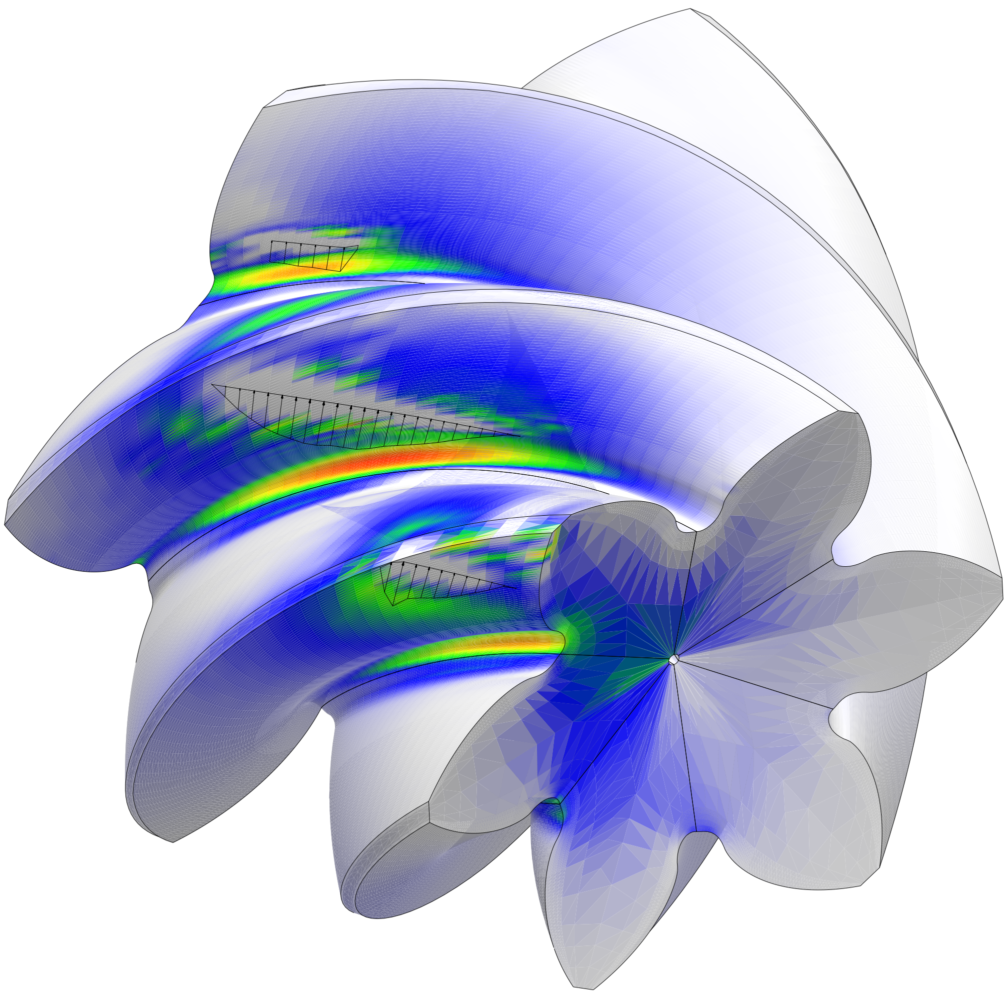
HypoidFaceHobbed User's Manual



Advanced Numerical Solutions
Hilliard OH

May 12, 2005

Contents

List of Figures

List of Tables

Preface

In developing the HypoidFaceHobbed computer program, we have received active support and encouragement from many people. We would especially like to thank Timothy Krantz of the Army Research Laboratory at the NASA Glenn Research Center for his support and encouragement.

Sandeep Vijayakar, Hilliard OH
April 2003

Chapter 1

Introduction

In some applications gears are needed to connect shafts which are neither parallel nor intersecting. For this purpose a variation of the spiral bevel gear, called a *hypoid gear*, has been developed. The unusual geometry of the *hypoid gear* allows the pinion to be large and strong even though it has only a few teeth.

Hypoid gears resemble bevel gears in some aspects. They are used on crossed-axis shafts, and there is a tendency to parts to taper as do bevel gears. They differ from true bevel gears in that their axes do not intersect. The distance between a hypoid pinion axis and the axis of a hypoid gear is called the *offset*. Figure 1.1 shows offset and other terms.

Hypoid pinions may have as few as five teeth in a high gear ratio. Since the various types of bevel gears do not often go below 10 teeth in a pinion, it can be seen that it is easy to get high ratios with *hypoid gears*. They do not have pitch diameters which are in proportion to their numbers of teeth. This makes it possible to use a large and strong pinion even with a high ratio and only a few pinion teeth. They are used in various applications such as passenger cars, industrial drives, tractors, trucks etc.

Several manufacturing processes are available for hypoid gears. The *HypoidFaceHobbed* package is meant for analyzing hypoid gears cut using the face hobbing process.

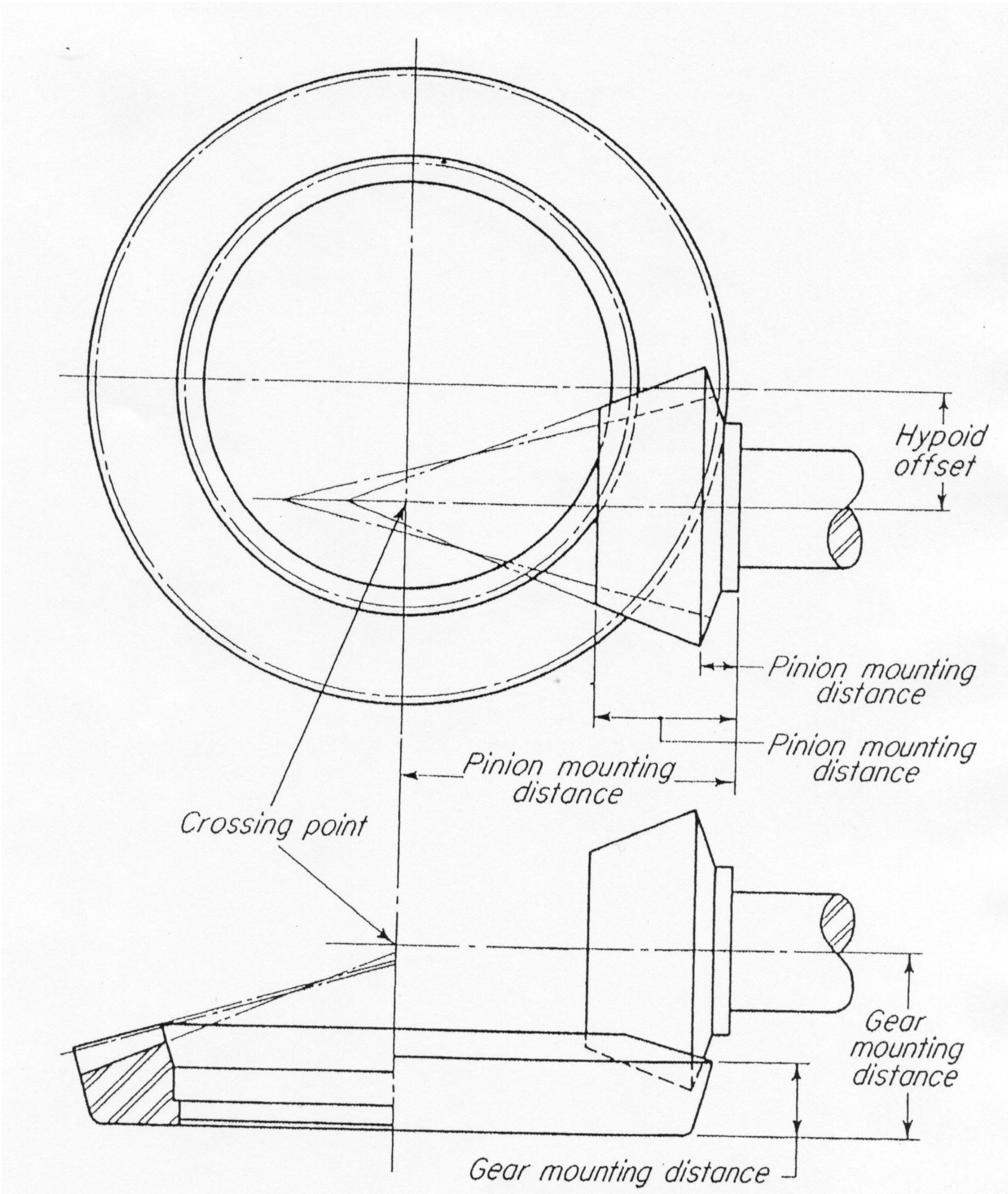


Figure 1.1: The hypoid gear arrangement

Chapter 2

HypoidFaceHobbed Software Package

This chapter explains the various features of the *HypoidFaceHobbed* software package.

2.1 HypoidFaceHobbed analysis package

Calyx is a powerful contact analysis code capable of analyzing a variety of contact problems, including 2D and 3D dynamic and static analysis of systems such as gears, compressors, and brakes. Because *Calyx* has to be capable of handling a variety of problems, it communicates with the outside world through a programming language. The programming language interface of *Calyx* brings flexibility at the expense of ease of use. Such an interaction is appropriate for an advanced *Calyx* user, but not for a gear design engineer.

In order to address this issue, the program *Multyx* is used. *Multyx* is capable of communicating with the user through an easy to use menu-based interface. It translates the user's commands into the appropriate programming language statements and sends them on to *Calyx*. A typical user does not even need to know that *Calyx* is running in the background.

In addition to the user interface, Multyx also has built-in model generators. The hypoid gear tooth models, described in this manual are all generated by *Multyx*. It also has post-processing and data extraction code, to help the user extract the results of analysis from *Calyx*.

