLOT commands (page 5-51 and page 5-77). Its function is similar to that of the ARC command (page 5-24), except that it does allow arcs of 180 to 360 degrees to be defined. For each field point all the currently available system variables are calculated and stored (see page 5-10).

The circular arc is specified by its radius (RADIUS), the azimuthal coordinates of its end points (TH1 and TH2) and the axial coordinate (ZC). The plane of the arc is the local XY plane through local coordinates (0, 0, ZC), using the local coordinate system defined with the SET command (see page 5-88).

The GUI dialog box issues a SET command to define a local coordinate system for the field calculations, followed by the CIRCLE command. Following the CIRCLE command this local coordinate system will remain in affect for further field calculations until reset using the SET command.

The vector field quantities are evaluated with respect to the Global Coordinate System for the active file. The field quantities are evaluated at NP+1 points along the arc.

It is possible to store the evaluated field quantities in a file using the TABLE command (page 5-101).

The **CLEAR** Command

Summary	Clear all data and re-initialise all commands
Summury	Cieur un dute une re initianse un communas

Menu Route File ↓ Clear all data

Command Line		
Parameters	Command	CLE
	No Parame	ters

Notes The CLEAR command puts OPERA-3d back to the state it was in when it first started. It deletes all the data and user variables and sets all parameters back to their default values.

The **COLOUR** Command

Summarv	Adjust the colours	s used for the display.

Icon



Menu Route

View \downarrow

Change colours

Command Line Parameters

C				
Command	COLOUR	•		
Parameter	Default	Function		
OPTION	SET	Option:		
		LOAD	Load current values as defaults	
		SET	Set new values	
LABEL TEXT		Label of colour to be redefined.		
		BACKGROUND	The background colour.	
		CONDUCTORS	The colour of conductors.	
		MAPFIRST- CONTOUR	The colour of the minimum contour level (MAP com-mand).	
		MAPLAST- CONTOUR	The colour of the maximum contour level (MAP com-mand).	
		SURFACE- FIRST- CONTOUR	The colour of the minimum contour level (THREED and VIEW commands).	
		SURFACELAST- CONTOUR	The colour of the maximum contour level (THREED and VIEW commands).	
		TEXT	The colour of text and axes.	
		TRAJECTORIES	The colour of trajectories.	

5-36

Command	COLOUR (continued)		
Parameter	Default	Function	
LABEL (contd)	TEXT	VECTOR- INSIDE	The colour of the inside of vector cones.
		VECTOR- OUTSIDE	The colour of the outside of vector cones.
		material_name	The colour of a material.
RED	none	Amount of red for colour.	
GREEN	none	Amount of green for colour.	
BLUE	none	Amount of blue	for colour.
TRANSLUCENT	NO	NO	Switch off colour trans- lucency
		YES	Set colour property to be translucent

Notes The COLOUR command allows the user to adjust the colours used for parts of the display. With OPTION=SET, the colour identified by LABEL is redefined using the new values for RED, GREEN and BLUE given by numbers in the range 0 to 255 and a TRANSLUCENT flag.

- XXXXFIRSTCONTOUR and XXXXLASTCONTOUR: The program interpolates colours between the colours used for the extreme values.
- **CONDUCTORS** and materials: When a database is loaded the colours used by the Modeller or pre processor are also loaded (this applies to databases created by version 8.1 and above).
- Making a material colour translucent (TRANSLUCENT=YES) makes other materials inside it visible. Only one material (or all the contour colours) can be translucent at a time. All other materials will be opaque (TRANSLUCENT=NO). There are two sets of contour colours, one for the MAP command, and one for the THREED command (which are also used for particle tracks on the 3D display in the VIEW command). Only one set, or one material, can be made translucent at a time.

The **COMBINE** Command

Summary	Select both magnetic and electric fields for particle trajectory calculation			

Menu Route Trajectories Use magnetic and electric fields

Command Line Parameters

I

Command	COMBINE			
Parameter	Default	t Function		
BOTH	YES	Combine magnetic and electric fields for particle trajectories: YES or NO.		
SYMMETRY	1	Rotational symmetry of additional field around local Z axis.		
RXY	NO	Reflection of	of additional field in local XY plane.	
		NO	No reflection.	
		YES	Field parallel to XY.	
		INVERSE	Field normal to XY.	
RYZ	NO Reflection of additional field in local		of additional field in local YZ plane.	
		NO	No reflection.	
		YES	Field parallel to YZ.	
		INVERSE	Field normal to YZ.	
RZX	NO Reflection of additional field in local ZX plan		of additional field in local ZX plane.	
		NO	No reflection.	
		YES	Field parallel to ZX.	
		INVERSE	Field normal to ZX.	
MAGSCALE	1	Scaling factor for magnetic field.		
ELESCALE	1	Scaling factor for electric field.		

Notes

The COMBINE command switches combined field particle tracking on or off. Before it is used, a TOSCA or SCALA database containing electric and magnetic fields must be activated and loaded. Additional magnetic fields are used in the SCALA program for particle trajectory calculations. They can exist as a result of conductor data in the SCALA database file or can be added to the database using the TABLE command (page 5-101). The same TABLE command method can also be used to add electric or magnetic fields to TOSCA databases which already contains the fields of the other type.
The geometric symmetry of the electrostatic and magnetostatic models should be the same. However, the field symmetry will often be different. If rotational and reflection symmetry is necessary on the ACTIVATE command for the model already activated, then the appropriate additional field symmetry should be specified on the COMBINE command using parameters SYMMETRY, RXY, RYZ and RZX.

The electric and magnetic fields can be scaled using the factors MAGS-CALE and ELESCALE.

After switching the combined tracking option on, the **TRACK** command can be used in the usual way.

The **CONDUCTOR** Command

Summary Define, erase, modify, list, import and export conductors.

Icons	0	New solenoid
	0	New racetrack
	\$	New bedstead
	\diamond	New helical end
	0	New constant perimeter end
	\mathbf{N}	New straight bar
	ノ	New arc
	8	New 8-node brick
	20	New 20-node brick
	Ç	Toggle conductor picking
		Modify selected conductors
		Erase selected conductors
		List conductor data

Menu Route

Conductors

Command Line Parameters

Command	CONDUCTOR	{			
Parameter	Default	Function			
ACTION	ADD	Create list of conductors, operate on con- ductors in list, etc.			
		ADD	Add conductor(s) to list.		
		DEFAULT	Set default values to match selected conductors.		
		DEFINE	Define a new conduc- tor.		
		DEFRESET	Reset default values to match last conductor defined.		
		ERASE	Erase selected conduc- tors.		
		EXPORT	Export conductors to a data file.		
		IMPORT	Import conductors from a data file.		
		LIST	List conductor data.		
		MODIFY	Modify selected con- ductors.		
		PICK	Add or remove conduc tor from list.		
		REMOVE	Remove conductor(s) from list.		
		RESET	Empty list of conduc- tors.		
		STARTPICK	Switch on picking facility.		
		STOPPICK	Switch off picking facility.		
LABEL	ALL_CONDU CTORS	Conductors to be added or removed from selection list:			
		number	Conductor number.		
		drive_label	Drive label.		
		ALL_CONDUC TORS	All conductors.		

Command	CONDUCTOR	(continued)		
Parameter	Default	Function		
TYPE	SOLENOID	Conductor type: SOLENOID, RACE- TRACK, BEDSTEAD, HELICALEND, CPEND, STRAIGHT, ARC, BR8 or BR20.		
XCEN1	0	X coordinate of origin of local system 1.		
YCEN1	0	Y coordinate of origin of local system 1.		
ZCEN1	0	Z coordinate of origin of local system 1.		
THETA1	0	Euler angle θ (local system 1).		
PHI1	0	Euler angle ϕ (local system 1).		
PSI1	0	Euler angle ψ (local system 1).		
XCEN2	0	X coordinate of origin of local system 2.		
YCEN2	0	Y coordinate of origin of local system 2.		
ZCEN2	0	Z coordinate of origin of local system 2.		
THETA2	0	Euler angle θ (local system 2).		
PHI2	0	Euler angle ϕ (local system 2).		
PSI2	0	Euler angle ψ (local system 2).		
IRXY	0	Reflection code in xy plane of local system 1.		
IRYZ	0	Reflection code in yz plane of local system 1.		
IRZX	0	Reflection code in zx plane of local system 1.		
SYMMETRY	1	Symmetry code.		
CURD	100	Current density.		
TOLERANCE	1	Tolerance on flux density (negative for single filament approximation).		
PHASE	ONE	Drive label.		
X1	3	X coordinate of corner of conductor cross section.		
Y1	3	Y coordinate of corner of conductor cross section.		
X2	3	X coordinate of corner of solenoid cross section.		
Y2	4	Y coordinate of corner of solenoid cross section.		
ХЗ	4	X coordinate of corner of solenoid cross section.		
Y3	4	Y coordinate of corner of solenoid cross section.		
X4	4	X coordinate of corner of solenoid cross section.		

I

I

I

Command	CONDUCTOR	(continued)	
Parameter	Default	Function	
Y4	3	Y coordinate of corner of solenoid cross section.	
CU1	0	Curvature of cross section of solenoid (points 1 to 2).	
CU2	0	Curvature of cross section of solenoid (points 2 to 3).	
CU3	0	Curvature of cross section of solenoid (points 3 to 4).	
CU4	0	Curvature of cross section of solenoid (points 4 to1).	
A	1	Thickness of conductor in x or radial direction.	
В	1	Width of conductor in y or azimuthal direction.	
H1	5	Length of straight section.	
H2	1	Length of upright (BEDSTEAD)	
		Local Z coordinate of midpoint of cross- over (HELIX).	
R1	0.5	Radius: inner radius of arc (RACE- TRACK, BEDSTEAD, ARC). Radius of cylinder (HELIX_CPEND)	
R2	0.6	Radius: inner radius of arc (BED- STEAD). Width of cross-over (HELIX).	
		Radius of generating cylinder (CPEND).	
PHI	30	Angle of ARC.	
ALPHA	45	Angle of straight from mid plane of cylinder (HELIX and CPEND).	
BETA	60	Angle of end of helix (HELIX), or cutter (CPEND).	
FIT	TANGENTIAL	Fit of straight section to cylinder: TAN- GENTIAL or FITTING (CPEND).	
XBn	A bar, starting at the origin,	X coordinate of node of 8 or 20 node brick conductor $(1 \le n \le 20)$.	
YBn	with cross sec- tion 1x1 and	Y coordinate of node of 8 or 20 node brick conductor $(1 \le n \le 20)$.	
ZBn	length 10.	Z coordinate of node of 8 or 20 node brick conductor $(1 \le n \le 20)$.	
FILE	none	Name of file for IMPORT or EXPORT	

Notes This command controls the definition, modification, import and export of conductors. The operation of the command is controlled by the ACTION parameter:

- ACTION=DEFINE: defines a new conductor. The definition of the parameters and how they apply to each conductor shape is given in Chapter 4 (see "The CONDUCTOR Command" on page 4-24).
- ACTION=ERASE: erases a list of conductors. Before conductors can be erased the list must be formed using actions RESET, ADD, PICK and REMOVE (see below). After conductors have been erased the remaining conductors are renumbered to form a contiguous set starting at 1 and the selection list is emptied.
- ACTION=MODIFY: modifies a list of conductors. Before conductors can be modified the list must be formed using actions RESET, ADD PICK, and REMOVE (see below). The default values of the conductor parameters should also be set to correspond to the conductors selected using ACTION=DEFAULT. After modifying, the default values can be returned to correspond to the last conductor defined using ACTION=DEFRESET.

It is possible to set new values for parameters to expressions involving the old values, for example to double the current densities. The command sequence to do this to all conductors would be:

```
CONDUCTOR ACTION=ADD LABEL=ALL_CONDUCTORS
CONDUCTOR ACTION=DEFAULT
CONDUCTOR ACTION=MODIFY CURD=CURD*2
CONDUCTOR ACTION=DEFRESET
```

- ACTION=DEFAULT: sets the default values to correspond to the selected list of conductors in preparation for modification. If the conductor type is unique, the string variable CONDUCTOR_TYPE is given the appropriate value.
- ACTION=DEFRESET: resets the default values to correspond to the last conductor defined after using ACTION=MODIFY.
- ACTION=LIST: lists conductor data. If there is a selected list of conductors, only those conductors will be listed. Otherwise, all conductors will be listed.
- ACTION=IMPORT: reads conductor data from a conductor data file. Conductor data files are compatible with those created by the pre processor CONDUCTOR sub-command WRITE (page 4-48) and the Modeller command EXPORT (page 3-73). For hints on how to write compatible command scripts for all 3 programs see "Conditional commands" on page 2-20.
- ACTION=EXPORT: writes conductor data to a file. If there is a selected list of conductors, only those conductors will be included. Otherwise, all conductors will be included.

- ACTION=ADD: adds conductors to the selected list by label. Labels can be conductor numbers, drive labels or ALL_CONDUCTORS.
- ACTION=REMOVE: removes conductors from the selected list by label. Labels can be conductor numbers, drive labels or ALL_CONDUCTORS.
- ACTION=STARTPICK switches on conductor picking.
- ACTION=PICK adds conductor number given by LABEL to the list if it is not already in the list, or removes it if already there.

This command is automatically generated by pointing at the appropriate conductor and double-clicking the left mouse button.

- ACTION=STOPPICK switches off conductor picking.
- ACTION=RESET: empties the selected list of conductors.

The system variable **CONDUCTORS** holds the number of conductors.

The END Command

Summary	End the post processor
Menu Route	File \downarrow Exit
Command Line Parameters	Command END No Parameters

The END command stops the OPERA-3d post processor. All data files are closed.

Notes

The **ENERGY** Command

Summary	Calculate volume integrals to obtain stored energy, power loss and Lorentz
	forces.

Icon



Menu Route

Integrals ↓ Energy, power & force

Command Line Parameters

Command	ENERGY			
Parameter	Default	Function		
ACTION	INTEGRATE	Create list of volumes or integrate:		
		ADD	Add volume(s) to	
			list.	
		INTEGRATE	Integrate.	
		REMOVE	Remove volume(s)	
			from list.	
		RESET	Empty list of vol-	
			umes.	
LABEL	ALL_VOLUMES	volumes to be added or removed from		
		list:		
		material	Material name.	
		label	Volume label.	
		ALL_VOLUMES	All volumes.	
TAVERAGE	YES	Time-average switch.		
		NO	Calculate integrals	
			at time of SET	
			command.	
		YES	Calculate time-	
			average integrals.	

Command	ENERGY (continued)		
Parameter	Default	Function	
ADAPTIVE	NO	Adaptive integration switch. NO Use 8 gauss-poin in each element.	
		YES	Use up to 216 gauss-points in each element.
MULTIPOLE	NO	Calculate multipol NO.	e moments: YES or

Notes

I

The ENERGY command integrates the stored energy, Lorentz force on induced currents, power loss and volume in the whole problem space or in a labelled set of elements and updates the following system variables:

Integrals			
Variable	Integrand		
ENERGY	linear energy:		
	$\frac{1}{2}\mathbf{B} \cdot \mathbf{H}$ (magnetics)		
	$\frac{1}{2}\mathbf{D} \cdot \mathbf{E}$ (electrostatics)		
COENERGY	$\int (\mathbf{B} \cdot d\mathbf{H}) \text{ (non-linear magnetostatics)}$		
NLENERGY	non-linear energy:		
	2*ENERGY-COENERGY		
ELECENER	$\frac{1}{2}\mathbf{D} \cdot \mathbf{E}$ (high-frequency)		
FX	$J_y B_z - J_z B_y$ (eddy currents)		
FY	$J_z B_x - J_x B_z$ (eddy currents)		
FZ	$J_x B_y - J_y B_x$ (eddy currents)		
POWER	$ \mathbf{J} ^2/\sigma$ (current flow and eddy currents)		
HYSENERGY	Total time average stored energy for		
	hysteretic materials (complex μ)		
HYSPOWER	Additional time average power loss for		
	hysteretic materials (complex μ)		
VOLUME	1		

The command operates as a 2-stage process:

- 1. form a list of volumes. Initially all volumes are in the list.
 - ACTION=RESET empties the list.

- ACTION=ADD adds volumes to the list by LABEL.
- ACTION=REMOVE removes volumes from the list by LABEL.
- Labels can be material names, volume labels including element and potential types and user labels or ALL_VOLUMES.
- 2. integrate forces using ACTION=INTEGRATE.

The volume integral of other expressions of system variables can be calculated using the VOLUME command (page 5-121).

Integration Method

The basis for the integration is the finite element mesh. The integrals are performed in each element using first-order gaussian quadrature. However, in reduced potential volumes if the coil field is calculated by integration or anywhere if the total field is calculated by integration, the first-order quadrature is insufficient to match the field variation in an element. Switching on adaptive integration (+ADAPTIVE) enables the program to use up to 9^{th} -order gaussian quadrature in each element to increase the accuracy of the integrals. See "The SET Command" on page 5-88 for information about field calculation methods.

The ENERGY command includes rotated and reflected copies of the mesh specified by the ACTIVATE command (page 5-22) in its integration.

Steady-state AC problems

In steady-state alternating current problems, the energy and power are functions of time with the form

$$E = A + B\cos 2\omega t + C\sin 2\omega t \tag{5.3}$$

The values *B* and *C* have little meaning on their own. The time-average value, *A* is the value commonly required. This can be calculated directly using +TAVERAGE.