Multyx and *Calyx* are designed as portable code, and can run on any system that supports standard C++. In order to keep it portable, *Multyx*'s menu system is command line based, and does not use any of the GUI features such as buttons, windows or mouse interaction. The following dialog shows some of the command line interface of *Multyx*.

```
E:>multyx
MultyX v.1.06, Copyright Advanced Numerical Solutions Dec 21 2000
MultyX>post ok patt
MultyX.PostProc.1/11.Pattern>HELP
MENU            Show menu
?               Show menu
HELP            Show menu
EXIT            Return to main menu.
QUIT            Return to main menu.
START           Draw the contact pattern.
CLEAR           Clear the graphics page.
SURFACEPAIR     Surface pair (Currently=GEAR_SURFACE1_PINION_SURFACE1)
```

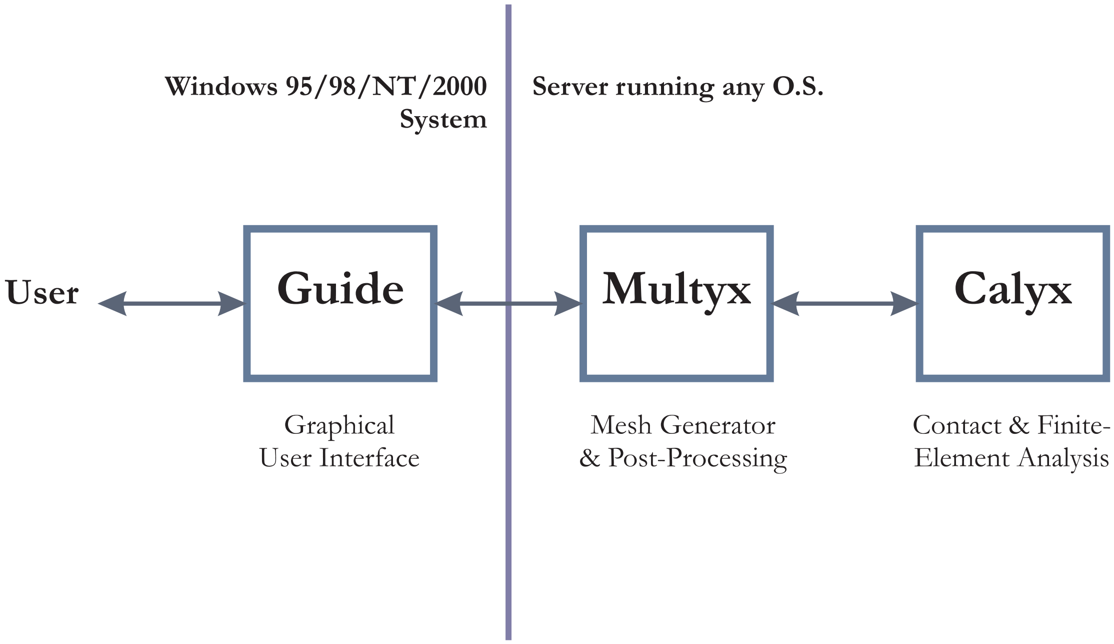


Figure 2.1: The computer programs in the *HypoidFaceHobbed* analysis package

```
MEMBER          Member (Currently=PINION)
TOOTHBEGIN      12              Tooth no. or instance no. of surface.
TOOTHEND        2               Tooth no. or instance no. of surface.
BEGINSTEP       1               Time/Roll angle step at which to begin search.
ENDSTEP         10              Time/Roll angle step at which to end search.
COLORS          Whether to render the model in color (Enabled)
CONTOURS        Whether to draw pressure contours (Enabled)
MINPRESS        4.000000E+004   Level of lowest press. contour.
MAXPRESS        4.200000E+005   Level of highest press. contour.
DELTAPRESS      4.000000E+004   Spacing between press. contours.
SMOOTH          (TRUE)         Whether to smooth the pressure contours.
OUTPUTTOFILE    Whether to write data to file. (Disabled)
MultyX.PostProc.1/20.Pattern>START
```

Guide is a program that provides a Graphical User Interface (GUI) to *Multyx*. *Guide* translates each of *Multyx*'s dialogs and presents them to the user in a graphical form. The command line menu described above is presented to the user as shown in Figure 2.2.

In addition, *Guide* provide the user with convenient ways of viewing the graphics, and helps the user convert the graphics into Microsoft formats and into Encapsulated PostScript (EPS) files.

Although *Guide* enhances the friendliness of *Multyx*, it is not required. All the features of *Multyx* can be accessed without *Guide*. The connection between *Guide* and Multyx is based on the TCP/IP telnet protocol when they are running on different computers. When running on the same computer, they communicate through named pipes. *Guide* is a heavy user of advanced operating system features including GUI support, multi-threading support, and inter-process communication support. *Guide* now runs on Windows 95/98/NT/2000 systems only.

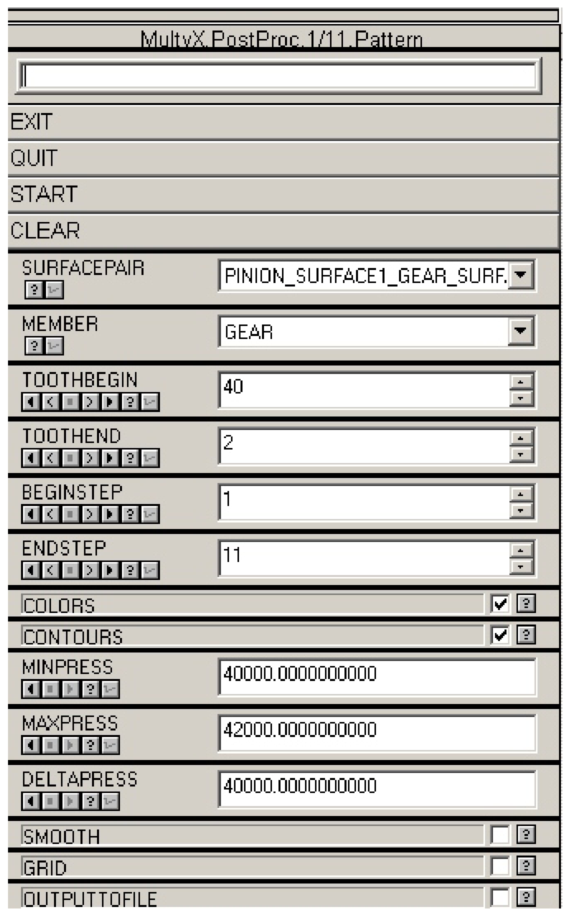


Figure 2.2: The menu presented to the user by Guide

2.2 Installation of the software package on windows platform

The procedure for installing the *HypoidFaceHobbed* software analysis package on Microsoft Windows NT/2000/XP platforms is as follows:

- Obtain the self-extracting file *HypoidFaceHobbed.EXE* from us either by e-mail, ftp or on a CD. If you are using an ftp client, make sure that the *HypoidFaceHobbed.EXE* file is downloaded when ftp is in binary mode.

- Before installing anything make sure that there are no previous copies of *guide.exe*, *multyx.exe* or *calyx.exe* in your *path*. If these files are present then you either have to move the old programs elsewhere or change the *path* so that they don't conflict with the new programs.

- Now you are all set to install the software package. Close all the other programs and run *HypoidFaceHobbed.EXE*. It will ask you questions about where to install the program and where to keep the working directory.

- After you answer these questions, it will display the *Computer-ID* and ask for a *licence key*. Copy the *Computer-ID* and click the button -*skip(or install key later)*. It will proceed with the installation and will install 3 icons under *Start/Programs/HypoidFaceHobbed*.

- Send the *Computer-ID* via email to support@ansol.com, and we will send you the *Licence Key*. Click on the *skip* button. You can return to this license key dialog by using the icon at *Start/Programs/HypoidFaceHobbed/Register*.

- After you receive the *Licence Key* from Ansol run- *Start/Programs/ HypoidFaceHobbed/Register* again and paste the *Licence Key* in the respective box. Now, click on the button-*Install Licence Key*.

- Now you are all set to run the analysis. Start the program by using the icon *Start/Programs /HypoidFaceHobbed/HypoidFaceHobbed*.

Chapter 3

Preliminaries

The previous chapter gave an overview of the software architecture. This chapter provides some information to help you get up and running with the program.

3.1 System of units

Any system of units can be used provided that all the inputs provided by the user are consistent with this system of units. The user is free to choose any units for force, time and length. All the inputs should then be in units that are consistent with this choice. For example, if the user chooses Kgf as the unit for force, seconds as the unit for time, and cm as the unit for length, then the input torque should be in $Kgf.cm$, the Youngs modulus in Kgf/cm^2, the Diametral pitch in $1/cm$ and the mass density in $Kgf.s^2/cm^4$. Outputs will also appear in consistent units.

3.2 Bodies

In multi-body contact analysis, the term 'body' is used to refer to an object that is capable of rigid body motion, and interacts with other bodies through surface contact and bearing connections (Figure 3.1).

There is a special body called the 'fixed body' which refers to ground.

In *HypoidFaceHobbed*, the hypoid pinion and hypoid gear are treated as separate bodies. The interaction between the hypoid gear and the hypoid pinion is through contact.

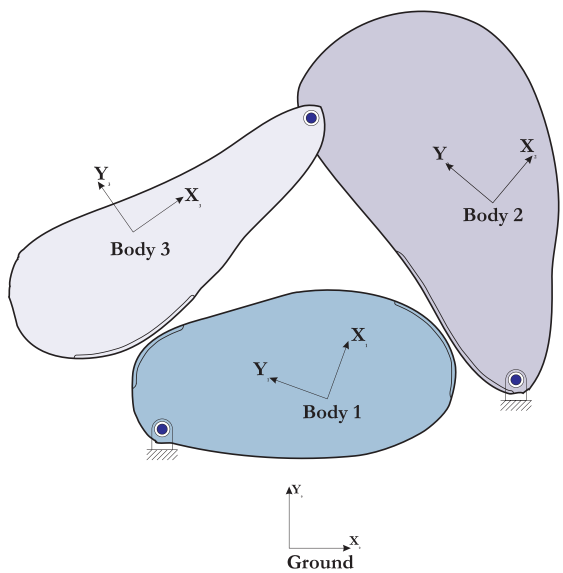


Figure 3.1: A multi-body system

3.3 Reference frames

Each of the bodies in the system has a reference frame to which it is rigidly attached. The reference frame has 6 rigid body type degrees of freedom, three translation components U_x, U_y and U_z, and three rotation components θ_x, $\theta_y$and and θ_z (Figure 3.2).

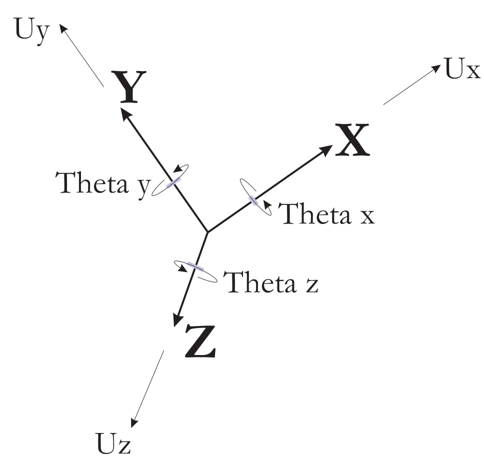


Figure 3.2: Reference frame degrees of freedom

In addition to the body reference frames, there is a special reference frame called the fixed reference frame that is considered as 'ground', and does not move. It is used as the reference for defining the locations of all other reference frames.

Figure 3.3 show how *HypoidFaceHobbed* sets up the pinion and gear reference frames relative to the fixed reference frame in Hypoid gear set. Its Z axis is parallel to the axes of rotation of the gear. The pinion and gear reference frames have their origins at their crossing points, with the Z axis being the axis or rotation. At time $t = 0$, the gear X, Y and Z axes are parallel to the corresponding axes of the fixed reference frame.

Manufacturing and assembly errors applied to the system might perturb the location of these reference frames slightly from their nominal location.

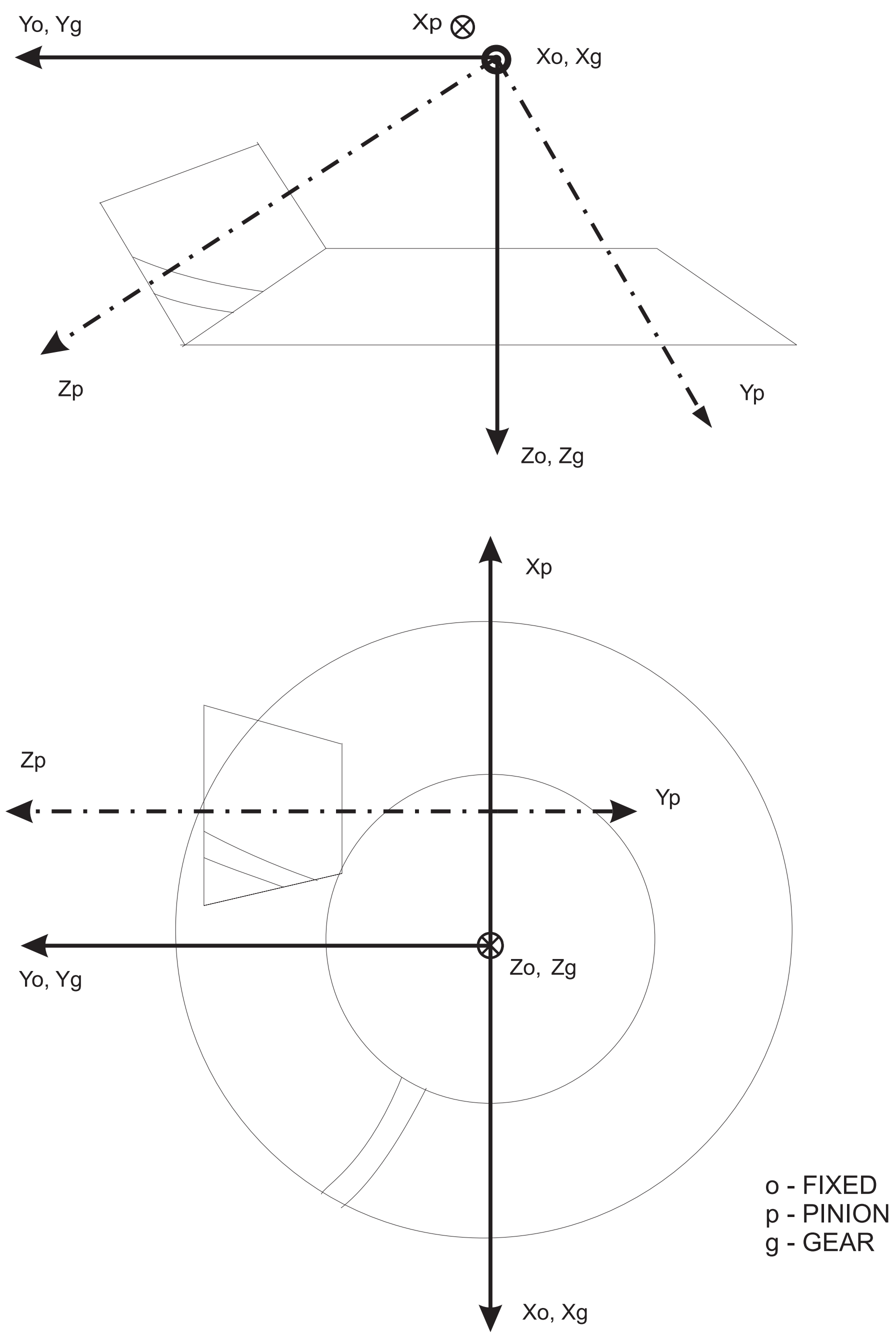


Figure 3.3: The reference frames set up for a pair of HypoidFaceHobbed gears

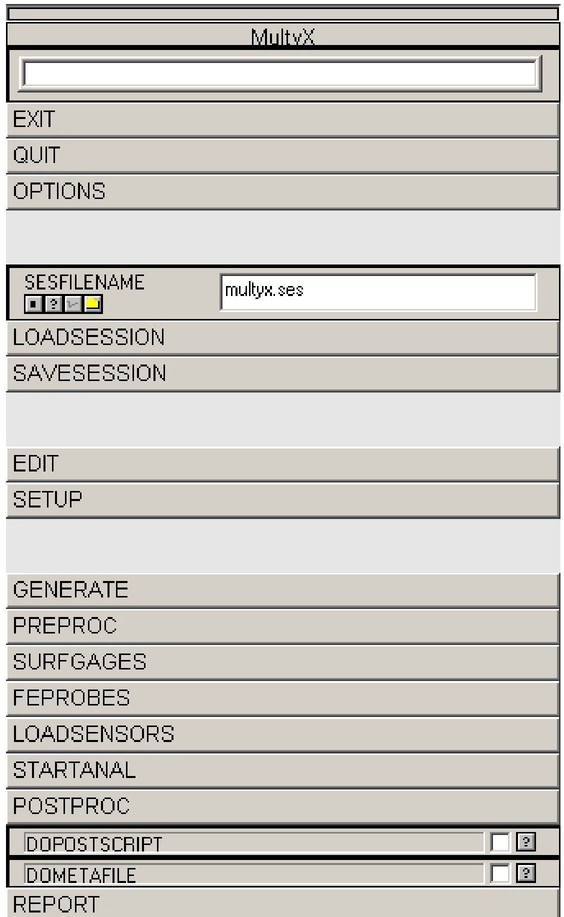


Figure 3.4: The main menu.

3.4 The main menu

The *HypoidFaceHobbed* package is started by clicking on an icon created during the installation process. After the *HypoidFaceHobbed* package is started, the main menu shown in Figure 3.4 comes up.

All user provided data is saved in a file called the session file. The name of this session file can be changed by typing the name in the SESFILENAME box. Changing the file name does not actually write the data to the new file, nor does it read data from the new file. Data is written to the session file through the SAVESESSION command. Data can be loaded from an existing session file using the LOADSESSION command.

The QUIT command terminates the program without saving any data in the session file. The EXIT command first writes data to the session file, and then terminates the program.

All data entry occurs in a hierarchy of submenus accessed through the EDIT command on this main menu.

After data entry is completed, the GENERATE command may be used to generate the model. At this point, a consistency check is carried out. If any errors or inconsistencies are detected in the user's inputs, then error messages are displayed, and the model is not generated. If the program detects something that it thinks is questionable, but is still able to proceed, then it displays warning messages, but proceeds with generating the model.

The REPORT command is used to generate an ASCII file called `report.txt`, describing all the inputs the user has supplied to the program.

The PREPROC command allows the user to graphically inspect the latest model. If the user has changed some parameters after the last GENERATE action, then the PREPROC command detects this and calls the GENERATE command itself.

The SETUP command is used to set up an analysis, and the FEPROBES, SURFGAGES, and LOADSENSORS commands are to control the data created by the analysis.

The POSTPROC command is used to graphically inspect the results of the analysis.

Chapter 4

The Graphical User Interface

HypoidFaceHobbed's user interface is presented by *Guide* in graphical form, as shown in Figure 4.1. *HypoidFaceHobbed* also sends out a stream of informational, error and warning messages to the user. These messages are separated by *Guide*, and presented in separate windows as shown. The user activates these message windows by hitting the appropriate "Error", "Information" or "Warning" tab. Graphical information sent by *HypoidFaceHobbed* is directed to a graphics window.

4.1 Menu command items

In the example shown in Figure 4.1, the large buttons such as those labeled EXIT ,QUIT, OPTIONS, LOADSESSION, EDIT send commands to *HypoidFaceHobbed* when hit by the user. In response to the command, *HypoidFaceHobbed* might carry out an action, as in the case of the LOADSESSION command, or lead the user to a different menu, as in the case of the EDIT command. Moving the mouse over a button without depressing it will cause *Guide* to momentarily pop up a balloon (a tool tip) containing a short description of the use of that button. The tool tips can be disabled by the View|DisableToolTips item in the *Guide* main menu.