The values of *B* and *C* can be found by setting the times to 0, 45 and 90 (page 5-88), to give values of *E* at each time: E_0 , E_{45} and E_{90} with the **-TAVERAGE** option.

Version 10.0

$$A = \frac{E_0 + E_{90}}{2}$$

$$B = \frac{E_0 - E_{90}}{2}$$

$$C = E_{45} - A$$

(5.4)

The following commands can be used to achieve this:

```
Example - time-average energy:
set time=0
ener -tave
$ cons #en0 energy
set time=45
ener
$ cons #en45 energy
set time=90
ener
$ cons #en90 energy
$ para #ena 0.5*(#en0+#en90)
$ para #enb 0.5*(#en0-#en90)
$ para #enc #en45-#ena
```

The values E_0 , E_{45} and E_{90} have little meaning on their own.

Multipole Moments

MULTIPOLE moments can be calculated for all non-AIR materials in TOSCA magnetostatic simulations. The values are stored in a separate output file, with the name taken from the OPERA-3d database, but with extension *.mpole*. The volume, dipole, quadrupole and octupole moments are stored for all materials and for each material. Each moment value is given in two unit sets: amp metreⁿ and gamma ftⁿ (1 gamma = 10^{-9} Tesla).

The **FIT** Command

Summary	Fit Fourier series to field values previously calculated along a line or cal-
	culate field values on a sphere and fit Legendre polynomials.

Menu Routes	Fields \downarrow
	Fit Fourier series to values
	Fit Legendre polynomials to values

Command Line				
Parameters	Command	FIT		
	Parameter	Default	Function	
	FILE	TEMP	Name of the table file containing the field points and values. FILE=TEMP means do not use a file.	
	TYPE	FOURIER	R Type of fitting	
			FOURIER	FOURIER series fitting to ARC, CIRCLE or LINE values.
			LEGENDRE	LEGENDRE polynomial
				fitting to values on a sphere around local coordinate sys- tem origin
	COMPONENT	Х	Field component for fitting.	
	PRINT	YES	Print options.	
			LOG	Output to log file opera.lp.
			SCREEN	Output to screen.
			YES	Output to both SCREEN and LOG file.
			NO	No output.
	RADIUS	1	Radius of sphere for LEGENDRE polynomial fitting.	
	ORDER	10	Maximum degree of LEGENDRE polynomials.	
	COLUMN	4	Column of table file.	

Notes

The FIT command is used to fit FOURIER series or LEGENDRE polynomials to field values.

FOURIER series fitting uses field values previously calculated by the ARC, CIRCLE or LINE commands (page 5-24, page 5-33 and page 5-68). The values are assumed to span a complete cycle. The values can be those stored in the program (FILE=TEMP) or can be read in from a file by the FIT command. The structure of the files is given in section "TABLE Files" on page 5-19. If no file name extension is given the extension *table* is assumed.

LEGENDRE polynomial fitting first calculates values on the surface of a sphere which must be completely inside the problem space. It may be necessary to ACTIVATE the results with symmetry parameters in order to achieve this (page 5-22). The radius of the sphere is given by the RADIUS parameter, and its origin and orientation are defined by the local coordinate system of the SET command (page 5-88). The results are given for a spherical polar coordinate system whose axis is in the local Z direction. The maximum order of polynomial can be set with the ORDER parameter; it is limited to 30. However for 9≤ ORDER ≤14, coefficients up to order 14 will be given and for ORDER ≥14 coefficients up to order 30 will be given. The values at the field points used by the fitting algorithm can be displayed by the MAP commands (page 5-72).

If a file is not used (FILE=TEMP), expressions for the COMPONENT can use as variables any of the system variables which are currently available (see "System Variables" on page 5-10). If a table file is used, the component name and values are read from the chosen COLUMN of the file.

The **GRAPH** Command

Menu RouteFile↓Graph data in text file

Command Line Parameters

Command	GRAPH			
Parameter	Default	Function		
FILE	none	File name		
XVALUE	COL1	Expression for x-axis	values.	
XLABEL	Х	Label for x-axis.		
XMINIMUM	*	X-axis minimum valu scaling.	e. Use * for automatic	
XMAXIMUM	*	X-axis maximum valu scaling.	e. Use * for automatic	
YVALUE	COL2	Expression for y-axis	values.	
YLABEL	Y	Label for y-axis.		
YMINIMUM	*	Y-axis minimum value. Use * for automatic		
		scaling.		
YMAXIMUM	*	Y-axis maximum value. Use * for automatic		
		scaling.		
TITLE	none	Additional title for lin	e.	
STYLE	AUTOMATIC	Line style:		
		AUTOMATIC	Program chooses a	
			different style for	
			each graph drawn on	
			the same axes.	
		0	Solid line.	
		>0	Broken line.	

Command	GRAPH (continued)			
Parameter	Default	Function		
COLOUR	AUTOMATIC	Line colour:		
		AUTOMATIC	Program chooses a	
			different colour for	
			each graph drawn on	
			the same axes.	
		name	Colour name: IEXI,	
			RED, GREEN,	
			CVAN or	
			MAGENTA	
		>0	Colour number	
SVMPOL		V Symbols at data point		
STIVIDUL	AUTOWATIC	Symbols at data points	S:	
		AUTOMATIC	Program chooses a	
			each graph drawn on	
			the same axes	
		0	No symbol	
			NO Symbol.	
	VEO	>0	Symbols number.	
ERASE	TES	Old graph erasure swi	tch:	
		NO	New line drawn on	
			existing axes.	
		YES	Graphics window	
			cleared and new axes	
			drawn.	
OPTION	VALUES	Display option:	1	
		DERIVATIVES	Display the first	
			derivatives of the data	
		INTERPOLATIONS	Display cubic-spline	
			interpolations	
			between data values.	
		VALUES	Display straight lines	
			between data values.	

Notes

The **GRAPH** command is a general purpose command for displaying graphs of data read from external files.

The parameters of the GRAPH command control the axes limits (XMINI-MUM, XMAXIMUM, YMINIMUM and YMAXIMUM), the line STYLE and COLOUR and whether a SYMBOL should be displayed at the data points. For graphs with more than one line, the second and subsequent lines

5-54

should be drawn with –**ERASE**. A line **TITLE** can be specified for each line. The line titles appear in a legend at the bottom left corner of the graph.

The data format expected by the graphs command is flexible. The file should consist of up to 20 columns of numbers, with any number of values in each column. However, text can be embedded within the numerical data. Each line of the file is parsed into a maximum of 20 fields separated by spaces or commas. The fields are identified as character data or numerical data. If there is numerical data on a line it is kept; character data is ignored. The number of columns is given by the number of numerical data items on the first line which contains any numbers. If subsequent lines contain less numerical values, the number of columns is reduced.

The values for the x and y coordinates of the points plotted on the graphs can be calculated using expressions in terms of corresponding entries from the columns. The simplest use would be to use the first column for the xcoordinates and the second for the y-coordinates (XVAL=COL1, YVAL=COL2), but much more complicated expressions can be used. For example, the percentage difference between two columns could be calculated and displayed as a graph using YVAL=100*(COL3-COL2)/COL2. The ROW number can also be used in expressions.

The data evaluated from the expressions XVALUE and YVALUE can be displayed in 3 ways:

- OPTION=VALUE shows straight lines between the evaluated data points.
- **OPTION=INTERPOLATIONS** uses cubic-spline interpolations between the data points.
- **OPTION=DERIVATIVES** shows the first derivative of the cubic-spline interpolations.

INTERPOLATIONS and **DERIVATIVES** can only be used if the expression for **XVALUE** results in a mono-tonically increasing set of values.

The **GRID** Command

Summary	Calculate field values on a grid of points and write a table file.
Menu Route	Tables \downarrow Table of fields values on a grid

Command Line Parameters

Command	GRID	
Parameter	Default	Function
X0	0	X-coordinate at corner of grid.
Y0	0	Y-coordinate at corner of grid.
ZO	0	Z-coordinate at corner of grid.
DXG	1	X increment between grid points
DYG	1	Y increment between grid points
DZG	1	Z increment between grid points
NXG	1	Number of points in X direction
NYG	1	Number of points in Y direction
NZG	1	Number of points in Z direction
FILE	none	Name of file to store values.
BINARY	NO	Binary file switch: YES or NO.
		NO implies a text file.
F1	Х	Expression for values in column 1
F2	Y	Expression for values in column 2
F3	Z	Expression for values in column 3
F4,, F12	none	Expression for values in column 4
		to 12
NAME1,, NAME12	F1,, F12	Names for columns
UNIT1	LENGU	Unit expression for column 1
UNIT2	LENGU	Unit expression for column 2
UNIT3	LENGU	Unit expression for column 3
UNIT4,, UNIT12	1.0	Unit expressions for columns 4 to 12

Notes

The GRID command is provided to facilitate an interface to other post processing programs. It calculates field values over a 1, 2 or 3 dimensional space defined by one corner (X0, Y0, Z0), the increments in each direction

(DXG, DYG, DZG) and the number of points in each direction (NXG, NYG, NZG). The coordinates of the points are defined with respect to the local coordinate system of the SET command (page 5-88). The field points and components are output to a file in the Global Coordinate System.

Expressions for values in up to 12 columns can use any of the system variables which are currently available (see "System Variables" on page 5-10). In text files (-BINARY) table file format is used (see "TABLE Files" on page 5-19). The column names and unit expressions can be set using the NAMEn and UNITn parameters. Binary files (+BINARY) are written using grid file format (see "GRID files" on page 5-18.)

If no file name extension is given, the extension *table* is supplied for text files or *grid* for binary.

The **GUIOPTIONS** Command

Summary	Set console	visibility and	other v	vindow	options.
Summer y	000000000000000000000000000000000000000	, isionity and			opnonor

Menu Route

Window ↓ Preferences

Command Line Parameters

I

Command	GUIOPTIONS		
Parameter	Function		
OPTION	Action of command		
	LOAD	Used by the GUI.	
	SET	Update the values.	
CONSOLEVIEW	Console vi	sible: YES or NO	
CONSOLEDOCK	Console docked: YES or NO		
CONSOLEBUFFER	Number of lines saved in the console.		
EXTRASVIEW	View Units, Model Data and Local Coordi- nates: YES or NO		
PRINT3DCARD	Print from graphics card: YES or NO		
WINDOWPOSSAVE	Save window position and size on exit: YES or NO		
BACKINGSTORE	Graphics backing store: YES or NO		

The GUIOPTIONS command can be used to change the options which affect the appearance and behaviour of the GUI window. Default settings are loaded from and changes are saved in the registry (on UNIX systems, file ~/.vectorfields-options). Saving the window size and position is optional - see page 5-59.

- Console the command input and text output console can be visible or hidden (CONSOLEVIEW) and when it is visible it can be docked (at the bottom of the main window) or undocked (a separate window) (CONSOLEDOCK). The scrolling buffer can be limited to a number of lines (CONSOLEBUFFER).
- Information box, containing UNITS, PROBLEM DATA and Local Coordinates can be visible or hidden (EXTRASVIEW).
- Printing the option to print the image stored in the graphics card is preferable with most graphics card (PRINT3DCARD) but can be

changed if necessary. The alternative copies the image from the screen display.

- Window size and position the current size and position can be saved when the program exits (WINDOWPOSSAVE). To preserve the size and position from subsequent changes, WINDOWPOSSAVE must be set to NO, the next time the program is started.
- Backing store must be enabled to allow the program to re-display 2D pictures if the window size is changed. However, for very complex models, e.g. graphs of large numbers of trajectories, backing store can make the graphics run more slowly.

The IDEAS Command

Menu Route	Tables \downarrow			
	SDRC	I-DEAS	Unv	file

Command Line

Parameters

Command	IDEAS			
Parameter	Default	Function		
FILE	none	Name of Universal File.		
MODE	APPEND	Mode of operation:		
		APPEND	Append results to an existing file.	
		CREATE	Create a new file with nodes, elements, material names, boundary conditions and results.	
		DATAONLY	Create a new file with node, elements, material names and boundary conditions.	
TYPE	REAL	Result type:		
		COMPLEX	Results from steady-state ac analysis	
		REAL	Results from statics or tran- sient analysis.	
BASIS	NODE	Basis of results:		
		ELEMENT	Values at every node of every element	
		NODE	Values at every node	
FIELD	SCALAR	Field type:		
		SCALAR	Scalar field	
		VECTOR	Vector field	
COMPONENT	X	Expression for real part of scalar field values.		
ICOMPONENT	Y	Expression for imaginary part of scalar field values (TYPE=COMPLEX)		
VX	Х	Expression of real part of x-component of vector field.		

Command	IDEAS (continued)	
Parameter	Default	Function
IVX	X	Expression for imaginary part of x-compo- nent of vector field
VY	Y	Expression of real part of y-component of vector field.
IVY	Y	Expression for imaginary part of y-compo- nent of vector field
VZ	Z	Expression of real part of z-component of vector field.
IVZ	Z	Expression for imaginary part of z-compo- nent of vector field
LABEL	1	Analysis dataset label
NAME	VF	Analysis dataset name

Notes

The IDEAS command creates or appends to an I-DEAS Universal file. When the file is created (MODE=CREATE) the finite element data is written first, followed by a results dataset if available. In APPEND mode, only the results dataset is written. In DATAONLY mode a file is created with only data and no results datasets.

Finite Element Data

The finite element data written to a Universal File consists of the following datasets:

Dataset Number	Dataset Name
151	Header
164	Units
1700	Material Database Header
1703	Material Database Property
1705	Material Database Variable
1710	Material Database Material
789	Physical Property Table
2411	Nodes - Double Precision
2412	Elements
792	Boundary Conditions

Dataset 164 specifies the length unit for the data. The 'Units description' includes the units of flux density, current density and electric field strength.

Dataset 1710 is repeated for each 'material', where a material is formed for each combination of material name, potential type and element type which exists in the model, for example a typical Universal File Material Name would be IRON_TOTAL_QUADRATIC. The 'Material Number' and 'Material Name' are the only significant data in the dataset 1710.

Dataset 792 is repeated for each volume face of the original model which has a boundary condition.

Results Data

Results data are written using dataset 2414 'Analysis Data'. The simplest form is a real scalar field at nodes (**TYPE=REAL**, **BASIS=NODE**, **FIELD=SCALAR**) which can be specified using one COMPONENT expression. At the other extreme is a complex vector field on elements, allowing discontinuity between each element and its neighbours (**TYPE=COMPLEX**, **BASIS=ELEMENT**, **FIELD=VECTOR**). This would be specified using 6 expressions, for the real and imaginary parts of the X, Y and Z components.

All other combinations of TYPE, BASIS and FIELD are also allowed, for example, to store the electric field strength in an electrostatic example, not allowing for any discontinuities:

type=real,basi=node,fiel=vect,vx=ex,vy=ey,vz=ez

Limitations for Quadratic Elements

If the finite element model in the OPERA-3d database was created from a Universal File using the IDEAS command in the pre processor (page 4-117), and it also contains quadratic elements, the results should not be APPENDED to the original Universal File, since the positions and ordering of the quadratic nodes will not match. A new Universal File should be CREATED to contain the finite element model and results.

Integration Commands

In many situations, the result which is required is calculated by integrating field quantities along a line, over a surface or in a volume. OPERA-3d post processor has several commands which can perform integration, some for specific purposes and others which allow more general facilities.

Line Integrals

Line integrals are calculated by the PLOT command. The line should be defined and field quantities evaluated first by ARC, CIRCLE or LINE. The PLOT command (page 5-77) draws a graph of any field component expression and also calculates the line integral.

The **GRAPH** command (page 5-53) can also be used to calculate line integrals. It uses values read from a data file.

Surface Integrals

There are two types of surface integral:

- *Over a surface of the model:* The surface is the currently **SELECT**ed surface (page 5-84). It can be the surface of a material or potential type or a boundary condition surface.
 - The INTEGRATE command (page 5-65) integrates the Maxwell Stress tensor over the surface to calculate force and torque.
 - The SURFACE command (page 5-94) integrates any field component expression over the surface.
- Over a general surface patch: The surface should be defined and the field quantities evaluated first using the CARTESIAN or POLAR commands.
 - The MAP commands (page 5-72) display distribution of the field component expression and also calculate the area integral.

Volume Integrals

Volume integrals can be performed over volumes selected by labels (including material names, potential names, element types, NOTAIR and ALL_VOLUMES). The basis for the integration is the finite element mesh; higher order integration formulae are available for functions which vary rapidly over the elements.

- The ENERGY command (page 5-47) has built-in integrands to calculate energy, power-loss and Lorentz forces in the complete model including replications specified on the ACTIVATE command.
- The VOLUME command (page 5-121) can integrate any field component expression. Replications of the model are omitted from the integration.

The BODY command (page 5-28) integrates Lorentz forces over the volume of the conductors.

The INTEGRATE Command

Summary Integrate Maxwell stress over selected surfaces to obtain forces on enclosed volume.

Icon



Menu Route

Integrals \downarrow Maxwell stress on selected surfaces

Command Line Parameters

Command	INTEGRATE			
Parameter	Default	Function		
X0	0	X coordinate of point of action for torque.		
Y0	0	Y coordinate of point of action for torque.		
ZO	0	Z coordinate of point of action for torque.		
TAVERAGE	YES	Time-average switch.		
		NO	Calculate integrals at time of SET com-	
			mand.	
		YES	Calculate time-average integrals.	