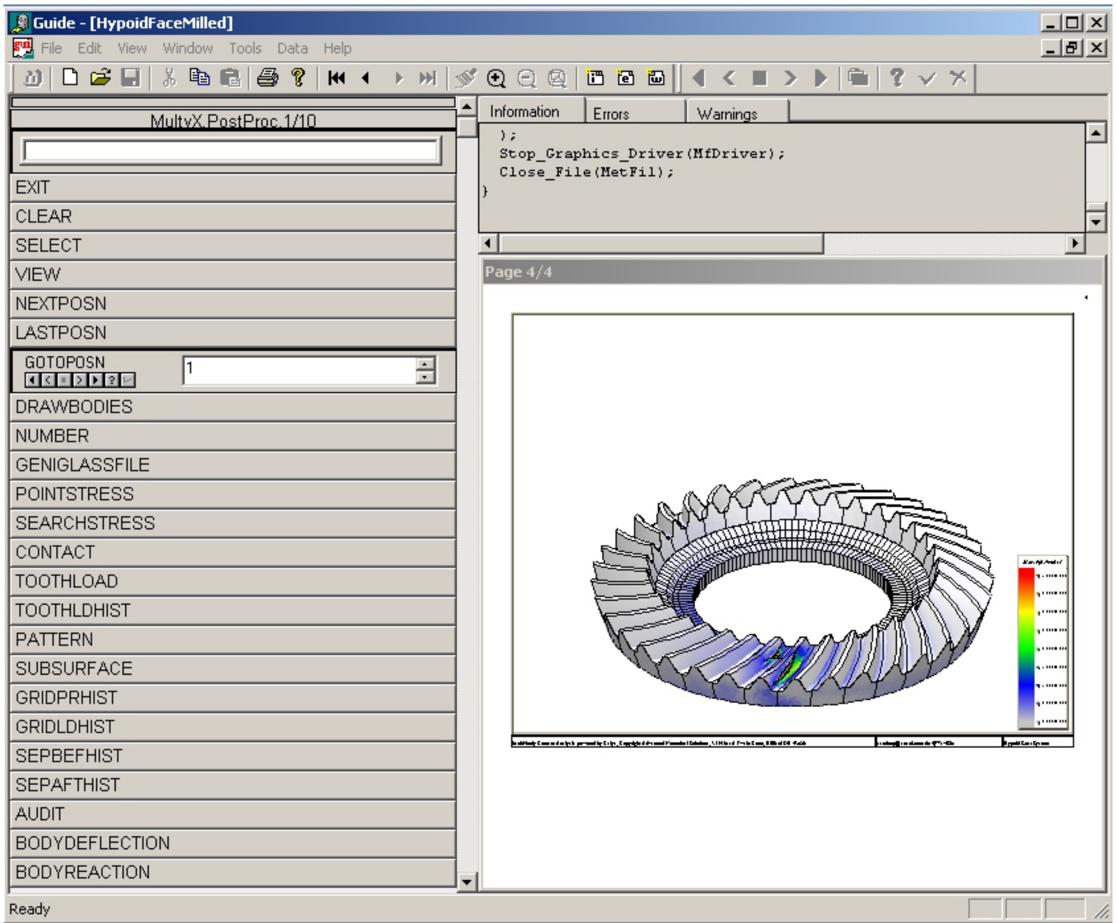


Figure 4.1: *HypoidFaceHobbed*'s user interface.

4.2 Integer menu items

Integer data items are entered through a dialog box of the kind shown in Figure 4.2. The current value appears in a box in the dialog box. If the value of the data item is undefined, then the box appears blank.

Figure 4.2: An integer data entry box

4.3 Floating point menu items

Floating point data is entered through the dialog box shown in Figure 4.3.

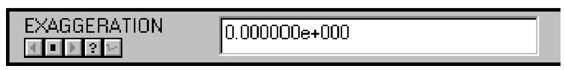


Figure 4.3: An floating point data entry box

4.4 Boolean menu items

Boolean data items are those that can only take a YES/NO or TRUE/FALSE type of value. Their value is set by checking or clearing the box as shown in Figure 4.4.

Figure 4.4: An boolean data entry box

4.5 String menu items

String data items contain ASCII strings. The dialog box shown in Figure 4.5 allows the user to enter string type data.

Figure 4.5: An string data entry box

4.6 Switch type menu items

The last kind of data item is of the ‘switch’ type. This item can be switched between a fixed set of valid choices. The choice is made through a drop down list as shown in Figure 4.6.

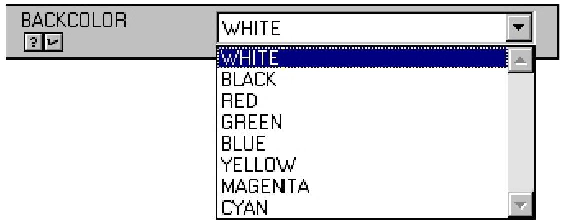


Figure 4.6: An switch type data entry box

4.7 Commonly occurring buttons

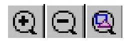
The data entry dialog boxes use a few small buttons as short cuts for common tasks as shown in the Table 4.1. Some of these buttons may be disabled depending upon the particular item and its value.

Table 4.1: Common buttons

Button	Purpose
	Select the minimum allowable value
	Decrement the value by 1
	Select the default value
	Increment the value by 1
	Select the maximum allowable value
	Accept the value just typed in
	Discard the value just typed in
	Get additional information
	Change the current graphics page
	Change the zoom level
	Refresh the graphics page

4.8 Graphics

Guide directs the graphical output from *HypoidFaceHobbed* to a graphics window. The graphics are stored as separate pages. A new page is started when *HypoidFaceHobbed* clears the graphics screen. The user can move between screens using the  buttons on the toolbar.

Double clicking anywhere in the graphics window with the left mouse button or dragging the mouse in the graphics window with the left button depressed lets you zoom in. To zoom out, double-click with the right mouse button. The  buttons on the toolbar can also be used to zoom in, zoom out and to return to the original view.

By default, the graphics are refreshed automatically when necessary. However, this behaviour can be undesirable if the graphics are very complex. This auto-refresh behavior can be toggled using the View|EnableAutoRefresh and View|DisableAutoRefresh commands. If auto-refresh is disabled, then the user can ask refresh the graphics using the  button.

It is possible to save a sequence of graphics pages in a metafile (a .MET file) using the File|SaveReplayFile command. This file can later be replayed in *Guide* using the File|ReplayGraphicsFile command.

The graphics currently displayed can be saved in Windows Metafile format (a .WMF file) by using the File|SaveWindowsMetafile command. This .WMF file can subsequently be loaded by another application such a word processor. An encapsulated PostScript file (a .EPS file) can be created by using the File|CreateEPSFile command. This command creates an .EPS file containing only the visible part of the current graphics page. Parts of the page that are not visible because of the zoom level will be cropped from the .EPS file.

The Edit|Copy command will copy the graphics in Windows Metafile format onto the clipboard.

Graphics pages can be printed by using the File|Print command on *Guide*'s main menu.

Chapter 5

Building a Model

All data describing the model is entered in sub-menus of the EDIT menu. Figure 5.1 shows the EDIT menu. In this EDIT menu, and in all sub-menus under it, the QUIT command takes the user back to the parent menu after discarding all changes made in the sub-menu and all sub-menus under it. The EXIT command takes the user back to the parent without discarding changes.

There are three sub-menus under this EDIT menu. The SYSTEM command leads to a menu for entering system level data. The PINION and GEAR commands lead to separate sub-menus for entering data specific to the pinion and gear, respectively.

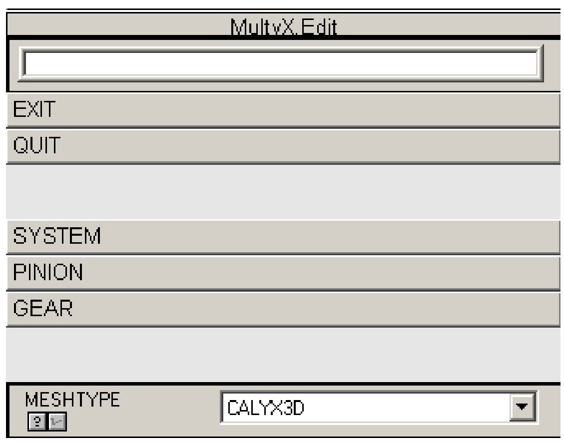


Figure 5.1: The EDIT menu.

5.1 Special Analysis File

If the hypoid gear is manufactured using the Gleason's manufacturing process then the finite element analyst is given a special analysis file by the gear manufacturer. This file contains the design data used for the manufacturing of the hypoid gear. An example of a special analysis file record is shown in Appendix B. This file contains many such records. We will need records 1 through 58 for analysis.

Table 5.1: System configuration parameters

Item	Description	Special Analyis File Details
MODELTYPE	Switch, Type of mesh to generate (CALYX3D/CAPP)	
CONFIGFILE	String, Configuration file for CAPP (if MODELTYPE=CAPP)	
HANDPINION	Switch, Hand of the pinion (Lefthanded/Righthanded)	Record#6,Item#8
OFFSET	Float, Shaft offset distance	Record# 1, Item# 7
ANGLE	Float, Shaft angle (Deg)	Record# 1, Item# 4
LOADEDSIDE	Switch, Side of the gear to be loaded(Convex/Concave)	
DRIVER	Switch, Member that is driving (Pinion/Gear)	
MU	Float, The coefficient of Coulomb friction	
TORQUE	Float, The pinion torque magnitude (always positive)	
RPM	Float, Pinion angular velocity in RPM	
DOASSEMBLYERRORS	Boolean, Whether or not to include assembly errors	

5.2 System level data

The SYSTEM command in the EDIT menu of Figure 5.1 leads to the SYSTEM menu shown in Figure 5.2. The parameters in this menu are summarized in Table 5.1.

The kind of mesh the mesh generator will create is controlled by the MODELTYPE switch. The choices are CAPP type and CALYX3D type. If the MODELTYPE option in the main menu is set to CAPP then the name of the configuration file can be specified in the item CONFIGFILE. This file is created by the program. CAPP is an older analysis and post-processing package. You are only able to generate the model if you select the CAPP type model. The preproc, postproc, setup, startanal, probes, loadsensors, surfgages menus cannot be accessed for the CAPP type model.

The HANDPINION switch controls whether the pinion is lefthanded or righthanded. If the Special Analysis Record value for this item is 1 then the pinion is lefthanded and if the value is 2 the pinion is righthanded.The shaft offset value is specified in the OFFSET menu. Offset is the perpendicular distance between the axes of the hypoid gear and the pinion. Refer to Figures 5.3 and 5.4 for the sign convention for the offset menu. The shaft ANGLE is the angle between the axes of the gear and the pinion. It is measured in degrees. The Special Analysis File (Record and Item numbers specified in Table 5.1) has value for this angle in radians. Conversion to degrees is done before specifying it in the ANGLE menu.

The LOADEDSIDE switch controls whether the convex side or the concave side of the gear tooth is going to carry the load. The DRIVER switch defines which member is the input member. Power will enter the gear pair at this member. The angular velocity and the torque for the pinion are entered in the RPM and TORQUE items respectively.

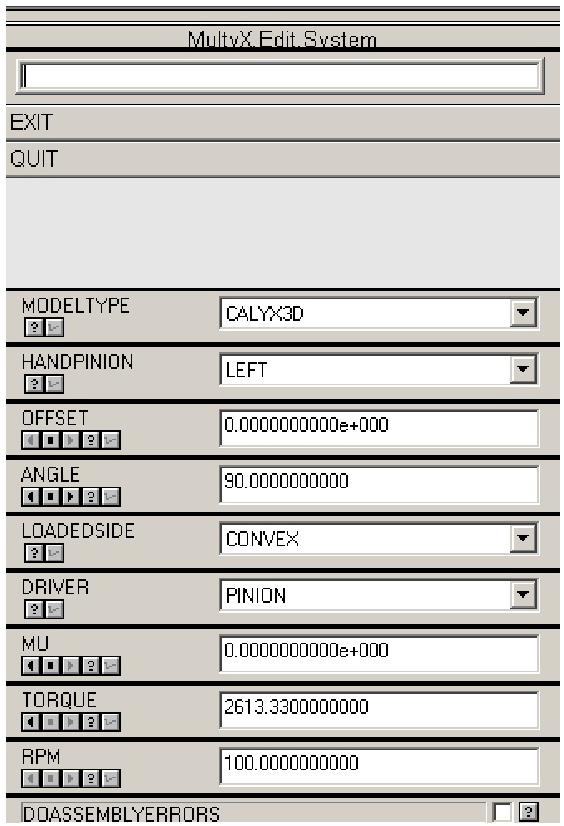


Figure 5.2: The system data menu.

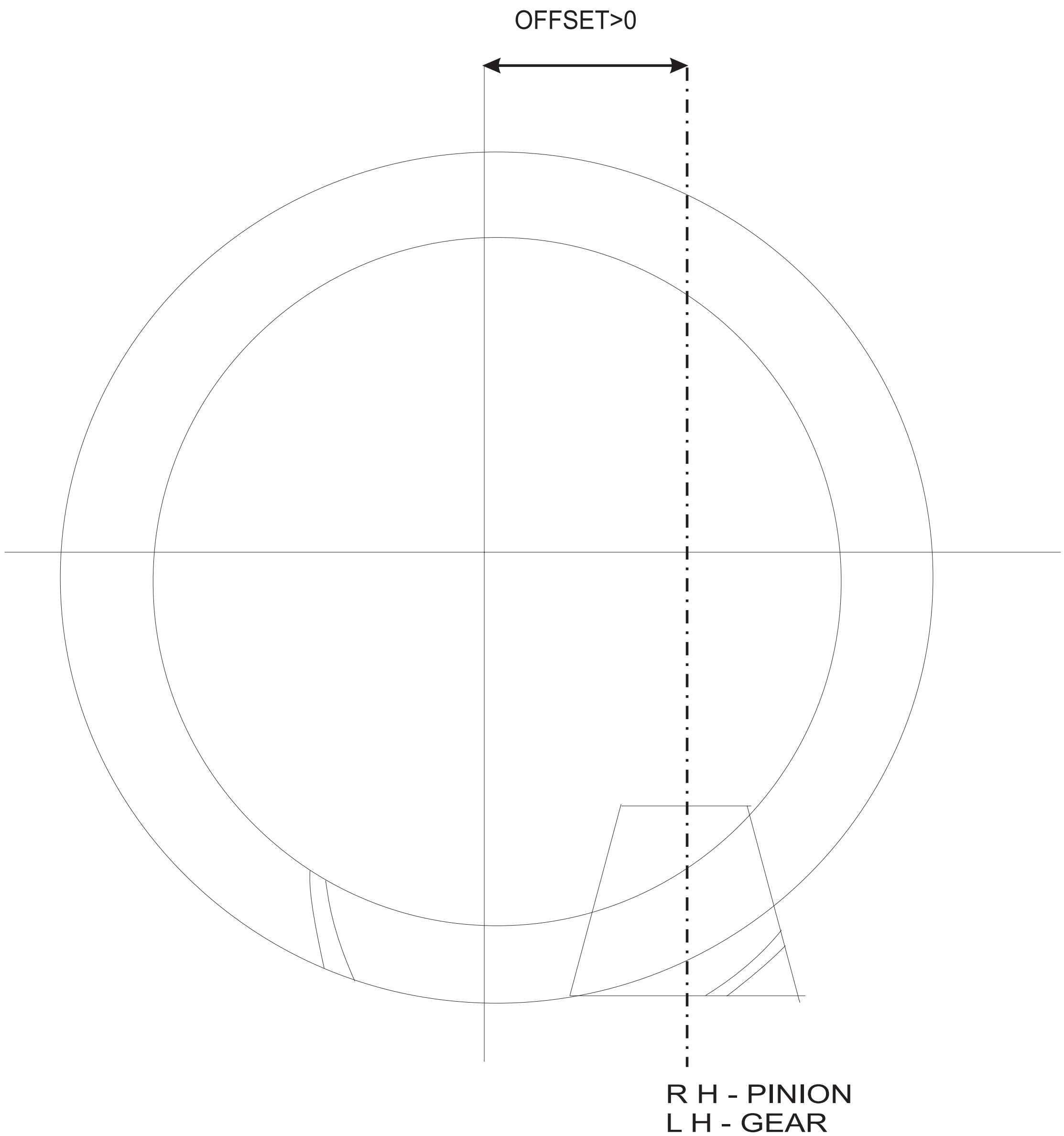


Figure 5.3: Sign convention for offset menu for lefthanded gear.

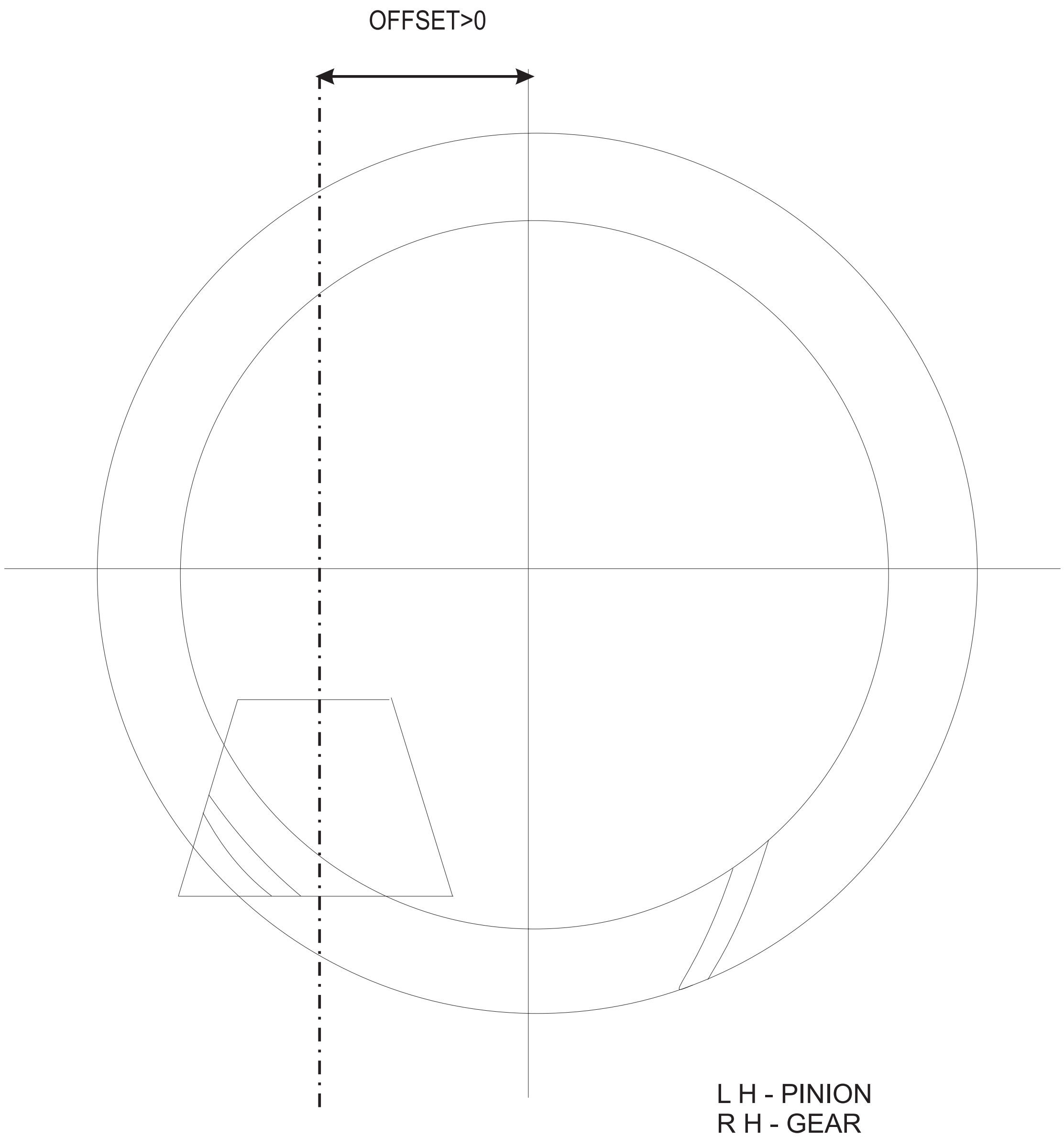


Figure 5.4: Sign convention for offset menu for righthanded gear.

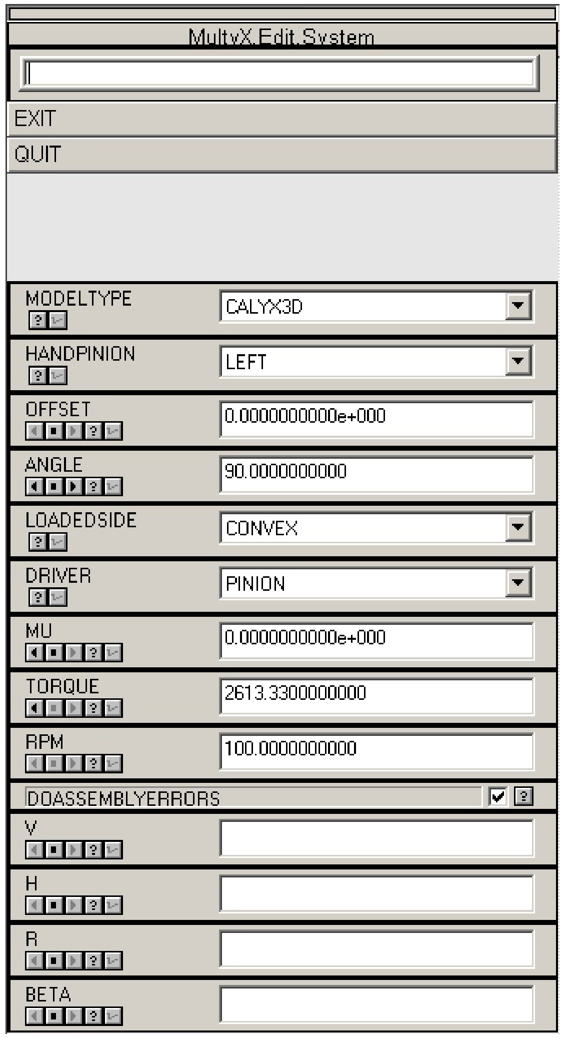


Figure 5.5: The assembly errors menu.