Notes The INTEGRATE command integrates the Maxwell Stress tensor over the surface selected by the SELECT command (page 5-84) to calculate the force and torque. The parameters (X0, Y0, Z0) define the global coordinates of the point of action of the torque. The results are also stored in the system variables FX, FY and FZ and TORQX, TORQY and TORQZ.

For magnetic field simulations functions integrated are:

$$F_{x} = \int_{s} \left[\frac{1}{\mu} B_{x} (\mathbf{B} \cdot \mathbf{n}) - \frac{1}{2\mu} |\mathbf{B}|^{2} n_{x} \right] ds$$

$$F_{y} = \int_{s} \left[\frac{1}{\mu} B_{y} (\mathbf{B} \cdot \mathbf{n}) - \frac{1}{2\mu} |\mathbf{B}|^{2} n_{y} \right] ds$$

$$F_{z} = \int_{s} \left[\frac{1}{\mu} B_{z} (\mathbf{B} \cdot \mathbf{n}) - \frac{1}{2\mu} |\mathbf{B}|^{2} n_{z} \right] ds$$
(5.5)

For electrostatic field simulations functions integrated are:

$$F_{x} = \int_{s} \left[\frac{1}{\mu} D_{x} (\mathbf{D} \cdot \mathbf{n}) - \frac{1}{2\mu} |\mathbf{D}|^{2} n_{x} \right] ds$$

$$F_{y} = \int_{s} \left[\frac{1}{\mu} D_{y} (\mathbf{D} \cdot \mathbf{n}) - \frac{1}{2\mu} |\mathbf{D}|^{2} n_{y} \right] ds$$

$$F_{z} = \int_{s} \left[\frac{1}{\mu} D_{z} (\mathbf{D} \cdot \mathbf{n}) - \frac{1}{2\mu} |\mathbf{D}|^{2} n_{z} \right] ds$$
(5.6)

Better results can often be obtained if a layer of air is added to the selected surfaces before INTEGRATE to move the integration surface away from the material corners where the field may be singular (See "The SELECT Command" on page 5-84.). In any case, on a material surface, the INTE-GRATE command uses the values from the air side of the interface.

If the force required is acting on the coils, it is usually more accurate to select the surface of the reduced scalar potential volume containing the coils and INTEGRATE over that or use the BODY command (page 5-28),

The MAP command can also be used to integrate over more general surfaces. The functions necessary for calculating forces are given in terms of \$ PARAMETER commands in section "Examples" on page 5-16. The SURFACE command can be used to integrate other field quantities over the selected surface (page 5-94).

In steady-state alternating current problems, the force is a function of time with the form

$$F_{x} = A + B\cos 2\omega t + C\sin 2\omega t \tag{5.7}$$

The values *B* and *C* have little meaning on their own. The time-average value, *A* is the value commonly required. This can be calculated directly using **+TAVERAGE**.

The values of B and C can be found by setting the times to 0, 45 and 90

(page 5-88), to give values of F_x at each time: F_0 , F_{45} and F_{90} with the -TAVERAGE option.

$$A = \frac{F_0 + F_{90}}{2}$$

$$B = \frac{F_0 - F_{90}}{2}$$

$$C = F_{45} - A$$
(5.8)

The following commands can be used to achieve this:

```
Example - time-average F<sub>x</sub>:
set time=0
inte -tave
$ cons #fx0 fx
set time=45
inte
$ cons #fx45 fx
set time=90
inte
$ cons #fx90 fx
$ para #fxa 0.5*(#fx0+#fx90)
$ para #fxb 0.5*(#fx0-#fx90)
```

The values F_0 , F_{45} and F_{90} have little meaning on their own.

The LINE Command

Summary	Calculate fields along a straight lir	ıe.
---------	---------------------------------------	-----

Icon



Menu RouteFields ↓Fields on a straight line

Command Line Parameters

Command	LINE	
Parameter	Default	Function
X1	0	X-coordinate of the first point on the line.
Y1	0	Y-coordinate of the first point on the line.
Z1	0	Z-coordinate of the first point on the line.
X2	none	X-coordinate of the last point on the line.
Y2	none	Y-coordinate of the last point on the line.
Z2	none	Z-coordinate of the last point on the line.
NP	10	Number of steps between the first and last points, i.e $NP+1$ points.

Notes

The LINE command evaluates field quantities along a straight line (Figure 5.1) for use by the FIT and PLOT commands (page 5-51 and page 5-77). For each field point all the currently available system variables are calculated and stored (see "System Variables" on page 5-10).

The line is specified by its end points (X1,Y1,Z1 and X2,Y2,Z2), the positions of which are affected by any local coordinate system defined with the SET command (page 5-88). The vector field quantities are evaluated with respect to the Global Coordinate System for the active file. The field quantities are evaluated at NP+1 points along the line.

It is possible to store the evaluated field quantities in a file using the TABLE command (page 5-101).



Figure 5.1 The ARC, CIRCLE and LINE

The LOAD Command

Summary	Nominate an active database for display of geometry and field calculations.			
Icon	Open (activate+load)			
Menu Route	File ↓ Open (activate+load) Re-load active database Close loaded database Deactivate loaded database			

Commana Line Parameters			
	Command	LOAD	
	Parameter	Default	Function
	FILE	none	Name or number of an active database file.
	FILE	none	Name or number of an active database file.

Notes

. . .

The LOAD command makes an active database file resident. Files are activated by the ACTIVATE command (page 5-22). The LOAD command makes the data in an active file available for the display and field evaluation commands. The details of the loaded or resident file are shown in the Information Box at the right hand side of the display.

The parameter FILE sets the file name. If no file name extension is given an extension of *op3* is assumed. FILE can also specify the number of the active file. This can be useful if the same file has been activated more than once with different parameters on the ACTIVATE command. The active files can be listed with the SHOW command (page 5-92). If no file name or number is given the most recently activated file is assumed.

For some types of simulation, LOAD defines system variables:

- ANGLE: the rotor angle of a CARMEN simulation.
- FREQ: the frequency of an ELEKTRA-SS or SOPRANO simulation; the rotational frequency (rpm) of a CARMEN simulation.
- **TTIME**: the time of a CARMEN or ELEKTRA-TR simulation.

There are two special forms of the LOAD command:

- Close loaded database (**LOAD FILE=0**) can be used to close the currently loaded file so that it can be accessed by other programs. If the simulation was unsolved, the file is also removed from the list of active files.
- Deactivate loaded database (LOAD FILE=-1) can be used to close the currently loaded file and remove it from the list of active files.

The menu system automatically executes the LOAD command when a file is opened with the Open (activate+load) option.

The MAP Command

Summary Display contours of field vectors on 2-dimensional surface patches

Icon



Menu Route	Fields \downarrow		
	Contour or	vector	map

Command Line Parameters

Command	MAP			
Parameter	Default	Function		
FILE	TEMP	Name of table file containing field points and values. FILE=TEMP means do not use a file.		
CONTOUR	ZONES	Display COMPONENT as a contour map.		
		HISTOGRAM Contours displayed as a histo		
			gram.	
		LINES	Contour lines.	
		NO	No contour map.	
		ZONES	Coloured zone contours	
COMPONENT	Х	Field component for contours.		
MIN	*	Minimum contour value. * for automatic set-		
		ting.		
MAX	*	Maximum contour value. * for automatic set-		
		ting.		
VECTORS	NO	Vector display	switch.	
		NO	Vectors not displayed.	
		YES	Vectors displayed as cones with components specified by VX, VY and VZ.	
VX	Х	Expression for x-component of vectors.		
VY	Y	Expression for y-component of vectors.		
VZ	Z	Expression for z-component of vectors.		
PRINT	NO	Print options.		
		YES	Output to log file.	
		NO	No output.	

Command	MAP (continued)			
Parameter	Default	Function		
HEIGHT	SIZE/3	Height of histogram.		
ERASE	YES	Multiple map options:		
		YES	Replace any existing map.	
		NO	Add to existing maps.	
LINES	10	Number of line contours.		
SCALE	1	Scaling factor for vectors.		
COLSCALAR	4	Table file column for scalar value.		
COLVECTOR	5	First table file column for vector values.		

Notes

The MAP command is used to display field values calculated with the CARTESIAN, FIT TYPE=LEGENDRE and POLAR commands (page 5-31, page 5-51 and page 5-81) as a contour map or as vectors. These can be values stored in the program (FILE=TEMP) or read in from a table file by the MAP command. The structure of the files is given in section "TABLE Files" on page 5-19. If no file name extension is given the extension *table* is assumed.

The map is overlaid on the three dimensional pictures of the geometry created by the THREED command (page 5-104). Any number of contour maps can be added to the 3D Viewer (ERASE=NO) or a map can replace any previously displayed (ERASE=YES). The maps and the scale of values can be temporarily removed from the display using the WINDOW command (page 5-124).

Two types of map are available:

 CONTOUR of a scalar COMPONENT. The contour map can appear at the location of the field points (CONTOUR=ZONES or CON-TOUR=LINES) or displaced in proportion to the component value as a 3D histogram (CONTOUR=HISTOGRAM).

LINES sets the number of contour lines.

HEIGHT sets the height of the histogram. The default value gives a height of one third of the **THREED** command **SIZE** parameter.

• VECTORS of a field vector with components specified by VX, VY and VZ.

CONTOURS and **VECTORS** can be displayed with one command.

If a file is not used (FILE=TEMP), expressions for the scalar COMPO-NENT and the components of the vectors, VX, VY, VZ can use as variables any of the system variables which are currently available (section "System Variables" on page 5-10). If a table file is used, the component name and values are read from the column given by COLSCALAR; the vector components are read from the columns given by COLVECTOR and the two subsequent columns.

The values of the maximum and minimum contour values can be set with MAX and MIN. If either MAX or MIN is set to * the value is calculated from the range of values to be displayed.

The lengths of the vectors can be scaled by using a scaling factor SCALE which will multiply the values of VX, VY and VZ. Automatic scaling can be achieved by using the system variable MAXIMUM from a previous MAP command where the COMPONENT contoured was the magnitude of the vectors to be scaled.

The MAP command also calculates the integral of the COMPONENT, and stores its value in the system variable INTEGRAL. The maximum and minimum values of COMPONENT are stored in system variables MAXI-MUM and MINIMUM.

The MAP command can optionally **PRINT** the data is uses to the file *Opera3d_Post_nn.lp*:

- the field point coordinates
- the values of the scalar **COMPONENT**
- the values of the vector components, VX, VY and VZ

If LINE contours have been selected the start and end coordinates of all the line segments which make up the contours are also printed.
The **MOUSE** Command

	Summary	Swap the	functionality	of middle a	and right mouse	buttons.
--	---------	----------	---------------	-------------	-----------------	----------

Icon

I

I

S.

Menu Route	Options \downarrow	
	Toggle right mouse button	

Command Line							
r arameters	Command	MOUS	MOUSE				
	Parameter	Default	Funct	tion			
	NORMAL	YES	Set st	ate of right mouse button:			
			YES	Normal behaviour.			
			NO	Emulates the middle button.			

Notes The **MOUSE** command with **normal=no** swaps the functionality of the middle and right mouse buttons so that the software can be used with a 2-button mouse.

The default (**normal=yes**) mouse functions in the 3D Graphics Window are:

- left: rotate
- middle: zoom
- right: translate

The **PICTURE** Command

C	n 1	1. 1 .	C'1	.1 1 1 1
Summary	Save the current	display in a	file or on	the clipboard.

Icons



Menu Route	File \downarrow
	Copy to file
	Copy to clipboard

Command Line Parameters

I

Notes

Command	PICTUF	RE	
Parameter	Default	Funct	ion
SAVE	NO	NO	Copy file to clipboard
		YES	Save picture in a file
FILENAME	none	The n	ame of the file to be saved
TYPE	PNG	PNG	Type of image file format used to save the
		BMP	picture
		XPM	

This command allows the current display to be stored on the clipboard or in a file.

If SAVE=NO the image is placed on the clipboard and can then be pasted into another application. Under UNIX, the image is placed on the X-Selection.

With SAVE=YES, the image is stored in the format specified in the given file.

An image can be printed directly using the **PRINT** command.

The **PLOT** Command

Summary	Plot a graph and integrate field values from LINE, ARC or CIRCLE com- mands.
Icon	
Menu Route	Fields \downarrow Plot graph of field values

Command Line Parameters

I

Command	PLOT			
Parameter	Default	Function		
FILE	TEMP	Name of table f	ile containing field points	
		and values. FIL	E=TEMP means do not	
		use a file.		
COMPONENT	X	Field compone	nt for graph.	
YMINIMUM	*	Minimum value	e for Y-axis of graph. * for	
		automatic settin	ng.	
YMAXIMUM	*	Maximum valu	e for Y-axis of graph. *	
		for automatic s	etting.	
ERASE	YES	Erase screen sw	vitch:	
		NO	Draw new graph on	
			existing axes.	
		YES	Erase old picture first	
			and draw new axes.	
PRINT	LOG	Print options:		
		LOG	Output values to log file,	
			<i>Opera3d_Post_</i> nn. <i>lp</i> .	
		NO	No output of values.	
		SCREEN	Output values to screen.	
		YES	Output values to screen	
			and log file.	
GRAPH	YES	Graph drawing	switch:	
		NO	Do not draw graph.	
		YES	Draw graph.	

Command	PLOT (contin	ued)			
Parameter	Default	Function			
TITLE	none	Additional title for graph.			
LOCAL	YES	Local coordina	te switch:		
		NO	Report coordinates with		
			respect to Global Coordi-		
			nate System.		
		YES	Report coordinates with		
			respect to local coordi-		
			nate system.		
ORDINATE	NUMBER	Ordinate of gra	iph:		
		DISTANCE	DISTANCE along line.		
		NUMBER	Point NUMBER.		
OFFSET	0	Offset in distan	ice for overlaying graphs.		
STYLE	AUTOMATIC	TIC Line style:			
		AUTOMATIC	Program chooses a dif-		
			ferent style for each		
			graph drawn on the same		
		-	axes.		
		0	Solid line.		
		>0	Broken line.		
COLOUR	AUTOMATIC	Line colour:			
		AUTOMATIC	Program chooses a dif-		
			ferent colour for each		
			graph drawn on the same		
			axes.		
		name	Colour name: IEXI,		
			RED, GREEN, BLUE,		
			TELLOW, CTAIN OF		
			Colour number		
	4	ZU Table file cal	Colour number.		
COLUIVIN	4	1 able file colui	mn for values.		

Notes

The PLOT command is used to plot graphs and list values of field components previously calculated by the ARC, CIRCLE or LINE commands (page 5-24, page 5-33 and page 5-68). These can be values calculated by the last ARC, CIRCLE or LINE command (FILE=TEMP) or read in from a table file by the PLOT command. The structure of the files is given in section "TABLE Files" on page 5-19. If no file name extension is given the extension *table* is assumed. The coordinates of the field points can be displayed with respect to the local coordinate system of the **SET** command (**PLOT +LOCAL**) or with respect to the Global Coordinate System (**PLOT -LOCAL**).

The parameters allow choices of where the numeric values should be displayed (PRINT options), whether or not a graph should be plotted or not (+GRAPH or -GRAPH), whether or not a graph should overlay a previously plotted graph (+ERASE or -ERASE) and what the Y axis scale limits should be (YMIN and YMAX).

A TITLE, in addition to the value of the integral and the COMPONENT, can be added to the key at the bottom of the graph.

When multiple graphs are drawn on the same axes, each line is drawn in a different style or colour. The choice of **STYLE** and **COLOUR** can be **AUTOMATIC** or selected by the user.

If more than one graph is drawn on the same set of axes (-ERASE), the horizontal coordinates or ORDINATES of the points used can correspond to the point NUMBERS or to the DISTANCE along the line. If ORDI-NATE=DISTANCE an OFFSET can be added to the distance along the line.

If a file is not used (FILE=TEMP), expressions for the COMPONENT can use as variables any of the system variables which are currently available (see "System Variables" on page 5-10). If a table file is used, the component name and values are read from the chosen COLUMN of the file.

The PLOT command also calculates the integral of the COMPONENT, and stores its value in the system variable INTEGRAL. The maximum and minimum values of COMPONENT are stored in system variables MAXI-MUM and MINIMUM.

The **POINT** Command

Summary	Calculate fields values at a point		
Icon	•		
Menu Route	Fields \downarrow Fields at a point		

Commana Line Parameters	Command	POINT	
	Parameter	Default	Function
	XP	0	X-coordinate of the point.
	YP	0	Y-coordinate of the point.
	ZP	0	Z-coordinate of the point.
	COMPONENT	Х	Field component to be printed.

Notes

The POINT command evaluates field values at a point. The point is specified by parameters XP, YP and ZP which are with respect to the local coordinate system of the SET command (page 5-88).

All the currently available field quantities described in section "System Variables" on page 5-10 are calculated and their values can be used in subsequent \$ CONSTANT or \$ PARAMETER commands. The POINT command prints out the coordinates and the value of the COMPONENT. Expressions for the COMPONENT can use any of the system variables and any user variable.

The **POLAR** Command

Summarv	Calculate fields over a	patch specified i	n rθz coordinates.