5.3 Assembly errors

If you set the DOASSEMBLYERRORS flag shown in Figure 5.2 then the assembly errors will also be included in the analysis(Figure 5.5). Assembly error V is the assembly error of the gear with respect to the pinion in the plane of rotation of the gear in a direction perpendicular to the pinion axis. Assembly error H is the error in the axial direction of the pinion. It is positive when the pinion is moved out of mesh relative to the gear. Assembly error R is the error in the axial direction of the gear. It is positive when the gear is moved out of mesh relative to the pinion. Assembly error BETA is the difference in the shaft angle from the design shaft angle. It is positive when the actual shaft angle is larger than the design shaft angle. Figures 5.6 and 5.7 shows the sign convention used to model the assembly errors for the left handed and right handed gear respectively.

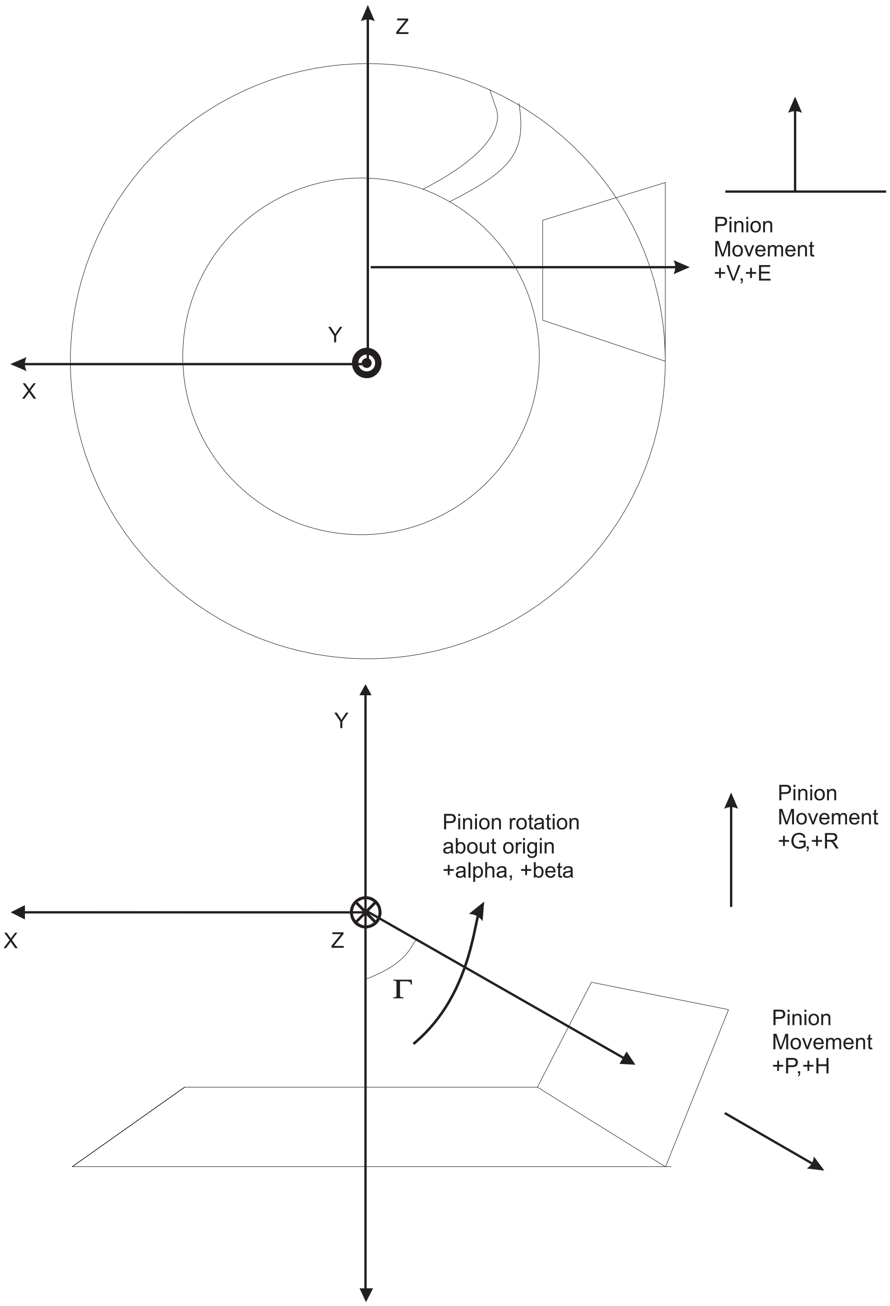


Figure 5.6: Sign convention for modeling assembly errors for a lefthanded gear

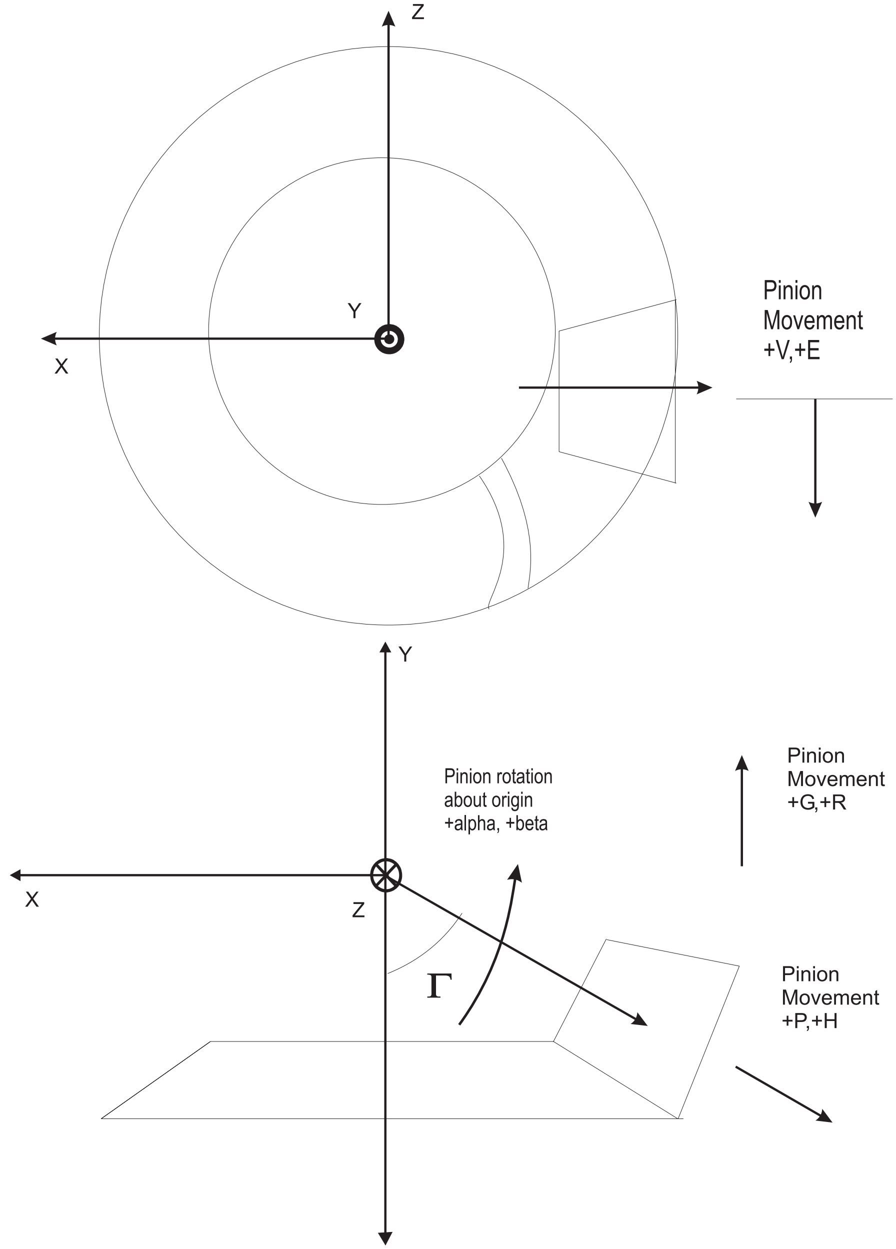


Figure 5.7: Sign convention for modeling assembly errors for a righthanded gear

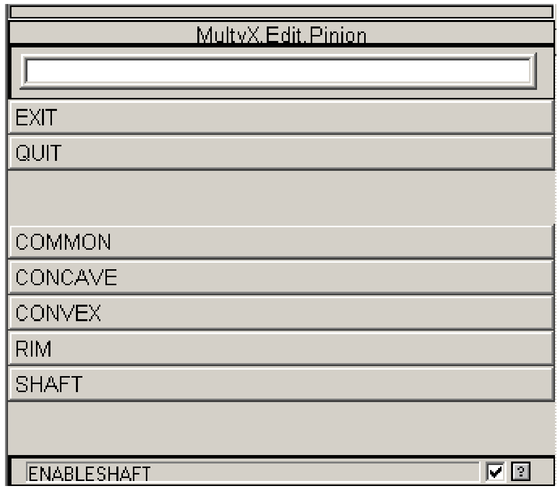


Figure 5.8: The pinion data menu.

5.4 Pinion and gear data

The PINION command in Figure 5.1 leads to the pinion data menu shown in Figure 5.8. It has 4 submenus- COMMON(common design and blank data), CONCAVE(concave side data), CONVEX(convex side data) and RIM(rim data). The GEAR command in Figure 5.1 leads to the gear data menu shown in Figure 5.9. The gear menu is similar to the pinion menu except for the feature TYPE. This feature decides whether the gear type is Generated or Formate. For a Formate gear the ratio of rolls(ratio of number of teeth on the imaginary gear to the number of teeth on the gear being cut) is zero in the Special Analysis File. For a Generated type this value is non-zero.