Icon



Menu Route

Fields \downarrow Fields on a polar patch

Command Line Parameters

Command	POLAR	
Parameter	Default	Function
R1	none	R-coordinate of the first corner of the surface.
T1	none	θ -coordinate of the first corner of the surface.
Z1	none	Z-coordinate of the first corner of the surface.
R2	none	R-coordinate of the second corner of the surface.
T2	none	θ -coordinate of the second corner of the surface.
Z2	none	Z-coordinate of the second corner of the surface.
R3	none	R-coordinate of the third corner of the surface.
Т3	none	θ -coordinate of the third corner of the surface.
Z3	none	Z-coordinate of the third corner of the surface.
R4	none	R-coordinate of the fourth corner of the surface.
T4	none	θ -coordinate of the fourth corner of the surface.
Z4	none	Z-coordinate of the fourth corner of the surface.
N1	10	Number of points on sides 1 and 3.
N2	10	Number of points on sides 2 and 4.

Notes

The POLAR command evaluates field quantities on 4-noded surface patches in cylindrical polar coordinates. The results can be displayed by the MAP commands. For each field point all the currently available system variables are calculated and stored (see section "System Variables" on page 5-10).

The patch is specified by its corner points (R1,T1,Z1, R2,T2,Z2, R3,T3,Z3 and R4,T4,Z4) in cylindrical polar coordinates. The θ =0 plane is the ZX plane.

The GUI dialog box issues a SET command to define a local coordinate system for the field calculations, followed by the POLAR command. Following the POLAR command this local coordinate system will remain in effect for further field calculations until reset using the SET command.

The field vectors are evaluated in the Global Coordinate System.

The field quantities are evaluated at N1*N2 points. The coordinates of points other than the corners are found by linear interpolation in R, θ and Z using a 4-noded isoparametric two-dimensional finite element shape function. The surface therefore can be a circular cylinder, a plane, a cone or even a spiral. To achieve a circular patch, or a sector of a circle it is necessary to put two of the 4 defining points at zero radius, each with the same θ coordinate as one of the other two points.

It is possible to store the evaluated field quantities in a file using the TABLE command (page 5-101).

The **PRINT** Command

Summary	Print the current disp	lay.
Icon	a	
Menu Route	File \downarrow Print	
Command Line Parameters	Command No parameters	PRINT
Notes	This command allows the current display to be printed. A dialog of the available printers and their options for printing is opened. The current display can then be printed on the selected printer.	
	The printers available	e are determined from the system.

An image can be saved to file or copied to the clipboard using the PIC-TURE command ("The PICTURE Command" on page 5-76).

The **SELECT** Command

Summary Select surface facets or elements to be displayed.

Icons



Default select and refresh: SELECT ACTION=DEFAULT SELECT ACTION=SELECT OPTION=SURFACES THREED OPTION=REFRESH

Repeat select and refresh: SELECT ACTION=SELECT THREED OPTION=GETVIEW THREED OPTION=SETVIEW

Menu Route

View ↓ Select

Command Line Parameters

Command	SELECT		
Parameter	Default	Function	
ACTION	ADD	Create list of labels	or select:
		ADD	Add label to list of
			labels to be selected.
		DEFAULT	Create a list of labels suitable for type of analysis loaded.
		HIDE	Add label to list of labels to be hidden.
		REMOVE	Remove label from list.
		RESET	Empty the list of labels.
		SELECT	Make selection.

Command	SELECT (cd	ontinued)		
Parameter	Default	Function		
LABEL	NOTAIR	Label to be added or removed from list.		
		material_name	Material names including AIR or NOTAIR	
		potential_type	Potential type: REDUCED, TOTAL or VEC- TOR.	
		element_type	Element type: LIN- EAR or QUAD- RATIC.	
		bc_name	Boundary condition name.	
		user_label	Labels added by the Modeller or pre processor including	
		drive_label	Drive label: all con- ductors with this drive.	
		conductor_number	er Conductor number.	
OPTION	SURFACES	Type of selection ACTION=SELE	with CT.	
		ADD	Select surfaces then add a number of LAYERS of ele- ments.	
		SURFACES	Select surfaces.	
		ELEMENTS	Select elements.	
COIL	NO	Conductor selection switch.		
		NO	Conductors selected for geometry only.	
		YES	Conductors selected	
CUT	NO	Cut plana switch	for field display.	
		NO	No CLIT plana	
		YES	Select surfaces in the CUT plane.	
		FRONT	Select surfaces in front of CUT plane.	
		BACK	Select surfaces behind CUT plane.	

Command	SELECT (co	ntinued)	
Parameter	Default	Function	
THETA	0	θ Euler angle of Cl	JT plane.
PHI	0	φ Euler angle of Cl	JT plane.
ZCUT	0	Z-coordinate of CUT plane in local coor- dinate system defined by THETA and PHI.	
ZTOLERANCE	1.0E-5	Tolerance on ZCUT for CUT=YES .	
ACCURACY	0	Maximum facet size on conductors.	
		0	Use element sizes.
		>0	Subdivide facets so that largest sub-divi- sion of a facet is ≤ ACCURACY
LAYERS	1	Number of layers added (or removed if LAYERS is negative) with OPTION=ADD.	

The SELECT command creates a 'display buffer' containing a selection of the surfaces of the finite element mesh and conductors which are to be drawn with the THREED command (page 5-104). This is a two stage process:

- 1. create a list of labels: labels can be
 - added to the list (ACTION=ADD LABEL=name)
 - removed from the list (ACTION=REMOVE LABEL=name)
 - all removed from the list (**ACTION=RESET**)
 - added or removed in groups with additional names:

Additional labels	Meaning
ALL_BOUNDARIES	all surfaces with boundary condition
ALL_CONDUCTORS	all conductors
ALL_ELEMENTS	all element types
ALL_MATERIALS	all material names
ALL_POTENTIALS	all potential types
ALL_SURFACES	all surface labels
ALL_USERSURFACES	all user surface labels
ALL_USERVOLUMES	all user volume labels
ALL_VOLUMES	all volume labels
NOTAIR	all material names except air

Notes

- marked as hidden (ACTION=HIDE LABEL=name). Hidden take precedence over *selected* and can be used to hide parts of the model already selected. For example, to see the QUADRATIC AIR elements, add AIR and hide LINEAR.
- chosen to match the current simulation type (ACTION=DEFAULT).

For conductors, the label name can be either the conductor number or the drive label.

- 2. operate on the list of labels:
 - **OPTION=SURFACES** selects three types of element facets:

facets which lie in Modeller surfaces or pre processor facets which have labels given by the LABEL parameter.

facets which lie on the surface of Modeller cells or pre processor volumes which have the labels given by the LABEL parameter.

LABEL=ALL indicates the element facets on the exterior surface of the mesh.

- OPTION=ELEMENTS selects surfaces of all the elements which have volume labels given by the LABEL parameter. LABEL=ALL indicates all elements.
- OPTION=ADD selects surfaces and then moves the selected surface outwards (if number of layers is greater than zero) or inwards (if number of layers is negative) by a number of LAYERS.

The part of the finite element mesh selected can also be restricted by a cut plane. The CUT parameter can be set to FRONT or BACK to select surfaces of elements in front or behind the plane or to YES to select surfaces which lie in the plane. The tolerance on the position of the plane (ZTOL-ERANCE) should be set greater than zero to ensure that element surfaces are found correctly for CUT=YES and facets which touch the cut plane from the "wrong side" are eliminated for CUT=FRONT and CUT=BACK. The cut plane is defined as local XY plane of a coordinate system given by Euler angles THETA and PHI and the normal distance to the global origin ZCUT.

The ACCURACY parameter can be used to subdivide conductor surface facets. If ACCURACY>0, the facets are subdivided into smaller facets so that no facet linear dimension is larger than ACCURACY. If ACCU-RACY=0, no subdivision takes place.

The **SET** Command

Summary	Set options which affect the field calculations: method, point locations, time, etc.
Icons:	97 🗯 🕒 💿 0
Menu Route	Options ↓ Field calculation method Field points coordinate system

AC time

Include Jc in coils

Command Line Parameters

Command	SET			
Parameter	Default	Function		
FIELDS	NODAL	Field calculation	method:	
		BOTH	Interpolation of NODAL	
			values with magnetic fields	
			and flux densities calcu-	
			lated by INTEGRATION	
		NODAL	Interpolation of NODAL	
			values.	
		INTEGRATION	INTEGRATION of equiv-	
			alent magnetisation and	
			current sources to calculate	
			magnetic fields and flux	
			densities.	
COIL	NODAL	Field calculation	method for conductors in	
		REDUCED potential elements (ignored if		
		FIELD=INTEGRATION).		
		NODAL	Interpolation of NODAL	
			values.	
		INTEGRATION	INTEGRATION of current	
			density.	
TIME	0	Time for steady-state ac results: angle in		
		degrees around ac	e cycle.	

Command	SET (continue	ed)		
Parameter	Default	Function		
SEARCH	RESIDENT	SEARCH for field points:		
		RESIDENT	In RESIDENT (loaded)	
			file only.	
		ALL	In ALL ACTIVE files.	
XLOCAL	0	X-coordinate of th	he origin of the input local	
		coordinate system	1.	
YLOCAL	0	Y-coordinate of the	he origin of the input local	
		coordinate system	1.	
ZLOCAL	0	Z-coordinate of th	ne origin of the input local	
		coordinate system	1.	
TLOCAL	0	θ Euler angle of in	nput local coordinate sys-	
		tem.		
PLOCAL	0		nput local coordinate sys-	
		tem.		
SLOCAL	0	ψ Euler angle of i	nput local coordinate sys-	
		tem.		
ABORT	YES	Abort field calcul	ations if a field point is	
		found to be outside the finite element		
JCOIL	NO	Calculate source current density at field		
		points in coils: YES or NO		
LOOK	ANYWHERE	Choose which material to use for field points		
		on a material inter		
		AIR	Alf	
		ANYWHERE	First material found which	
		NOTUD	contains the field point	
		NUTAIR	Any material other than air	
		SELECTION	Volumes SELECTed for	
			dısplay	

Notes

The **SET** command defines parameters which affect the way the field calculation commands operate.

Field Calculation Methods

The FIELDS and COIL parameters set the method of field calculation used when processing results. The method also depends on the potential type at the field point.

Nodal Fields FIELDS=NODA	 total scalar potential or vector potential regions: The total field is cal- culated by interpolation of the NODALly averaged values.
L	• reduced scalar potential regions: the field is a combination of the field from the conductors and the field from the finite element mesh. The COIL parameter can be set to:
	 NODAL to request interpolation of nodal values supplied by the analysis programs.
	 INTEGRATION to request evaluation of the conductor fields by direct integration of the defined currents.
Integral Fields FIELDS=INTE GRATION	• The magnetic field strength, H , and flux density, B , are calculated from magnetisation and current density sources (TOSCA (magnetostatics and current flow) and ELEKTRA simulations only). The field points can be anywhere inside or outside the model. Other field values will be set to zero.

Integral fields• All possible fields are first calculated using nodal interpolation. The
magnetic field strength and flux density values are then replaced with
values calculated by integration.

The FIELDS and COIL parameters have no effect when recovering fields from conductor-only problems. In this case the magnetic field strength, **H** and flux density, **B**, are calculated by integration.

Field Point Searching

It is possible to have several active database files with the OPERA-3d post processor. The SEARCH parameter is used to choose the RESIDENT file only or ALL active files. If ALL is selected then for each field point each file will be loaded in turn until the field point is found.

Some fields are multi-valued on the interface between different materials and potentials. In order to force the program to use the values on one particular side of an interface, the parameter LOOK can be set to

- AIR: look only in air.
- NOTAIR: look everywhere except in air.
- SELECTION: look only in the volumes SELECTed for display (page 5-84).
- **ANYWHERE**: use the first occurrence of the field point found irrespective of material or potential.

I

This options does not affect the THREED and SURFACE commands (the elements used are set by the SELECT command) or the INTEGRATE command (the air-side of an interface is used if possible).

Field Points outside the Finite Element Mesh

When fields are calculated at many points in one command (ARC, CIR-CLE, LINE, CARTESIAN, POLAR and TABLE commands, etc.), the calculations end if a point is outside the finite element mesh space and the values for all subsequent points are set to zero (SET ABORT=YES). With SET ABORT=NO, the field calculations continue for all the requested field points.

Local Coordinate System

The field points given with the ARC, CARTESIAN, CIRCLE, FIT, GRID, LINE, POINT and POLAR commands are all defined with respect to a local coordinate system for field calculations. This coordinate system is set by its origin (XLOCAL, YLOCAL, ZLOCAL) and Euler angles (TLOCAL, PLOCAL, SLOCAL). Details of the definition of Euler angles are given in section "Euler Angles" on page 2-34. The local coordinate system, if it is different from the Global Coordinate System, is shown on the display in orange.

Steady-state AC Results

Steady state ac results are stored in the post processor as complex numbers. The real and imaginary parts are combined by the program to provide the field values at an angle around the ac cycle (**TIME**=*angle*) (see also section "System Variables" on page 5-10).

Current Density in Coils

For field points inside coils the conductors, the current density vector can optionally be calculated and stored in system variables JCX, JCY and JCZ. This can be selected using **SET** +JCOIL.

The **SHOW** Command

Summary	List details of the active simulations including the local coordinate systems and replications.
Menu Route	File \downarrow

List	active	databases
------	--------	-----------

Command Line
Parameters

Command	SHOW			
Parameter	Default	Function		
TITLE	NO	Display us	er titles: YES or NO.	
OPTION	ACTIVE	Listing option:		
		ACTIVE	All ACTIVE files.	
		FULL	Full details of resident simulation.	
		LOADED	All simulations in resident file.	

Notes

The ACTIVATE and LOAD commands (page 5-22 and page 5-70) allow many simulations to be active, but only one to be loaded. The same simulation can be activated several times with different parameters. Each database can contain several simulations. To keep track of these files and simulations, the SHOW command displays information about their contents.

There are 3 options:

- OPTION=ACTIVE: this gives a summary about all the active simulations. More information is given about the currently loaded simulation.
- OPTION=FULL: this gives complete information about the currently loaded simulation including all the material properties and analysis specific data.
- OPTION=LOADED: this lists all the simulations in the file containing the currently loaded simulation with more information about the currently loaded simulation.

The information includes:

Database Information	Value of OPTION		
item	ACTIVE All active files	FULL Details of loaded simu-	LOADED All simula- tions in load-
index number	•	lation	• ed file
file name	•	•	
analysis program	•	•	•
analysis type	•	•	•
simulation number	٠	•	•
number of simulations in the file	•	•	
simulation status	٠	•	•
unit set		•	
user title	•	•	
file creation date, directory and computer	•	•	
time point (transient) or fre- quency (steady-state ac)	•	•	•
all time points (transient) or frequencies (steady state ac)		•	
linear or non-linear analysis	•	•	•
analysis program specific data		•	
number and types of conduc- tors and current density values		•	
circuit loops		٠	
boundary condition labels		•	
material names	•	•	
material data		•	
numbers of nodes and ele- ments		•	
local coordinate system and replications	•	•	
coordinate limits	•	•	

The **SURFACE** Command

C	T (C 11	·•,	1 (1)
Summary	Integrate field	quantity over	selected surfaces.

Icon

J۲ |

Menu Route	Integrals \downarrow
	Other surface integrals

Command Line Parameters

Command	SURFA	CE		
Parameter	Default	Function		
TAVERAGE	YES	Time-average switch:		
		NO	Calculate integrals at time of SET com-	
			mand.	
		YES	Calculate time-average integrals.	
COMPONENT	Х	Field	component to be integrated.	

Notes 7

I

The SURFACE command integrates a field component expression over the surface selected by the SELECT command (page 5-84). Expressions for the COMPONENT can use as variables any of the system variables listed in section "System Variables" on page 5-10 and any user variables.

The system variables **INTEGRAL** and **AREA** are updated with the value of the integral and the area of the selected surface.

In steady-state alternating current problems, for integrands which are the product of two field quantities, the integral is a function of time with the form

$$F_x = A + B\cos 2\omega t + C\sin 2\omega t \tag{5.9}$$

The values *B* and *C* have little meaning on their own. The time-average value, *A* is the value commonly required. This can be calculated directly using +TAVERAGE.

The values of *B* and *C* can be found by setting the times to 0, 45 and 90 (see "The SET Command" on page 5-88), to give values of F_x at each time: F_0 , F_{45} and F_{90} with the **-TAVERAGE** option.

$$A = \frac{F_0 + F_{90}}{2}$$

$$B = \frac{F_0 - F_{90}}{2}$$

$$C = F_{45} - A$$

(5.10)

The following commands can be used to achieve this:

```
Example - time-average F<sub>x</sub>:
set time=0
surf -tave
$ cons #fx0 fx
set time=45
surf
$ cons #fx45 fx
set time=90
surf
$ cons #fx90 fx
$ para #fxa 0.5*(#fx0+#fx90)
$ para #fxb 0.5*(#fx0-#fx90)
$ para #fxc #fx45-#fxa
```

The values F_0 , F_{45} and F_{90} have little meaning on their own.

The **SYSVARIABLE** Command

Summary Control which system variables are available for field calculations.

Icon:

麗

Menu Route Options ↓ Add system variables Delete system variables List variables in database List variables in program

Command Line Parameters

Command	SYSVARIAB	BLE			
Parameter	Default	Function			
MODE	PROGRAM	Action required:			
		ADD	Add a system variable to the program from the data- base.		
		DBASE	List the system variables in the database.		
		DELETE	Delete a system variable from the program.		
		PROGRAM	List the system variables in the program.		
NAME	none	Stem name of the variable to be added or deleted. ! for a list.			
TYPE	SCALAR	Type of varia	ble to be added or deleted:		
		SCALAR	scalar.		
		VECTOR	vector (X, Y and Z components).		
UNIT	DEFAULT	Unit conversi	on expression.		
		DEFAULT	default unit for known var- iable names.		
		expression	unit expression.		

Command	SYSVARIABLE (continued)			
Parameter	Default	Function		
REFLECTION	DEFAULT	Reflection in model boundaries:		
		DEFAULT	default reflection for known variable names and	
			simulation types.	
		INVERSE	change of sign.	
		NO	no reflection.	
		YES	same sign.	

Notes

I

The **SYSVARIABLE** command lists, adds and deletes system variables.

Most loading of system variables happens automatically within the program, variables being loaded when they are required. For example, for TOSCA magnetostatics, the potential, field strength and flux density are loaded when a database is activated and loaded. The source field strength is loaded automatically if integral coil fields are selected in the SET command. System variables are reloaded when a new simulation is loaded.

However, the user might sometimes need a variable which is available in the database but has not yet been loaded, or, for reasons of efficiency, might want to delete some of the system variables which have been loaded. The **SYSVARIABLE** command provides this functionality as well as listing the variables in the database and program.

See "Current Density in Coils" on page 5-91 for how to add system variables JCX, JCY and JCZ.

Adding System Variables

When system variables are loaded the program follows this procedure:

- TYPE=SCALAR, for example the magnetic scalar potential, NAME=POT: The program reads the real part (RPOT). If the imaginary part (IPOT) is also available it will be read as well. One of the following expressions will be defined:
 - Real scalars:

```
POT=RPOT
```

- Complex scalars:

```
POT=RPOT*COST+IPOT*SINT
POT0=SQRT(RPOT**2+IPOT**2)
POTP=ATAN2D(IPOT;RPOT)
```

where **COST** and **SINT** are calculated from the value of AC time defined by the **SET** command (page 5-88).

The following variables can be used by the user: POT, RPOT and (for steady state ac analyses) IPOT, POT0 and POTP.

- TYPE=VECTOR, for example the magnetic field strength, **NAME=H**: The program reads each component (real and imaginary parts if available) and defines expressions such as:
 - Real vectors:
 - HX=RHX
 - Complex vectors:

```
HX=RHX*COST+IHX*SINT
HX0=SQRT(RHX*2+IHX**2)
HXP=ATAN2D(IHX;RHX)
```

and similarly for the Y and Z components. It also defines the magnitude and the radial and azimuthal components:

```
HMOD=SQRT(HX**2+HY**2+HZ**2)
HR=(HX*X+HY*Y)/R
HT=(HX*Y-HY*X)/R
```

The following variables can be used by the user: HX, RHX, HY, RHY, HZ, RHZ, HMOD, HR, HT and (for steady-state ac analyses) IHX, IHY, IHZ, HX0, HY0, HZ0, HXP, HYP and HZP.

• TYPE=VECTOR, for example, **NAME=E_**: The program interpolates the edge or face values (indicated by '_')to provide vector quantities, e.g E_X, E_Y and E_Z and E_MOD, at each field point.

The following variables can be used by the user: E_X, RE_X, E_Y, RE_Y, E_Z, RE_Z, E_MOD.

The unit conversion expression for the quantity represented by the system variable should also be given so that the values are scaled appropriately to the user's choice of units. Unit expressions should be given in terms of the following names, which correspond to the parameter names of the UNITS command (page 5-112). The program recognises the variable names used by CARMEN, ELEKTRA, SCALA, SOPRANO, TEMPO and TOSCA and applies the correct unit expression if UNIT=DEFAULT is specified.

Unit conversion factors: LENGU, FLUXU, FIELU, SCALU, VECTU, CONDU, CURDU, POWEU, FORCU, ENERU, ELECU, DISPU.

The program also needs to know how to adjust the signs of the system variables when returning values in reflected images of the mesh. **REFLEC-**TION=YES gives the same symmetry as the principal field of the simulation, i.e. **H** for magnetic field problems (CARMEN, ELEKTRA, SOPRANO and TOSCA), **E** for electric field problems (SCALA and TOSCA) and ∇T for thermal problems (TEMPO). The program recog-

nises the variable names used by CARMEN, ELEKTRA, SCALA, SOPRANO, TEMPO and TOSCA and applies the correct reflection if **REFLECTION=DEFAULT** is specified.

Remember that when referencing system variables in the SYSVARIABLE command, only the stem of the name should be given after removing the leading \mathbf{R} or \mathbf{I} and, for a vector, the trailing \mathbf{X} , \mathbf{Y} or \mathbf{Z} .

Deleting System Variables

When system variables are deleted from the program it follows this procedure:

- TYPE=SCALAR, for example the magnetic scalar potential, NAME=POT: The program deletes the real part (RPOT). If the imaginary part (IPOT) is available it will be deleted as well. The following variables will be marked as no longer being available: RPOT, IPOT.
- TYPE=VECTOR, for example the magnetic field strength, NAME=H: The program deletes each component (real and imaginary parts if available). The following variables will be marked as no longer being available: RHX, IHX, RHY, IHY, RHZ, IHZ.

Remember that when referencing system variables, only the stem of the name should be given after removing the leading \mathbf{R} or \mathbf{I} and, for a vector, the trailing \mathbf{X} , \mathbf{Y} or \mathbf{Z} .

Listing System Variables

4 types of list are available:

- MODE=ADD, NAME=! and MODE=DBASE lists the system variables available in the database; MODE=DBASE also shows the type of variable:
 - *Node No. indexed* indicates a variable which is continuous throughout the model.
 - *Node indexed multivalued by material* indicates a variable which is discontinuous at the surfaces of materials.
 - *Node indexed multivalued by potential code* indicates a variable which is discontinuous at the surfaces of different potential types.
 - *Element No. indexed* indicates a variable which has a constant value over each element.
 - *Edge No. indexed* indicates solution of edge variable elements. This should be loaded as a vector.

- *Face No. indexed* indicates an additional solution from edge variable elements. It should also be loaded as a vector.

In general the last entries in the list are the results of the simulation. The earlier entries are stored by the software to enable restarts.

• MODE=DELE, NAME=! and MODE=PROGRAM lists the system variables in the program. MODE=PROGRAM also shows those defined within the software which are marked (program) and are always available and so cannot be deleted. Those marked (user) have been loaded from the database and therefore can be deleted.

The TABLE Command

Summary	Read and write table files of field values	at points.

Menu Route	Tables \downarrow				
	Read	and	write	table	files

Command Line Parameters

Command	TABLE	
Parameter	Default	Function
INFILE	none	Name of input data file.
OUTFILE	none	Name of output data file.
F1	Х	Expression for values in column 1
F2	Y	Expression for values in column 2
F3	Z	Expression for values in column 3
F4,, F12	none	Expression for values in column 4
		to 12
NAME1,, NAME12	F1,, F12	Names for columns
UNIT1	LENGU	Unit expression for column 1
UNIT2	LENGU	Unit expression for column 2
UNIT3	LENGU	Unit expression for column 3
UNIT4,, UNIT12	1.0	Unit expressions for columns 4 to
		12
COLUMNS	4	Number of columns in output file.

Notes

The TABLE command is provided to facilitate an interface to other post processing programs. It performs three tasks:

- 1. Read an input source, given by INFILE, to obtain field point coordinates and other columns of values.
- 2. Calculate field values at the points, if necessary.
- 3. Output the fields.

The value of **INFILE** specifies the source of the field point coordinates:

• INFILE=TEMP: the field points contained in the internal buffer created by the ARC, CARTESIAN, CIRCLE, FIT TYPE=LEGENDRE, LINE and POLAR commands are processed.

- INFILE=NODE or INFILE=DBAS: the coordinates of the nodes in the current database are used.
- INFILE=ELEM: the coordinates of the element centroids in the current database are used.
- INFILE=SELE: the coordinates of the nodes on the selected surfaces are used.
- INFILE=*filename*: field point coordinates are read from a table file.

The value of **OUTFILE** specifies the destination of the field values:

- OUTFILE=TEMP: the current set of field values is calculated and remain in the programs internal buffer.
- OUTFILE=DBAS: this is intended for use with INFILE=filename, where the file contains coordinates and additional fields. If the number of entries matches the number of nodes or the number of elements in the current database, the field values will be added to the database as nodal or element data as appropriate.
- OUTFILE=NULL: this option does no field calculation or output, but allows the format of an input file to be tested.
- OUTFILE=*filename*: the available set of field values is calculated and the values up to 12 component expressions are output to a table file. The number of values is specified by the COLUMNS parameter.

The expressions for the field values, F1 to F12, can use as variables any of the system variables listed in section "System Variables" on page 5-10 and any user variables. They are evaluated at each field point in the input file. All the values for one point appear in one record of the output file.

The name and unit expression for each column can also be set using parameters NAMEn and UNITn. If a NAMEn is not specified the corresponding Fn will be used instead.

If no file name extension is given, the extension *table* is supplied for the input and output data files.

The format of the data files is given in section "TABLE Files" on page 5-19 which includes an example output data file.

SCALA The following sequence of commands can be used to add fields from a magnetostatics analysis to a SCALA database, so that the SCALA particle tracking can make use of the magnetic fields.

I

I

The first step is to create a database file for SCALA, using either the Modeller or the pre processor. Before solving the database with SCALA, the "unsolved" database must be loaded into the post processor.

After the following commands have been completed, the SCALA problem should then be run.

/ Activate and load SCALA database acti file=space.op3 load / Form table of node coordinates tabl infi=node outf=nodes f1=x f2=y f3=z unit1=lengu unit2=lengu, unit3=lengu columns=3 / Activate and load magnetostatic problem acti file=magnet.op3 load / Form table of magnetic fields at nodes of SCALA model tabl infi=nodes outf=fields f1=x f2=y f3=z f4=rhx f5=rhy f6=rhz, unit1=lengu unit2=lengu unit3=lengu unit4=fielu unit5=fielu, unit6=fielu columns=6 / Add the fields to the SCALA database load file=1 tabl infi=fields outf=dbas

This technique can also be used to add magnetic fields to a TOSCA database which contains an electrostatic model or electric fields to a database containing a magnetostatic model, so that combined electric and magnetic field particle tracking can be done (see "The COMBINE Command" on page 5-38). It is also used by TEMPO to include eddy current heating (see OPERA-3d User Guide).

The THREED Command

Summary	Control the display of the geometry.
Icons	10 FX FX FX FY FX FZ FZ
Menu Routes	View↓ 3d display Refresh Views Fields↓
	Fields on the surface of the model

Command Line Parameters

I

Command	THREED			
Parameter	Default	Function		
TYPE	SURFACE	Type of display:		
		SURFACE	Surfaces of the model in material colours.	
		COMPONENT	Contours of field COM- PONENT on model.	
COMPONENT	Х	Field component for contours.		
MIN	*	Minimum contour value. * for automatic setting.		
MAX	*	Maximum contour value. * for automatic setting.		
VECTORS	NO	Vector display switch:		
		NO	Vectors not displayed.	
		YES	VECTORS displayed as	
			ment facets using com- ponents VX, VY and VZ,	
			and current directions in	
			conductors.	
VX	X	Expression for x-component of vectors.		
VY	Y	Expression for y-component of vectors.		
VZ	Z	Expression for z-component of vectors.		

Command Parameter OPTION

SIZE

ROTX ROTY ROTZ XORIGIN YORIGIN ZORIGIN

PERSPECTIVE

FACETANGLE

LINECOLOUR

SCALE

YES

1

10

YES

THREED (continued)			
Default	Function		
REFRESH	Command option:		
	GETVIEW	Retrieve view parame-	
		ters after mouse interac-	
		tion.	
	REFRESH	Refresh picture without	
		changing the view.	
	SETVIEW	Refresh picture using the	
		view parameters.	
10	Display extends from the origin by SIZE		
	in each direction. SIZE=0 requests the		
	initial view of the model		
20	Rotation of model around X axis.		
20	Rotation of model around Y axis.		
0	Rotation of model around Z axis.		
0	X coordinate at centre of picture		
0	Y coordinate at centre of picture		
0	Z coordinate at centre of picture		

Perspective switch:

Scaling factor for vectors.

Colour used for outlines:

Perspective view.

Material colour.

Text colour.

Largest angle subtended by a curved facet.

Orthographic view.

YES

NO

YES

NO

Notes

The **THREED** command updates the 3D picture of the model. The pictures consists of the three dimensional geometry of the finite element mesh and conductors. The **THREED** command uses as data the 'display buffer', which is created by the **SELECT** command. If 'flux-tubes' have been added to the display buffer by the VIEW command, they will be displayed as well. Pictures can be line-drawings or can show coloured surfaces. It is also possible to overlay the geometry with displays of the field quantities.

The colours in the pictures can represent the materials (TYPE=SUR-FACE) or alternatively, scalar field quantities, or single components of vector field quantities can be displayed as coloured zone contours (TYPE=COMPONENT). VECTOR field quantities can be displayed as vectors at the centroid of the element facets.

Expressions for the scalar COMPONENT and vector components (VX, VY, VZ) can use as variables any of the system variables listed in section "System Variables" on page 5-10, any user variable or numerical parameter name. All 3 components are scaled by the vector scaling factor, SCALE.

The values of the maximum and minimum contour zones can be set with MAX and MIN. If either MAX or MIN is set to * the value is calculated from the range of values to be displayed. The number of coloured zones is set by the number of colours available on a particular display type. The system variables MAXIMUM and MINIMUM are updated by the command.

After a new database is loaded, the first use of the command **THREED OPTION=REFRESH** will refresh the display to the default view which matches the size and position of the model. The view can be adjusted using the mouse buttons:

- left button: rotate
- middle button: zoom
- right button: pan

See "The MOUSE Command" on page 5-75.

The **OPTION** parameter also controls the view of the model:

- **OPTION=GETVIEW**: updates the values of the parameters **SIZE**, **ROTX**, **ROTY**, **ROTZ**, **XORIGIN**, **YORIGIN** and **ZORIGIN** following interaction with the mouse.
- **OPTION=SETVIEW**: uses the current values of SIZE, ROTX, ROTY, ROTZ, XORIGIN, YORIGIN and ZORIGIN.

SIZE=0: gives an initial view of the visible parts of the model, contours, vectors and trajectories depending on the settings of the WIN-DOW command parameters (page 5-124).

• **OPTION=REFRESH**: updates the picture if necessary without changing the view unless a new database has been loaded in which case the default view is used.

The MAP command (page 5-72) and VIEW command (page 5-115) can add contour maps or particle trajectories to the picture and the different parts of the display (axes, solid colours, outline, contour map, vectors, etc.) can be temporarily removed using the WINDOW command (page 5-124).

The TITLE Command

Summary Add title, date and time to the display.

Menu Route

View↓ Title

Command Line Parameters

I

Command	TITLE			
Parameter	Default	Function		
STRING	none	A graphics window title.		
POSITION	NONE	Title position:		
		BOTTOMCENTRE Bottom centre		
		BOTTOMLEFT	Bottom left	
		BOTTOMRIGHT	Bottom right	
		NONE	No title	
		TOPCENTRETop centre		
		TOPLEFT	Top left	
		TOPRIGHT	Top right	
DATE	TOPLEFT	Time/date position:		
		BOTTOMCENTRE	Bottom centre	
		BOTTOMLEFT	Bottom left	
		BOTTOMRIGHT	Bottom right	
		NONE	No date and time	
		TOPCENTRE	Top centre	
		TOPLEFT	Top left	
		TOPRIGHT	Top right	

Notes

The TITLE command controls the display of a title and the date and time. There is a choice of 6 positions for each. If the same position is chosen for both the title and the date, the title appears above the date and time. The version number of the software can be included in the title using the string and system variables, VERSION:

title string='Version: &version&'

or

```
title string='Version: %real(version,5)'
```

The TRACK Command

Summary	Calculate trajectories of charged particles in electric and magnetic fields.
Menu Route	Trajectories \downarrow Calculate particle trajectories

Command Line Parameters

I

Command	TRACK			
Parameter	Default	Function		
X0	0	Initial X-coordinate of the particle.		
Y0	0	Initial Y-coordinate of the particle.		
ZO	0	Initial Z-coord	linate of the particle.	
THETA	0	Theta Euler angle defining particle		
		direction.	direction.	
PHI	0	Phi Euler angl	e defining particle direc-	
		tion.		
PSI	0	Psi Euler angle defining particle direc-		
		tion.		
VOLTAGE	1	Acceleration VOLTAGE or initial		
		energy.		
CHARGES	-1	CHARGES on the particle in elemen-		
		tary charge units. -1 is the charge on an		
		electron.		
MASS	1	Particle rest MASS in electron rest mass		
DEAMOUDDENT	1	units.		
BEAMCURRENT	1	Current associated with the track. It can		
		be an expression in terms of the electric		
STED	1	STED longth between output points		
NOTED	1	STEP length between output points.		
NSTEP	100	Number of steps to be calculated.		
TOLERANCE	0.01	Accuracy required.		
OPTIONS	PARTICLE	Options:		
		PARTICLE	Single PARTICLE	
		FLUXTUBE	Flux line	
		BEAM	A test pattern defined by	
			PATTERN and LINES.	