5.5 Common design and blank data

The COMMON command in Figure 5.8 leads to the common design and blank data menu shown in Figure 5.10. The common design and blank data for the gear is similar to that of the pinion. The various parameters for this menu are explained in Table 5.2 & Table 5.3

By default the thickness will be assumed to be defined at the 'mean point' (the intersection of the mid-face cone) with the pitch cone. This is the normal chordal tooth thickness (measured along the normal plane). The normal chordal tooth thickness is related to the transverse thickness by the following relationship.

$$\text{Normal chordal tooth thickness} = \text{Transverse chordal tooth thickness} \times \cos(\text{spiral angle})$$

If the SPECIFYMSRPT item is turned on, then the thickness can be specified at a point at an arbitrary 'measuring addendum' distance from the tip at mid-face as shown in Figure 5.11

The face angle, back angle, front angle, spiral angle and pitch angle values are given in radians in the Special Analysis File. The user should convert them in to degrees before entering them in their respective menus. If the ISRACERIGID flag is set, then the inner diameter of the pinion/gear is assumed to behave like a rigid cylinder. If it is not rigid then the bearing race is modelled using Fourier shape function in the circular direction. The Rayleigh damping model assumes that the damping matrix C for a finite element is related to the mass matrix M

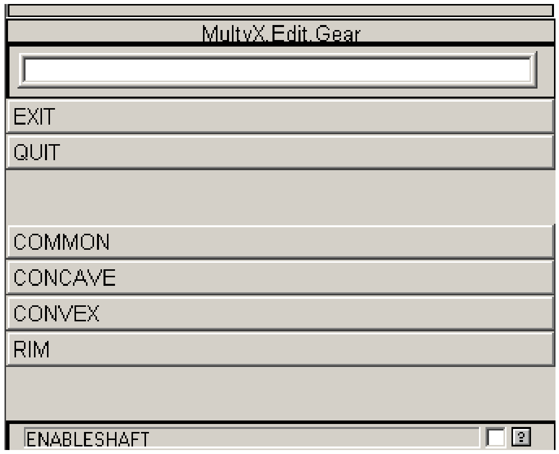


Figure 5.9: The gear data menu.

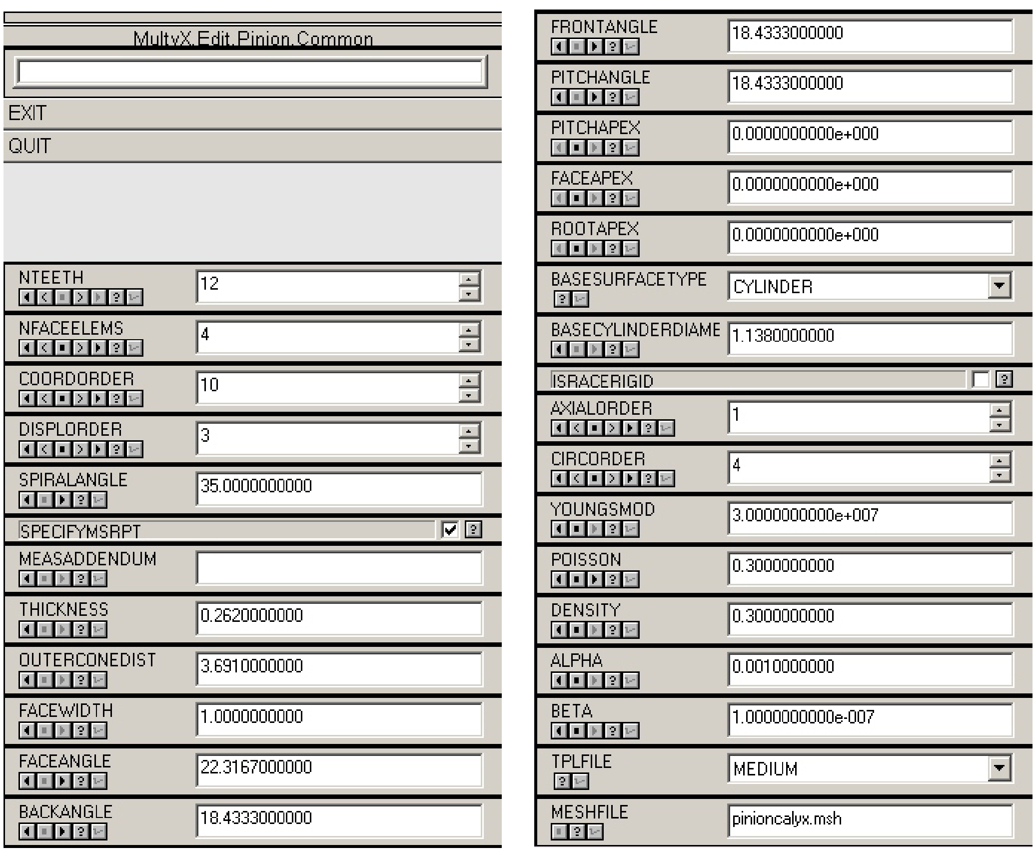


Figure 5.10: The common design and blank data menu.

Table 5.2: Common design and blank parameters

Item	Description	Special Analyis File Details
NTEETH	Integer, Number of teeth on the pinion/gear	Pinion-Record#1,Item#1 & Gear-Record#1,Item#2
NFACEELEMENTS	Integer, Number of elements across the face width of the pinion/gear	
COORDORDER	Integer, Limit on the order of coordinate axodes for the pinion/gear	
DISPLORDER	Integer, Limit on displ order of axodes	
SPIRALANGLE	Float, Angle between the tooth trace and an element of the pitch cone(Deg)	Pinion-Record#1,Item#10 & Gear-Record#8,Item#4
SPECIFYMSRPT	Boolean, Whether to specify the thickness measuring point	
MEASADDENDUM	Float, The measuring addendum	
THICKNESS	Float, Mean transverse tooth thickness	Pinion-Record#47,Item#7 & Gear-Record#47,Item#8
OUTERCONEDIST	Float, Distance from the apex of the pitch cone to the outer ends of the tooth	Pinion-Record#3,Item#9 & Gear-Record#3,Item#10
FACEWIDTH	Float, Face width of the tooth	Pinion-Record#1,Item#6 & Gear-Record#1,Item#5
FACEANGLE	Float, Angle between the element of the face cone and axis(Deg)	Pinion-Record#5,Item#6 & Gear-Record#7,Item#6
BACKANGLE	Float, Angle between the element of the back cone and the plane perpendicular to the axis of rotation(Deg)	Pinion-Record#5,Item#9 & Gear-Record#7,Item#9
FRONTANGLE	Float,Angle between the element of the front cone and the plane perpendicular to the axis of rotation(Deg)	Pinion-Record#5,Item#10 & Gear-Record#7,Item#10
PITCHANGLE	Float, Angle between the element of the pitch cone and axis(Deg)	Pinion-Record#5,Item#5 & Gear-Record#7,Item#5
PITCHAPEX	Float, Pitch apex beyond crossing point distance	Pinion-Record#5,Item#15 & Gear-Record#7,Item#15
FACEAPEX	Float, Face apex beyond crossing point distance	Pinion-Record#4,Item#3 & Gear-Record#4,Item#4
ROOTAPEX	Float, Root apex beyond the crossing point distance	Pinion-Record#5,Item#12 & Gear-Record#7,Item#12

Table 5.3: Common design and blank parameters

Item	Description	Special Analyis File Details
BASESURFACETYPE	Switch, Kind of surface to be used as the base surface(CYLINDER/CONE)	
BASECYLINDERDIAME	Float, Diameter of the cylinder used as the base of the tooth mesh(IF BASESURFACETYPE=CYLINDER)	
BASECONEANGLE	Float, Angle(Deg) between the element of the base cone and the axis(IF BASESURFACETYPE=CONE)	
BASECONEAPEX	Float, Inner cone apex beyond crossing point distance(IF BASESURFACETYPE=CONE)	
ISRACERIGID	Boolean, Whether the bearing race is a rigid surface	
AXIALORDER	Integer, Polynomial order in the face direction	
CIRCORDER	Integer, Fourier series order in the circular direction	
YOUNGSMOD	Float, Youngs modulus for the pinion/gear	
POISSON	Float, Poisson's ratio for the pinion/gear	
DENSITY	Float, Density of the pinion/gear	
ALPHA	Float, Damping constant alpha for the pinion/gear	
BETA	Float, Damping constant beta for the pinion/gear	
TPLFILE	Switch, Template file for the pinion/gear(Refer to appendix for details)	
MESHFILE	String, Mesh file name for the pinion/gear	

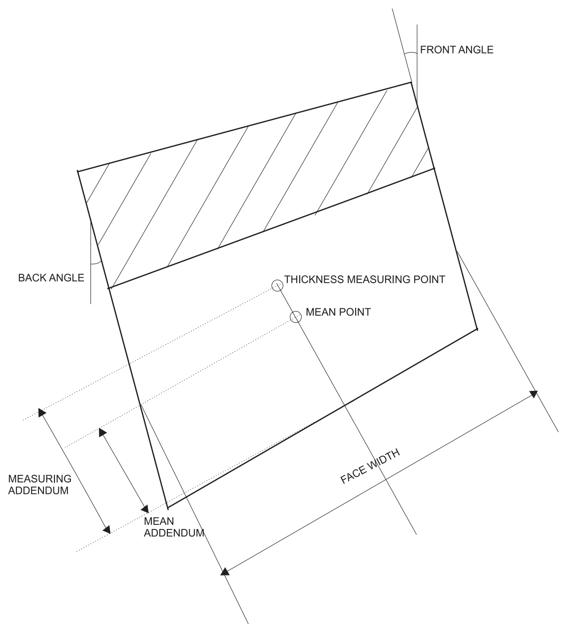


Figure 5.11: Thickness measurement at an arbitrary point.

and stiffness matrix K by the linear relationship C= alpha*M + beta*K. The variables ALPHA and BETA in the common data menu are the values of alpha and beta respectively in the above equation.The MESHFILE which contains the element connectivity and the geometrical information is created by the program. Figure 5.12 shows the Pinion common parameters. Figure 5.13 shows the Gear common parameters.