Commond	TDACK (continued)		
Command	IRACK (continued)		
Parameter	Default	Function	
PATTERN	1.E-4	Size of the rectangular grid TEST pat-	
		tern.	
LINES	5	Number of X and Y lines in a TEST pat-	
		tern.	
FILE	none	Name of file to save trajectories.	
STATUS	NEW	STATUS of the TRACK file:	
		NEW	Create a NEW file.
		OLD	Append data to an OLD
			file.
		CLEAR	Overwrite data in an
			OLD file.
PRINT	YES	PRINT trajectory coordinates to <i>Opera3d_Post_nn.lp</i> : YES or NO.	

The TRACK command calculates the trajectories of charged particles

through the electric and/or magnetic fields (including full relativistic correction) or follows flux lines. Combined field particle tracking is available for SOPRANO-SS simulations and also when additional fields have been added to an electrostatic or magnetostatics simulation (see section "The COMBINE Command" on page 5-38).

The parameter, **OPTION** selects single particles (**OPTION=PARTICLE**), test patterns (OPTION=BEAM) or flux lines (OPTION=FLUXTUBE).

Tracking starts at the point X0, Y0, Z0. The initial coordinates (and direction) are defined in the local coordinate system of the SET command. In steady-state ac simulations the time at the start of the track is also given by the SET command (page 5-88).

For charged particles, the initial direction (defined by the Euler angles THETA, PHI and PSI, see section "Euler Angles" on page 2-34), the initial energy (VOLTAGE), CHARGES and MASS of the particles can be set. The initial direction is along the z-axis of the local coordinate system defined by the Euler angles. The **BEAM** test pattern option allows many particles to be fired from the plane normal to the initial particle direction, from the intersection points of a square orthogonal grid. The grid can be given a size (PATTERN) and the number of grid LINES in each direction can be set.

A current is associated with each track. This can be set explicitly to a value in amps or can be calculated by the program from an expression in terms of the electric field (EX, EY, EZ), coordinates (X, Y, Z) and velocity (VELX, VELY, VELZ) at the start of the trajectory.

Notes
The calculation is controlled by the STEP length along the trajectory, the number of steps (NSTEP) and the relative TOLERANCE which applies to coordinates and velocities or flux densities. STEP only determines the distance between the displayed points on the trajectory (although the value should be related to the minimum size of any feature of the field distribution) - the TOLERANCE is achieved by adaptive integration. STEP is measured along the trajectory and hence the maximum total trajectory length calculated is STEP*NSTEP.

Note: there is a limit of 5000 steps per trajectory.

The trajectory coordinates are stored in a binary file. When the file status is set to NEW, a new file will be created to contain the data. Setting status to OLD causes the trajectories to be appended to an existing file. The third option CLEAR causes an old file to be overwritten. (The format of the track file is described in section "TRACK files" on page 5-21.) The trajectories stored in a TRACK file can be displayed and processed in a number of ways using the VIEW command (page 5-115).

The UNITS Command

Summary Set units to be used for physical quantities.

Icon



Menu Route

Options \downarrow Units

Command Line Parameters

Command	UNITS		
Parameter	Default	Function	
LENGTH	СМ	Unit for length:	
		СМ	centimetre
		INCH	inch
		METRE	metre
		MICRON	micron
		MM	millimetre
FLUX	GAUSS	Unit for magnetic flux density:	
		GAUSS	gauss
		KGAUSS	kilogauss
		TESLA	tesla
FIELD	OERSTED	Unit for magnetic field strength:	
		AM	ampere metre ⁻¹
		OERSTED	oersted
SCALAR	OCM	Unit for mag	netic scalar potential:
		AMP	ampere
		OCM	oersted centimetre
VECTOR	GCM	Unit for vector potential:	
		GCM	gauss centimetre
		WBM	weber metre ⁻¹

5-112

5-113

Command	UNITS (cont	tinued)	
Parameter	Default	Function	
CONDUCTIVITY	SCM	Unit for conductivity:	
		SCM	siemens centimetre ⁻¹
		SIN	siemens inch ⁻¹
		SM	siemens metre ⁻¹
		SMM	siemens millimetre ⁻¹
		SMU	siemens micron ⁻¹
CURD	ACM2	Unit for curr	ent density:
		ACM2	ampere centimetre ⁻²
		AIN2	ampere inch ⁻²
		AM2	ampere metre ⁻²
		AMM2	ampere millimetre ⁻²
		AMU2	ampere micron ⁻²
POWER	WAT	Unit for pow	ver:
		ERGS	erg second
		HP	horse power
		WATT	watt
FORCE	NEWTON	Unit for force:	
		DYNE	dyne
		GRAMME	gramme force
		KG	kilogramme force
		LBF	pound force
		NEWTON	newton
ENERGY	JOULE	Unit for ener	gy:
		BTU	British thermal unit
		ERG	erg
		JOULE	joule
ELECTRIC	VCM	Unit for elec	tric field strength:
		VCM	volt cm ⁻¹
		VIN	volt inch ⁻¹
		VM	volt metre ⁻¹
		VMM	volt millimetre ⁻¹
		VMU	volt micron ⁻¹

Command	UNITS (continued)			
Parameter	Default	Function		
DISPLACEMENT	CCM2	Unit for electric flux density:		
		CCM2 coulomb cm ⁻²		
		CM2	coulomb metre ⁻²	
MASS	GRAMME	Unit for mass:		
		GRAMME	gramme	
		KG	kilogramme	

Notes

The UNITS command sets the units to be used to interpret user input and display geometric and field data. Each of the parameters can be set to one of a series of predefined character strings corresponding to commonly used units.

N.B. The unit of electric scalar potential is always volt and of charge density is coulomb *length_unit*⁻³. The unit of temperature is always degrees Kelvin. Temperature gradient is given in K/length_unit, whilst heat flux is in *power_unit/length_unit*.

Unit conversion is performed on the basic system variables. Other system variables are derived from these, and will have the correct units only if consistent units are used for the basic quantities.

When a database is loaded, the current set of units is overwritten with the unit set chosen in the Modeller or pre processor when the database was created (see section "The SOLVERS Command" on page 4-150).

The current set of units is listed at the right hand side of the display.

The UNITS command updates a set of conversion factors for use in expressions. They are based on the equivalent parameters names, taking the first 4 characters, and adding the character U, giving: LENGU, FLUXU, FIELU, SCALU, VECTU, CONDU, CURDU, POWEU, FORCU, ENERU, ELECU, DISPU, MASSU. Values in internal units are divided by the unit factors to convert to user units.

The VIEW Command

Summary Display and process particle trajectories.

Menu Route Trajectories ↓ Display particle trajectories Graph trajectories Intersect trajectories with patch Current density map Create flux tubes

Command Line Parameters

I

I

	<u>-</u>		
efault	Function		
one	Name of file containi	ing trajectories.	
ISPLAY	Type of output requir	red:	
	DISPLAY	DISPLAY trajectories	
		on the current view of	
		the model.	
	INTERSECTIONS	Draw graphs of	
		INTERSECTIONS of	
		the trajectories with	
		CARTESIAN or	
		POLAR patch	
	TEMP	Calculate current den-	
	sities from intersec-		
		tions with	
		CARTESIAN or	
		POLAR patch.	
	TRACKS Draw graphs of		
		jectories.	
	TUBE	Create tubes of dis-	
		playable facets around	
		trajectories.	
,	The variable plotted on the horizontal axis of		
	the graph.		
	The variable plotted on the vertical axis of		
	the graph.		
	Lower limit for graph horizontal axis. (* for		
	automatic setting of limit.)		
	efault me ISPLAY	efault Function Mame of file containing ISPLAY Type of output require DISPLAY INTERSECTIONS TEMP TRACKS TUBE The variable plotted of the graph. Lower limit for graph automatic setting of 1	

5-115

Command	VIEW (co	ntinued)		
Parameter	Default	Function		
XMAXIMUM	*	Upper limit for graph horizontal axis. (* for automatic setting of limit.)		
YMINIMUM	*	Lower limit for graph automatic setting of	h vertical axis. (* for limit.)	
YMAXIMUM	*	Upper limit for graph automatic setting of	n vertical axis. (* for limit.)	
LINE	YES	Graph plotting style:		
		NO	Plot using symbols.	
		YES	Plot using lines.	
COLOUR	YES	Use of colour in the	displays:	
		FUNCTION	Colours represent the values of COMPO- NENT.	
		NO	Use text colour.	
		YES	Colours represent tra- jectory numbers.	
COMPONENT	X	Expression used to assign colours to the tra- jectories when COLOUR=FUNCTION or the radius of the tubes when PLOT=TUBE. The value is also printed when printing is enabled		
PRINT	NO	Printing switch:		
		NO YES	No printing. All points or intersec- tions printed to dia- logue file.	
SAMPLE	8	Sample size used to calculate current density for PLOT=INTERSECTIONS and PLOT=TEMP.		
ERASE	YES	Erase previous display (only for PLOT=INTESECTIONS and PLOT=TRACKS)		
		NO	Add to existing dis- play.	
		YES	Erase display first.	
SYMMETRY	NO	Include symmetry copies of tracks in DIS- PLAY, INTERSECTIONS and TUBES: YES or NO.		

I

I

Notes

The VIEW command re-displays and processes trajectories calculated by the TRACK command (page 5-107), or the Space Charge analysis program SCALA, and stored in TRACK files (see page 5-21).

Types of VIEW

- PLOT=DISPLAY: this option allows the trajectories to be displayed on the current view of the model using the text colour or multiple colours (page 5-118). The trajectories are overlaid on the three dimensional pictures of the geometry created by the THREED command (page 5-104). The tracks can be temporarily removed from the display using the WIN-DOW command (page 5-124).
- PLOT=TRACKS: this option displays graphs of the trajectories. The variables plotted on the axes of the graphs can be selected from the set of active system variables (page 5-118). For example to display an axisymmetric projection of the results, with Z on the horizontal axis of the graph, use

VIEW XAXIS=Z,YAXIS=R

By default, the axis limits are automatically set to contain the functions plotted but limits can be specified.

PLOT=INTERSECTIONS: The intersection positions on the current CARTESIAN or POLAR patch are calculated for all trajectories and symmetry images. These intersections are ordered with respect to the independent variable defined as the horizontal axis of the output graph. The current in the beams is projected onto the independent variable and the linear current density function, DENSITY, is calculated and added to the set of active system variables. The intersections are plotted as graphs using the set of active system variables (page 5-118). When the intersections are calculated, the linear current density (DENSITY) is also computed for each intersection point. This is evaluated statistically using varying sample sizes up to a maximum given by the SAMPLE parameter. The expected error is also calculated and will be displayed using error bars if YAXIS=DENSITY is selected. The sample size

should be less than $\frac{1}{10}$ of the number of intersections. If larger values

are used the results will be smooth but the errors will be larger.

For example: to find the radial distribution of current density for a circular beam, use

view plot=inte,yaxi=density,

xaxi=sqrt((x-x0beam)**2+(y-y0beam)**2+(z-z0beam**2))

This will collect together all the intersections at the similar radii in order to calculate the density as a function of radial coordinate.

 PLOT=TEMP: The intersection positions on the current CARTESIAN or POLAR patch are calculated for all trajectories and symmetry images. The current density is then calculated for each point on the patch by weighted integration over the patch and the values are smoothed using a fast fourier transform (FFT) with a SAMPLE size which determines the amount of smoothing. The current density (J) and power density (PJ) can then be displayed using the MAP command (page 5-72).

PLOT=TUBE: This option creates additional facets in the display buffer. The facets for tubes around the segments of the trajectories, with the radius determined by the **COMPONENT** expression. The tubes can then be displayed using the **THREED** command (page 5-104).

The flux tubes can only be cleared from the display buffer with the SELECT command. Use the Repeat selection and refresh icon

G which performs a **SELECT** and **THREED** command.

Viewing In Colour

In each of the PLOTTING options, the lines can be displayed using the text colour (COLOUR=NO), with colours representing the trajectory number (COLOUR=YES) or with colours representing some characteristic of the position along the trajectory (COLOUR=FUNCTION).

Functional colours are selected using the value of the COMPONENT expression. COMPONENT can be assigned to expressions in terms of the active system variables listed below. For example,

COLOUR=FUNCTION, COMPONENT=TOF

will display the graphs coloured according to the time of flight from the start points.

To colour flux tubes, the **COMPONENT** has to be specified in the **THREED** command. Colour contours are displayed on the surfaces which were added to the display buffer.

Active System Variables

The VIEW command has its own set of system variables which are available in expressions for the graph plotting variables (XAXIS and YAXIS) and the COMPONENT used to determine the colours of the lines.

	System	Variables for PLOT=DISPLAY and PLOT=TRACKS
Х		X coordinates of points on the trajectory.
Y		Y coordinates of points on the trajectory.
Z		Z coordinates of points on the trajectory.

System Variables for PLOT=DISPLAY and PLOT=TRACKS			
R	Radial coordinate of points on trajectory.		
XSTART	X coordinate of the first point on the trajectory.		
YSTART	Y coordinate of the first point on the trajectory.		
ZSTART	Z coordinate of the first point on the trajectory.		
RSTART	Radial coordinate of the first point on trajectory.		
VELX	X component of particle velocity.		
VELY	Y component of particle velocity.		
VELZ	Z component of particle velocity.		
CURRENT	The current in the track (beamlet).		
TOF	Time of flight from the start of the trajectory.		
Q	Electronic charge on the particle.		
М	Mass of the particle in electron units.		

S	ystem Variables for plot=intersections
Х	X coordinates of the intersection points.
Y	Y coordinates of the intersection points.
Z	Z coordinates of the intersection points.
X0BEAM	X coordinate of the centre of the beam on the intersection surface.
YOBEAM	Y coordinate of the centre of the beam on the intersection surface.
ZOBEAM	Z coordinate of the centre of the beam on the intersection surface.
TXBEAM	X component of the mean direction vector for the beam at the intersection surface.
TYBEAM	Y component of the mean direction vector for the beam at the intersection surface.
TZBEAM	Z component of the mean direction vector for the beam at the intersection surface.
DENSITY	The calculated linear current density in the beam defined by the set of tracks in the track file (units are amp).
VELX	X component of particle velocity.
VELY	Y component of particle velocity.
VELZ	Z component of particle velocity.
CURRENT	The current in the track (beamlet).
TOF	Time of flight from the start of the trajectory.
Q	Electronic charge on the particle.
Μ	Mass of the particle in electron units.

System Variables for PLOT=TUBE			
Х	X coordinates of the intersection points.		
Y	Y coordinates of the intersection points.		
Z	Z coordinates of the intersection points.		
VELX	X component of particle velocity.		
VELY	Y component of particle velocity.		
VELZ	Z component of particle velocity.		
CURRENT	The current in the track (beamlet).		
TOF	Time of flight from the start of the trajectory.		
Q	Electronic charge on the particle.		
Μ	Mass of the particle in electron units.		
All available field quantities.			

Printing Trajectory Data

When printing is selected (**PRINT=YES**) and **COLOUR=FUNCTION** has been set, the following data is output to the dialogue file, *Opera3d_Post_n.lp*:

• **PLOT=DISPLAY** and **PLOT=TRACKS**, for all points in all trajectories:

X Y Z VELX VELY VELZ COMPONENT

• **PLOT=INTERSECTIONS**, for all intersections:

CURRENT X Y Z VELX VELY VELZ COMPONENT

N.B. The unit of current density is amp *length_unit*⁻².

The VOLUME Command

es.

Icon



Menu Route	Integrals \downarrow		
	Other volume integrals		

Command Line Parameters

Command	VOLUME		
Parameter	Default	Function	
ACTION	INTEGRATE	Create list of volumes or integrate:	
		ADD	Add volume(s) to list.
		INTEGRATE	Integrate.
		REMOVE	Remove vol- ume(s) from list.
		RESET	Empty list of vol- umes.
LABEL	ALL_VOLUMES	S Volumes to be added or removed from list:	
		material	Material name.
		label	Volume label.
		ALL_VOLUMES	All volumes.
TAVERAGE	YES	Time-average switch.	
		NO	Calculate inte-
			grals at time of SET command.
		YES	Calculate time- average integrals.

Command	VOLUME (continued)		
Parameter	Default	Function	
ADAPT	NO	Adaptive integration switch.	
		NO YES	Use 8 gauss- points in each element. Use up to 216 gauss-points in each element.
COMPONENT	Х	Field component to be integrated.	

Notes

The VOLUME command integrates a field component expression in the whole problem space or in labelled sets of elements. Expressions for the COMPONENT can use as variables any of the system variables listed in section "System Variables" on page 5-10 and any user variables.

The command operates as a 2-stage process:

- 1. form a list of volumes. Initially all volumes are in the list.
 - **ACTION=RESET** empties the list.
 - **ACTION=ADD** adds volumes to the list by LABEL.
 - **ACTION=REMOVE** removes volumes from the list by LABEL.
 - Labels can be material names, volume labels including element and potential types and user labels or ALL_VOLUMES.
- 2. integrate forces using **ACTION=INTEGRATE**.

The system variables **INTEGRAL** and **VOLUME** are updated with the value of the integral and the total volume integrated over.

The basis for the integration is the finite element mesh. The integrals are performed in each element using first-order gaussian quadrature. However, in reduced potential volumes if the coil field is calculated by integration or anywhere if the total field is calculated by integration, the first-order quadrature is insufficient to match the field variation in an element. Switching on adaptive integration (+ADAPT) enables the program to use up to 9^{th} -order gaussian quadrature in each element to increase the accuracy of the integrals. See section "The SET Command" on page 5-88 for information about field calculation methods.

Reflected and rotated images of the model, specified by the ACTIVATE command, are NOT included in the integrations done by the VOLUME command.

I

In steady-state alternating current problems, for integrands which are the product of two field quantities, the integral is a function of time with the form

$$E = A + B\cos 2\omega t + C\sin 2\omega t \tag{5.11}$$

The values *B* and *C* have little meaning on their own. The time-average value, *A* is the value commonly required. This can be calculated directly using +TAVERAGE.

The values of *B* and *C* can be found by setting the times to 0, 45 and 90 (see section "The SET Command" on page 5-88), to give values of *E* at each time: E_0 , E_{45} and E_{90} with the **-TAVERAGE** option.

$$A = \frac{E_0 + E_{90}}{2}$$

$$B = \frac{E_0 - E_{90}}{2}$$

$$C = E_{45} - A$$

(5.12)

The following commands can be used to achieve this:

```
• Example - time-average energy:
set time=0
volu comp=0.5*(bx*hx+by*hy+bz*hz) -tave
$ cons #en0 integral
set time=45
volu
$ cons #en45 integral
set time=90
volu
$ cons #en90 integral
$ para #ena 0.5*(#en0+#en90)
$ para #enb 0.5*(#en0-#en90)
$ para #enc #en45-#ena
```

The values E_0 , E_{45} and E_{90} have little meaning on their own.

The WINDOW Command

Summar	Show	or hide parts	of the dis	splav
Summe	, , , , , , , , , , , , , , , , , , , ,	or mae parts	or the an	praj

Icons



Menu Route

View \downarrow Parts of the display

Command Line Parameters

Command	WINDO	W		
Parameter	Default	Function		
AXES	YES	Show coordin	nate axes: YES or NO	
SOLID	YES	Show solid v	iew of model: YES or NO	
OUTLINE	YES	Show outline	e view of model: YES or NO	
VECTORS	YES	Show vectors on the surface of the model: YES or NO		
CONTOURMAP	YES	Show contour map: YES or NO		
VECTORMAP	YES	Show vector map: YES or NO		
TRACKS	YES	Show trajectories: YES or NO		
LABELS	YES	Show contou	r or trajectory labels:	
		CONTOUR	Show labels for contour maps.	
		CYCLE	Cycle through available labels and none.	
		NO	Show no labels.	
		SURFACE	Show labels for contours on the surface of the model.	
		TRACK	Show labels for trajectories.	
		YES	Show default labels.	

The WINDOW command can be used to hide or show again parts of the display which exist. For example, if a contour map is added to the display with command, can the MAP it be hidden using WINDOW CONTOURMAP=NO or shown again using WINDOW CONTOURMAP=YES.

One set of labels for contour or trajectory colours can be displayed. If more than one set of labels is available, commands such as **WINDOW LABEL=SURFACE** can be used to display a particular labels set. In this case, for contours on the surface of the model will be shown. Alternatively, **WINDOW LABEL=CYCLE** can be used to cycle through the available label sets. This form of the command can be issued by clicking on the icon.

System Variables

The following is a list of system and string variables which are defined and updated by the program. As described in an earlier section (page 5-10), more system variables are defined to represent fields and potentials when databases are loaded; those names are not listed here.

Many of the string variables (e.g., ACTIVATE_ANGLES are defined so that the GUI dalog boxes can present the correct view of the current settings.

The variables are annotated **n** for numerical system variables or **s** for string variables; those marked with * are not intended for user use.

s*	ACTIVATE_ANGLES	Current settings of ACTIVATE parameters THETA, PHI and PSI:	
		ACTIVATEXYZ	0, 0, 0
		ACTIVATEYZX	90, 0, 90
		ACTIVATEZXY	90, 90, 180
		ACTIVATEOTHER	any other angles
n	ANGLE	Rotor angle of a CARM	MEN simulation.
n	AREA	Cross sectional area of conductor during CON- DUCTOR MODIFY. Surface area returned by SURFACE command.	

n	С	Speed of light.	Speed of light.	
n	CASE	Simulation number set mand.	Simulation number set by the ACTIVATE com- mand.	
n	CASES	Number of simulations	Number of simulations in database set by the ACTIVATE command.	
n	CHARGE	Charge density.		
s*	СМІМ	Value of the \$COMIN	PUT MODE parameter	
n	COENERGY	Co-energy calculated l	by the ENERGY command.	
n	COLn	Column values (n=1,	.,20) in GRAPH command.	
n	CONDU	Unit factor for conduc	tivity.	
n	CONDUCTORS	Number of conductors		
S	CONDUCTOR_TYPE	Conductor type during	CONDUCTOR MODIFY.	
s*	COND_PICK	Is conductor picking e	nabled?	
		STARTPICK	enabled	
		STOPPICK	disabled	
n	COST	In AC problems, cost	In AC problems, <i>cos</i> ω <i>t</i>	
n	CURDU	Unit factor for current	Unit factor for current density.	
n	CURRENT	Current in trajectories command.	Current in trajectories calculated by the VIEW command.	
s*	CUT_ANGLES	Current settings of SE and PHI:	Current settings of SELECT parameters THETA and PHI:	
		CUTXY	0,0	
		CUTYZ	90, 0	
		CUTZX	90, 90	
		CUTOTHER	any other angles	
n	DENSITY	Current density in traje	ectories calculated by the	
n	DISPU	Unit factor for displace	ement current.	
n	ELECENER	Energy in electric field calculated by the ENERGY command.		
n	ELECU	Unit factor for electric	field strength.	
n	ELEMENT	Element number.		
n	ENERGY	Energy in magnetic fie ENERGY command.	Energy in magnetic field calculated by the ENERGY command.	
n	ENERU	Unit factor for energy.		
n	EPSILON0	Permittivity of free spa	Permittivity of free space.	

n	FIELU	Unit factor for magnetic field strength.	
n	FILEEXISTS	Set by \$EXIST command:	
		1 file exists	
		0 file does not exist	
n	FLUXU	Unit factor for magnetic flux density	
n	FMOD	SQRT(FX*FX+FY*FY+FZ*FZ)	
n	FORCU	Unit factor for force.	
n	FREQ	Frequency or rotor speed (rpm).	
n	FX, FY, FZ	Force calculated by the BODY, ENERGY and	
s*	GLOBALVECTORXYZ	Holds the vector name in MAP and THREED, if	
		∇X , ∇Y and ∇Z are x, y, and z components of the	
		same vector.	
n	HCMOD	SQRT(HCX*HCX+HCY*HCY+HCZ*HCZ)	
n	HCX, HCY, HCZ	Coercive force.	
n	HCR	(HCX*X+HCY*Y)/R	
n	НСТ	(HCY*X-HCY*X)/R	
n	HYSENERGY	Hysteretic energy calculated by the ENERGY	
		command.	
n	HYSPOWER	Hysteretic power calculated by the ENERGY com-	
n	IJCX, IJCY, IJCZ	Source current density.	
n	INTEGRAL	Result of integration.	
n	J	Current density in a particle beam calculated by the	
		VIEW command.	
n	JCMOD	SQRT(JCX*JCX+JCY*JCY+JCZ*JCZ)	
n	JCR	(JCX*X+JCY*Y)/R	
n	JCT	(JCY*X-JCY*X)/R	
n	JCX0	SQRT(RJCX**2+IJCX**2)	
n	JCX	RJCX*COST+IJCX*SINT	
n	JCXP	ATAN2D(IJCX;RJCX)	
n	JCY0	SQRT(RJCY**2+IJCY**2)	
n	JCY	RJCY*COST+IJCY*SINT	
n	JCYP	ATAN2D(IJCY;RJCY)	
n	JCZ0	SQRT(RJCZ**2+IJCZ**2)	
n	JCZ	RJCZ*COST+IJCZ*SINT	
n	JCZP	ATAN2D(IJCZ;RJCZ)	

s*	LCS1_ANGLES	Current settings of CC	Current settings of CONDUCTOR parameters	
		THETA1, PHI1 and P	PSI1:	
		LCS1XYZ	0, 0, 0	
		LCS1YZX	90, 0, 90	
		LCS1ZXY	90, 90, 180	
		LCS10THER	any other angles	
s*	LCS2_ANGLES	Current settings of CC	NDUCTOR parameters	
		THETA2, PHI2 and P	·SI2:	
		LCS2XYZ	0, 0, 0	
			90, 0, 90	
		LCS2ZXY	90, 90, 180	
		LCS20THER	any other angles	
n	LENGU	Unit factor for length.		
n	Μ	Particle mass in electro	on units calculated by the	
		VIEW command.		
s*	MAP_VNNN	Are the MAP parameter	ers VX , VY and VZ the x, y	
		and z components of the	ne same vector?	
		ONE	same vector	
		THREE	different vectors	
n	MASSU	Unit factor for mass.		
n	MAXIMUM	Maximum COMPON and THREED.	and THREED.	
n	MINIMUM	Minimum COMPONE and THREED	Minimum COMPONENT value in MAP, PLOT and THREED	
n	MODELLER	2		
n	MUO	Permeability of free sp	Permeability of free space.	
n	NLENERGY	Energy in magnetic fie	eld calculated using non-lin-	
		mand		
n	NODE	Node number.		
S	NOW	Time in hh:mm:ss fo	ormat.	
n	NX, NY, NZ	Normal unit vector.		
n	PI	π		
n	P.I	Power density in a part	icle beam calculated by the	
		VIEW command.	tere beam calculated by the	
n	POSTPROCESSOR	3		
n	POWER	Power calculated by th	e ENERGY command	
n	POWEU	Unit factor for power.	Unit factor for power.	
n	PREPROCESSOR	1	1	
n	PROGRAM	3		

I

n	Q	Charge on a particle calculated by the VIEW com-
		mand.
	D	
n	ĸ	$SQRT(X^{**}Z+Y^{**}Z)$
n	RJCX, RJCY, RJCZ	Source current density.
n	ROTL11, ROTL33	Field point local coordinate system rotation
		matrix.
n	ROW	Row number in GRAPH command.
n	RSTART	Initial radial coordinate of trajectory set by the
		VIEW command.
n	SCALU	Unit factor of magnetic scalar potential.

n	SCALU	Unit factor of magnetic	e scalar potential.
s*	SET_ANGLES	Current settings of SE PLOCAL and SLOCA	T parameters TLOCAL, L:
		SETXYZ	0, 0, 0
		SETYZX	90, 0, 90
		SETZXY	90, 90, 180
		SETOTHER	any other angles
n	SINT	In AC problems, sinω	t

n	TH	ATAN2D(Y;X)	
s*	THREED_VNNN	Are the THREED parameters VX, VY and VZ the	
		x, y and z components of the same vector?	
		ONE	same vector
		THREE	different vectors
n	TOF	Time of flight of a parti	cle calculated by the VIEW
		command.	
n	TORQMOD	SQRT(TORQX*TOR ORQZ*TORQZ)	QX+TORQY*TORQY+T
n	TORQX, TORQY, TORQZ	Torque calculated by the BODY and INTE- GRATE commands.	
s*	TRACK_ANGLES	Current settings of TRACK parameters THETA,	
		PHI and PSI:	
		TRACKXYZ	0, 0, 0
		TRACKYZX	90, 0, 90
		TRACKZXY	90, 90, 180
		TRACKOTHER	any other angles
s*	TRACK_PARTICLE	Current settings of TR and CHARGES:	ACK parameters MASS
		ELECTRON	1, -1
		PROTON	1838.65, 1
		OTHER	any other values
n	TTIME	Transient time of a CA	RMEN or ELEKTRA-TR
		simulation.	
S	TODAY	Date in dd/mmm/yyy	y format.
n	TX, TY, TZ	Tangential unit vector.	
n	TXBEAM, TYBEAM, TZBEAM	Tangential unit vector to trajectory calculated by the VIEW command.	

n	VAL1, VAL2	Values from FILE1 an	d FILE2 in ARITHMETIC	
		command.		
n	VECTU	Unit factor for magnetic vector potential.		
n	VELMOD	SQRT(VELX*VELX+	VELY*VELY+VELZ*VEL	
		Z)		
n	VELR	(VELX*X+VELY*Y)/F	२	
n	VELT	(VELY*X-VELY*X)/R		
n	VELX, VELY, VELZ	Velocity.		
n	VERSION	Version number as a n	umber.	
S	VERSION	Version number as a cl	haracter string.	
s*	VIEW_INTE_AXES	Current settings of VIE	W parameters XAXIS and	
		YAXIS:		
		XINTE	X, DENSITY	
		YINTE	Y, DENSITY	
		RINTE	R, DENSITY	
		OINTE	any other values	
s*	VIEW_TRAC_AXES	Current settings of VIEW parameters XAXIS and		
		YAXIS:		
		XTRAC	Х, Ү	
		RTRAC	R, Z	
		OTRAC	any other values	
n	VOLUME	Volume of integration of	calculated by the ENERGY	
		and VOLUME comma	nds.	
n	X, Y, Z	Field point coordinates	3.	
n	X0BEAM, Y0BEAM, Z0BEAM	Coordinates at centre of	of beam calculated by the	
		VIEW command.		
n	XLOCAL, YLOCAL, ZLOCAL	Field point local coord	inate system origin.	
n	XSTART, YSTART, ZSTART	Coordinates of first poi	Coordinates of first point on a trajectory calculated	
		by the VIEW command	d.	
S	YESORNO	Pre-answer "yes-no" q	uestions with YES or NO.	

INDEX

Symbols

!	2-5
!!	2-16
#	
free format variables	2-18
user variables	2-22
\$ ABORT	2-16
\$ ASK	2-26
\$ ASSIGN	2-28
\$ BACKSPACE	2-29
\$ CD	
\$ CLOSE	2-27
\$ COMINPUT	2-25
\$ CONSTANT	2-21
\$ DO-loops	2-19
\$ ELIF	2-20
\$ ELSE	2-20
\$ END	2-21
\$ EULER	2-34
\$ FOR -each loops	2-20
\$ FORMAT	2-28
\$ IF	2-20
\$ OPEN	2-27
\$ OS	2-30
\$ PARAMETER	2-21
\$ PAUSE	2-26
\$ PROMPT	2-26
\$ READ	2-27
\$ SKIP	2-16
\$ STRING	2-21
\$ WHILE-loops	2-20
\$ WRITE	2-27

\$ -commands	
\$constant	
\$parameter 3-157	
\$VFDIR	
see VFDIR	
%COMPARE	
%ENV	
%EXPR	
%INT	
%REAL	
%VFDIR%	
see VFDIR	
, comma 2-6	
/, comment2-33	
;, semicolon 2-10	
l, command separator 2-33	
', quotation mark 2-11	
Numerics	
20-node brick	
3d Viewer menus 4-172	
8 node brick 3-42	
4	
-A-	
abbreviation	
abort	
DEFINE command .4-55, 4-59, 4-64,	
4-70	

1,0	
ACTIVATE command	5-22
adaptive integration metho	ds for RHS
4-80,	4-81
adaptive time-step	3-25

adding labels
air 3-102, 4-129
analysis 3-25, 3-136
frequencies3-66
ANALYSISDATA command3-23
anisotropic materials 4-15, 4-82
anisotropy 3-102, 4-129
arc
conductors4-36
ARC command
modeller3-27
post processor5-24
ARITHMETIC command5-26
arrow keys 4-3, 4-5
ASK
see <mark>\$ ASK</mark>
aspect ratio searching4-52
ASSIGN
see \$ ASSIGN
automatic potential cuts 3-26, 4-167

-B-

background	.3-124
BACKSPACE	

see \$ BACKSPACE

backup file	
base plane	4-50
subdivision editing	4-110
subdivision mode	4-67
bedstead	. 3-30, 3-62, 4-30
BEDSTEAD command	3-29
BEND command	3-31
BH curves3-102, 4-	15, 4-129, 4-162,
5-27	

BHDATA command

post processor	5-27
pre processor	4-15
BHDATA command	
bitmap	3-117, 5-76
BLEND command	3-35
block	3-36
BLOCK command	3-36
<i>bmp</i> file format	3-117, 5-76
BODY command	

boolean
operations
parameters
BOUNDARY command 3-37
boundary conditions 3-39
DEFINING 4-87
Definition Mode 4-86
EXTENDING the mesh 4-113
derivative4-88, 4-89, 4-93
displaying4-100, 4-171
functional 4-89
incident field 4-89
labels 4-156
Modification Mode 4-145
natural4-88, 4-89
non-zero total scalar potential 4-79,
4-81
normal field4-87, 4-93
perfect conductor 4-88
potential4-87, 4-88, 4-93
radiation4-87, 4-93
slip4-88, 4-89
symmetry 3-112, 4-88, 4-89, 4-93
tangential field 4-87, 4-93
vector potential 4-87, 4-88, 4-89, 4-93
voltage4-88, 4-89
boundary label
BOUNDARY sub-command
IDEAS command 4-119
brick
20 node
8 node 3-62
brick conductors4-37, 4-49
BRICK20 command 3-44
BRICK8 command 3-41
built-in commands 2-19

-C-

<i>c</i>	5-16
calculator	2-23
Capacitors	3-53
CARMEN	
CARTESIAN command	5-31
CASE sub-command	

SOLVERS command4-159
CASES 5-14, 5-23
CD
see \$ CD
CDBox
cell properties
CELLDATA command
chamfer 3-31, 3-35
character values2-10
character variables2-23
charge density 3-163, 4-81, 5-12
unit5-114
CHECK command
modeller3-49
pre processor4-18
CHECK sub-command
SOLVERS command4-160
checking3-49
CIRCLE command5-33
CLEAR command
modeller3-55
post processor5-35
pre processor4-21
clipboard 3-117, 5-76
CLOSE
see \$ CLOSE
coenergy5-48
coercive field 4-162, 5-12
coil field integrals3-25
COLOUR command
modeller3-56
post processor5-36
pre processor4-22
ColourBox 4-3, 4-8
colours3-56
COMBINE command
modeller3-58
post processor5-38
combining bodies3-58
COMINPUT
see \$ COMINPUT
••••••••••••••••••••••••••••••••••••••

command line interface	2-1
command separator	2-33
commands	2-4

built-in	2-19
post processor	5-3
pre processor	4-10
COMMENT command	3-60
comments	2-33
COMPONENT	5-10
conditional commands	2-20
conductivity 3-102, 4-129, 4-162,	5-14
unit	5-112
CONDUCTOR	
DEFINE sub-command	4-40
ERASE sub-command	4-43
MODIFY sub-command	4-44
PRINT sub-command	4-47
QUIT sub-command	4-47
WRITE sub-command	4-48
CONDUCTOR command	
modeller	3-61
post processor	5-40
pre processor	4-24
conductor file	5-44
conductor symmetries	
see symmetry	
conductors	
conductors	
arcs	4-36
bedsteads	4-30
bricks	4-37
constant perimeter ends 3-83, 3-	-145,
4-33	,
eddy currents	4-80
EXPORT command	3-73
forces	5-28
helical ends 3-86,	4-32
IMPORT command	3-90
in the mesh4-78, 4-79,	4-80
racetracks	4-29
single filament	4-25
solenoids	4-27
straight bars	4-35
cone	3-65
CONSTANT	
see \$ CONSTANT	

data3-95
data files
\$ COMINPUT 4-1, 5-2
<i>bh</i>
BH curves4-15
command input2-25
conductors3-73, 3-90, 4-48, 5-44
database4-151, 4-153, 4-154, 4-155,
5-18,
grid 5-18, 5-57
<i>igs</i>
<i>opc</i>
post processor5-18
pre processor 4-108, 4-111, 4-146,
4-177

sat
table 5-19, 5-52, 5-57, 5-73, 5-78,
5-101
time-tables 4-158
TRACK
universal 4-117
data storage level 3-47, 3-71, 3-74, 3-161
database
date2-24, 4-175
DBCASEDATA command 3-66
DE curves 4-15
DEBUG
DEFINE command 4-49
DEFINE sub-command
CONDUCTOR command 4-40
defining variables
delete
DELETE command
DENSITY
derivative boundary condition
see boundary conditions
derivative
DEVICE command 4-94
device nomination 4-1
DialogBox4-3, 4-7, 4-8
dialogue file
discretisation 4-49, 4-51, 4-60, 4-62
display 3-64, 3-88, 3-128, 3-159
colours
DISPLAY command 4-96
DO
see \$ DO
double-click
draft angle 3-142
DRIVE command
drive label 3-40, 4-25, 4-89, 4-156
DRIVE sub-command
SOLVERS command 4-160
drives 4-160
DUMP command 4-103
<i>г</i>
-E-

ε ₀	 5-16
eddy currents	 . 4-80, 4-152

ELEKTRA4-82
motion induced4-82
edge properties3-71
EDGEDATA command3-71
EDIT command4-108
eigen-values 4-152, 4-158
electric field strength5-12
unit 4-154, 5-112
electric flux density5-12
unit 4-154, 5-112
electric scalar potential5-12
unit 4-154, 5-114
electrostatics 4-81, 4-152
SCALA4-81
TOSCA4-81
ELEKTRA 1-2, 3-25, 4-82, 4-88, 4-152, 5-12
FI FKTR A-TR output times 3-66
element types $3_{-}77$ $3_{-}74$ $A_{-}155$
DEFINING 4-81
displaying A_{-100} A_{-171}
FI IF
see \$ FLIF
FLSE
see \$ FLSE
END command
modeller 3-72
not processor 5-46
pre processor 4-111
FND-DO
see \$ FND
END-FOR
see \$ END
END-IF
see \$ FND
see \$ FND
energy
unit 5-112
ENERGY command 5-47
environment variables 2-11 2-32 4-6
4-94
ERASE sub-command
CONDUCTOR command

Euler angles 2-34, 3-94, 3-153, 4-26,	
4-51,	
EXPORT command	
exporting conductors 3-73	
exporting data 3-125	
expressions2-8	
boundary conditions 4-89	
post processing5-10	
EXTEND command 4-112	
extending volumes 3-142	
EXTERNAL 4-18	
External circuits	
external fields 3-25, 4-167	
extrusion3-142, 4-51, 4-70	
direction 4-51	
EXTENDING the mesh	
extrusion direction 4-51	

-F-

face layering	3-74
FACEDATA command	3-74
Facet Definition Mode	4-61
EDITING	4-109
Facet Group Operations Mo	de 4-64
field calculation methods	5-89
field point coordinates	5-12
field values	
along a line	5-68
at a point	5-80
on a general surface	5-32. 5-81
field vectors	5-73, 5-124
filamentary conductors	
Filaments	3-53
file	
open	3-96
save	3-125
save as	3-125
FileBox	.4-3, 4-5, 4-8
filenames	
FILL command	
modeller	3-77
pre processor	4-114
FILTER command	3-80
FIT command	5-51

fitted constant perimeter end	d3-83
FITTEDCPE command	
fixed temperature	3-39
flux tubes	5-105, 5-110
FOR	

see **\$ FOR**

force	
integral	
unit	
FORMAT	

see \$ FORMAT

Fourier series fitting	5-52
free format input	2-18
frequency	3-66, 5-14
functional boundary condition	
see boundary conditions	
functional	

-G-

general morph	3-109
global coordinate system	3-165
GRAPH command	5-53
graphical output 3-117, 3-119,	5-76, 5-83
graphical user interface	2-1
graphs	5-78
GRID command	5-56
GUI	2-1

-H-

heat flux	3-39
heat transfer	3-39
helical conductors	4-32
helical end	3-86
HELICALEND command	3-85
HELP command	4-116
help character	2-5
hide	3-80, 3-88
HIDE command	3-88
hide menus	4-9
hiding objects	
high frequency	4-152
history	. 3-89, 3-156
HISTORY command	
homogeneity	5-17

horizontal menu		
hysteresis		5-48
	-/-	

IDEAS	
BOUNDARY sub-command	. 4-119
MATERIAL sub-command	. 4-121
QUIT sub-command	. 4-122
READ sub-command	. 4-122
IDEAS command	
post processor	5-60
pre processor	. 4-117
IF	

see \$IF

IMPORT command	3-90
importing conductors	3-90
inductors	3-53
inserting components	3-96
inserting OPERA-2d files	3-91
integral	5-15, 5-63
field calculations	5-90
line	5-79
Lorentz force	5-28, 5-48
Maxwell Stress	5-16, 5-65
power loss	5-48
stored energy	5-48
surface	5-74, 5-94
volume	5-48, 5-122
INTEGRATE command	5-65
interface	
reduced scalar potential	4-79,4-80,
4-81	
vector potential	4-80
intersecting bodies	
isotropy	3-102, 4-129
iteration data	3-26, 4-167

-J-

Jacobean	. 4-19
JE curves	. 4-15
joining bodies	. 3-58

-L-

LABEL command
modeller
pre processor4-123
labels
boundary conditions 4-88, 4-156
DEBUG4-18
displaying 4-99, 4-170
element types4-82
EXTERNAL4-18
material names4-82
potential types4-82
user defined4-123
laminations 4-15, 4-82
layering3-74
LCS
name3-124
LCS command
Legendre polynomial fitting5-52
length
unit 4-154, 5-112
limits1-4
LINE command5-68
linear elements 3-136, 4-81
linear extrusions 4-51, 4-71
linear motion4-152
linearity 3-102, 4-129
list
LIST command3-95
LOAD command
modeller3-96
post processor5-70
local coordinate system 3-31, 3-94, 3-165
anisotropic materials4-82
conductors4-26
DEFINE command (UVW)4-51
field calculations 5-79, 5-91
name3-124
permanent magnets4-82
post processing5-23
system variables5-14
lofting3-98
loops2-19

-M-

Μ5	5-119, 5-120
μ ₀	5-16
magnetic field strength	5-12
unit	4-154, 5-112
magnetic flux density	5-12
unit	4-154, 5-112
magnetic scalar potential	3-39. 5-12
unit	4-154, 5-112
magnetic vector potential	3-39.5-12
unit	4-154 5-112
magnetostatics	4-78 4-152
MAP command	5-72
material	
label 3-47 3	3-102 4-129
name A_78	1_{-155} 4_{-162}
NI II I	3_108
orientation	
properties 3 102 /	1 120 1 161
Material Definition Mode	129, 4-101 1 78
CADMEN	
	4-79, 4-80
ELENING the mach	4-79, 4-60
SODDANO	
TOSCA	
IUSCA	
MATERIAL sub-command o	TIDEAS
MATERIAL sub-command of	1 the SOLV
ERS command	
MATERIALS command	2 100
modeller	
pre processor	
maximum value	
Maxwell Stress	
integral	5-65
menu interface	
menu off	4-9
menus	4-3
merging faces	3-31, 3-35
MESH command	
modeller	3-104
pre processor	4-131

mesh control	3-47, 3-74
mesh size	3-71, 3-161
MessageBox 2-25, 2-26, 2-	-33, 4-3, 4-7
minimum value	5-15
MODEL command	3-107
MODIFY command	4-133
MODIFY sub-command	
0.011010700	

CONDUCTOR comm	and4-44
MORPH command	3-109
morphing 3-3	1, 3-109, 3-140
MOUSE command	
post processor	3-110, 5-75
	4.93

moving parts	
multiply connected	
multipole moments	
multi-valued potential	4-79, 4-80

-N-

name3-124
negative periodicity3-113
NODALLY averaged fields5-90
non-linear 3-102, 4-129
non-linear analysis 3-25, 4-166
non-linear analysis options
normal electric
normal field boundary condition
1
see boundary conditions
normal field
normal field sources and sourc
normal magnetic
see boundary conditions normal field normal magnetic
see boundary conditions normal field normal magnetic
see boundary conditions normal field normal magnetic
see boundary conditions normal field normal magnetic

-0-

OPEN

see \$ OPEN

OPERA-3d	1-1
Opera3d_Modeller_n.log .	2-3
Opera3d_Modeller_n.lp	2-3
Opera3d_Post_n.log	2-3
Opera3d_Post_n.lp	2-3, 5-120
Opera3d_Pre_n.log	2-3

Opera3d_Pre_n.lp	
operaanl	
operating system commands.	2-30
orientation	3-163
OS	

see \$OS

output times		3-66
--------------	--	------

-P-

π	5-16
packing factor	
page number	4-175
PARAMETER	

see **\$ PARAMETER**

ParameterBox4-3, 4-4, 4-8
parameters 2-4
particle trajectories 5-117, 5-124
paste
PAUSE
see \$ PAUSE
PEC
ELEKTRA 4-88
SOPRANO 4-88
perfect conductor boundary condition
see boundary conditions
perfect conductor
perfect insulator 3-39
periodic boundary 3-111
periodicity
periodicity boundary condition
see boundary conditions
symmetry
PERIODICITY command 3-111
PERIODICITY sub-command
SOLVERS command 4-163
permanent magnets 4-82
permeability 3-102, 4-129, 4-162, 5-14
permittivity 3-102, 4-129, 4-162, 5-14
phase angle of conductors 4-25
phase lag3-102, 4-129
pick 3-80
PICK command 3-114
pick options 3-80
picking 4-4

picking filter3-80
PICTURE command
modeller3-117
post processor5-76
planar face
PLOT command5-77
<i>png</i> file format
POINT command
Point Definition Mode4-55
EDITING
Point Modification Mode4-134
POLAR command5-81
positional assignments2-6
positive periodicity
potential boundary condition
see boundary conditions
potential
potential cut
potential types 3-47, 4-155
DEFINING4-78
displaying 4-100, 4-171
POTENTIAL_CUTN
potentials
multi-valued 4-79, 4-80
potential cut 4-79, 4-81
reduced scalar
single valued 4-79, 4-80
total scalar4-79, 4-80, 4-81
vector 4-80, 4-81
power
density5-15
unit5-112
power loss
integral5-48
PRECISIONDATA command3-118
PRINT command
modeller3-119
post processor5-83
PRINT sub-command
CONDUCTOR command4-47
printing 3-119, 5-83
prism3-120
PRISM command3-120
program sizes1-4

PROMPT see \$ PROMPT

prompting	
for parameter values	
in command input files	2-26
properties	3-95
pyramid	3-120

-Q-

Q	.5-119, 5-120
quadratic elements	3-136, 4-81
quadratic extrusions	4-51, 4-71
QUIT sub-command	
CONDUCTOR comman	nd 4-47
IDEAS command	4-122
SOLVERS command	4-164

-R-

R, Z
racetrack
RACETRACK command 3-121
radiation 3-39
radiation boundary condition
see boundary conditions
radiation
radio-buttons 4-7
READ
see \$ READ
READ command 4-146
READ sub-command
IDEAS command 4-122
redo 3-156
REDO command 3-123
reduced scalar potential .4-78, 4-79, 4-80
interface4-79, 4-80, 4-81
reflect 3-153
regularise 3-67
relaxation factor 3-24
removing objects 3-67
RENAME command 3-124
Resistors 3-53
RHS integrals
rotate
rotational motion 4-152

rotational velocity	
RSTART	
rubber-box	
Runge-Kutta	3-26, 4-167
updates	
r	

-S-

SAVE command3-125
save picture 3-117, 5-76
saving data
SCALA 3-25, 4-81, 4-88, 4-152, 5-12
scalar potential boundary condition
see boundary conditions
potential
scalar property 4-81, 4-82
scale
scroll bars 4-5, 4-8
SELECT command
modeller3-127
post processor5-84
selecting4-3
selecting parts for display3-128
SET command5-88
SETTINGS sub-command of the SOLV-
ERS command4-165
SHOW command5-92
simple updates3-25
simulation title3-60
simulations4-153
single filament conductors4-25
single valued potential 4-79, 4-80
slider bars4-8
SLIP command4-149
slip surface 4-89, 4-149
solenoid3-62, 3-133, 4-27
SOLENOID command3-132
SOLVERS command
modeller3-135
pre processor4-150
SOPRANO 1-2, 3-25, 4-87, 4-152, 5-12
source current density5-12
source magnetic field strength5-12
space charge beam4-152
sphere3-137

-*T*-

TABLE command
post processor 5-101
pre processor 4-168
tangential constant perimeter end . 3-145
tangential electric 3-39
tangential field boundary condition
see boundary conditions
tangential field
tangential magnetic 3-39
tangential magnetic

thermal conductivity3-102
THREED command
post processor 3-147, 5-104
pre processor4-169
time
time of flight 5-119, 5-120
time stepping 3-26, 4-167
timers4-8
time-step3-25
time-table files4-158
TITLE command
post processor 3-149, 5-104
pre processor4-174
titles3-149, 4-156, 4-174, 5-107
TODAY 2-24
TOF 5-119, 5-120
tolerance
conductor fields4-25
group operations4-65
topology problems3-49
torque5-15, 5-16, 5-28, 5-65
torus3-151
TORUS command3-151
TOSCA 1-2, 3-25, 4-88, 4-152, 5-12
total scalar potential 4-79, 4-80, 4-81
TRACK command5-109
TRANSFORM command
modeller3-152
pre processor4-176
transformations 3-109, 3-111, 3-153
transforming base plane coordinates
4-64
transient
translate
IVVISI command
IXBEAM, IYBEAM, IZBEAM5-119

-U-

3-156
3-156
3-58
3-136
4-154
4-16

post processing	
UNITS command	5-112
units display	4-175
upwinding	.3-25, 3-26, 4-167
user interface	
user variables	

-V-

VARIABLE command 3-157
variables
character
in expressions2-8, 2-9
in free format input 2-18
numerical 2-22
system5-10, 5-97
VECTOR command 3-159
vector maps5-73, 5-124
vector potential 4-80, 4-81
vector potential boundary condition
see boundary conditions
vector potential
vector potential interface 4-80
vector property
velocity 3-163, 4-82, 4-85, 5-12
VELX
VELY
VELZ 5-119, 5-120
version number 2-24
VERTEXDATA command 3-161
vertical menu 4-3
VFDIR 2-11, 2-32, 4-7, 4-15
VFGRAPHICS 4-94
VFINV 4-95
VFWINDOWH 4-95
VFWINDOWW 4-95
VIEW command 5-115
voltage 3-39
voltage boundary condition
see boundary conditions
voltage
volume5-15, 5-48
label 3-163
properties
VOLUME command

modeller	
post processor	
VX, VY and VZ	

-W-

WCS
WCS command3-165
WHILE
see \$ WHILE
Windings3-53
WINDOW command
modeller3-166
post processor5-124
working coordinate system 3-94, 3-165
WRITE
see \$ WRITE
WRITE command4-177
WRITE sub-command
CONDUCTOR command4-48

-X-

Х, Ү	5-118, 5-119	, 5-120
X0BEAM, Y0BEA	M, ZOBEAM	5-119
xpm file format	3-11	7, 5-76
XSTART		5-119

-Y-

YESORNO	2-24
YSTART	5-119