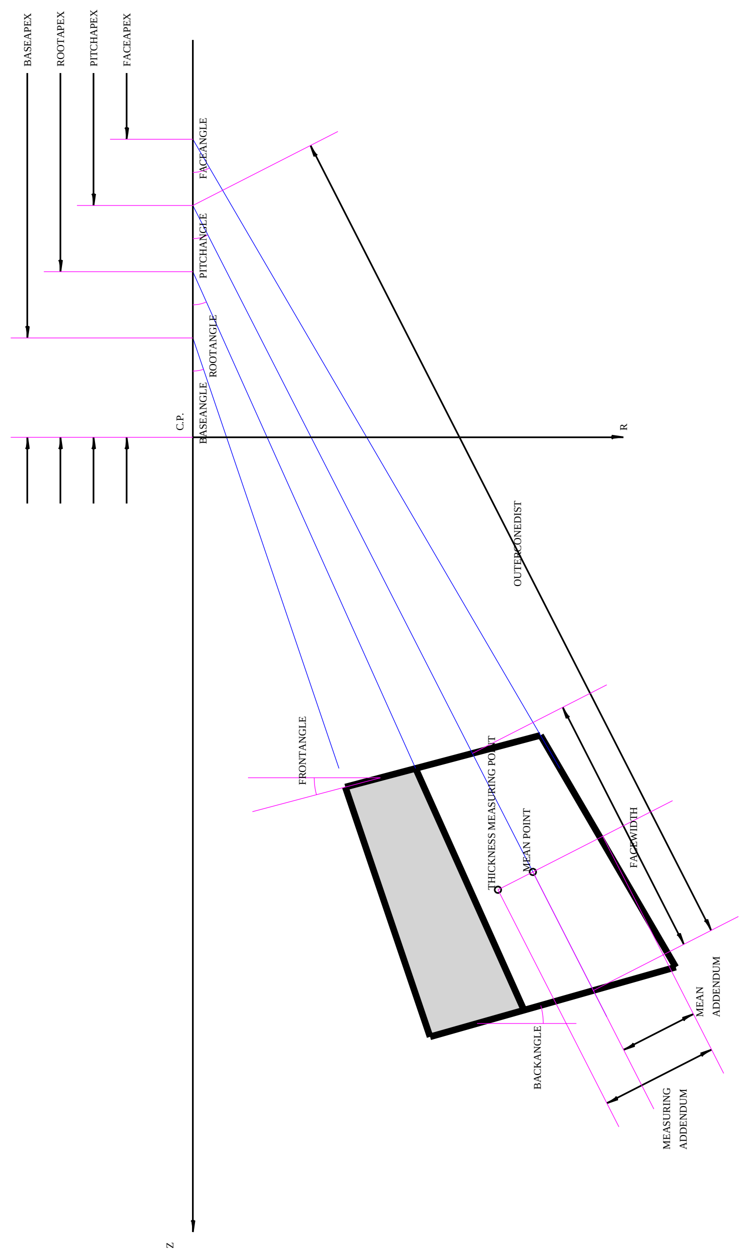


Figure 5.12: The Pinion Common Parameters.

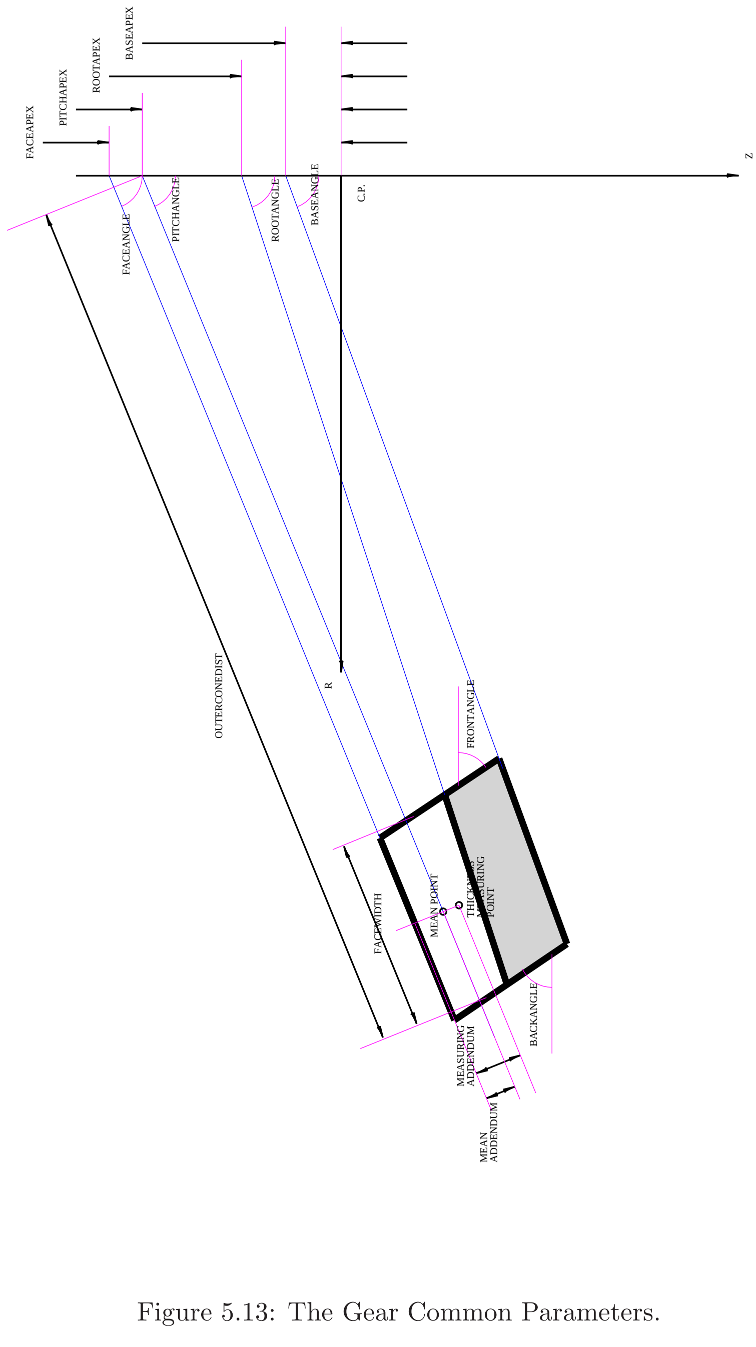


Figure 5.13: The Gear Common Parameters.

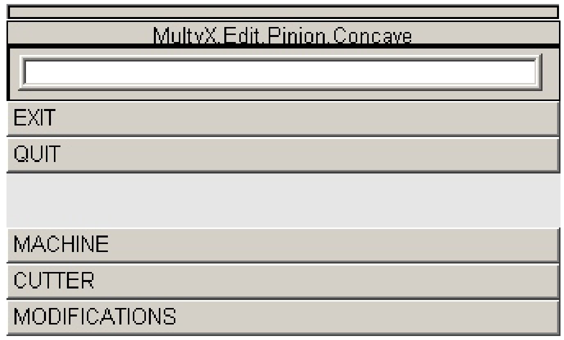


Figure 5.14: The Concave tooth side data menu.

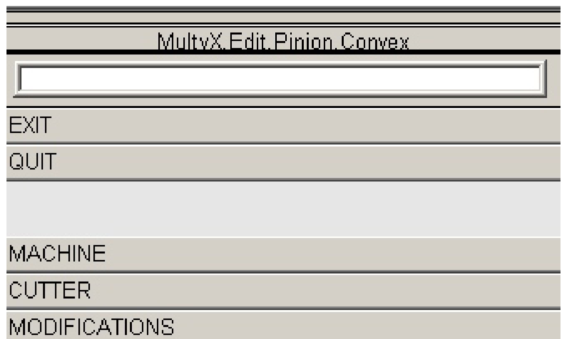


Figure 5.15: The Convex tooth side data menu.

5.6 Concave and Convex side data

The CONCAVE and CONVEX command in the pinion menu(Figure 5.8) leads to the concave and convex side data menus shown in Figures 5.14 & 5.15 . There are three submenus-MACHINE, CUTTER and MODIFICATIONS. All three data submenus are similar for both the concave and convex side data except that the data comes from a different part of the Special Analysis File. For the generated type gear the data menus for concave and convex side gear tooth are similar to those for the pinion. They are slightly different for the formate type gear.

5.7 Machine settings for the pinion

The MACHINE command in Figures 5.14 & 5.15 leads to the details of the face hobbing operation settings used for the manufacturing of the hypoid pinion/gear shown in Figure 5.16. Table 5.4 explains all the machine parameters in this menu.

The values for TILTANGLE, SWIVELANGLE, ROOTANGLE, and the CRADLEANGLE are given in Radians in the Special Analysis File. The user should convert them in to degrees before entering them into the menu.

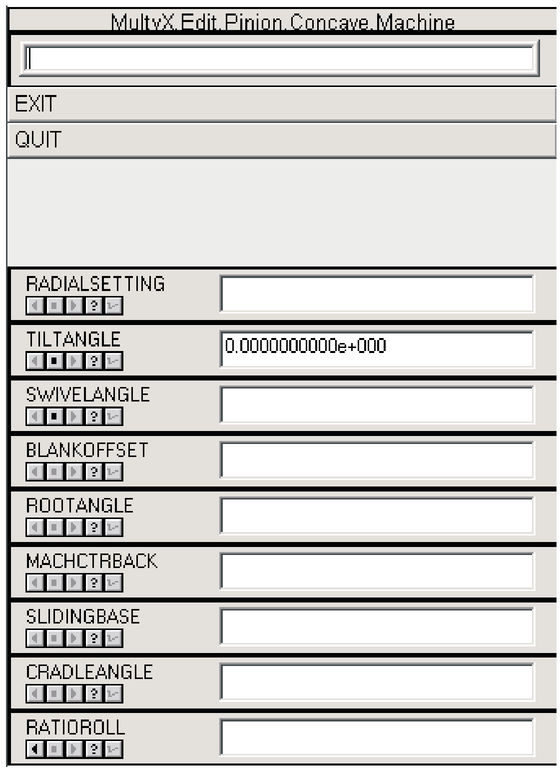


Figure 5.16: Machine settings for the pinion

Table 5.4: Machine setting parameters for concave and convex tooth sides of the face-hobbed pinion

Item	Description	Special Analyis File Details
RADIALSETTING	Float, Distance between the cradle axis and the cutter axis when they are parallel	Concave side-Record#15,Item#1 & Convex side-Record#18,Item#1
TILTANGLE	Float, Tilt of the cutter axis with respect to the direction of the cradle axis(Deg)	Concave side-Record#15,Item#2 & Convex side-Record#18,Item#2
SWIVELANGLE	Float, Direction of the cutter spindle tilt with respect to the gear being generated(Deg)	Concave side-Record#15,Item#3 & Convex side-Record#18,Item#3
BLANKOFFSET	Float, Offset between the work spindle axis and the cradle axis	Concave side-Record#15,Item#4 & Convex side-Record#18,Item#4
ROOTANGLE	Float, Angle between the element of the root cone and its axis(Deg)	Concave side-Record#15,Item#5 & Convex side-Record#18,Item#5
MACHCTRBACK	Float, Axial distance from the root apex of the gear to the spindle mounting surface	Concave side-Record#15,Item#6 & Convex side-Record#18,Item#6
SLIDINGBASE	Float, Position of the sliding base with respect to the machine plane	Concave side-Record#15,Item#7 & Convex side-Record#18,Item#7
CRADLEANGLE	Float, Angular position of the cradle axis with respect to the gear generated	Concave side-Record#16,Item#9 & Convex side-Record#19,Item#9
RATIOROLL	Float, Ratio of the number of teeth on the imaginary gear to the number of teeth on the gear being cut	Concave side-Record#15,Item#13 & Convex side-Record#18,Item#13

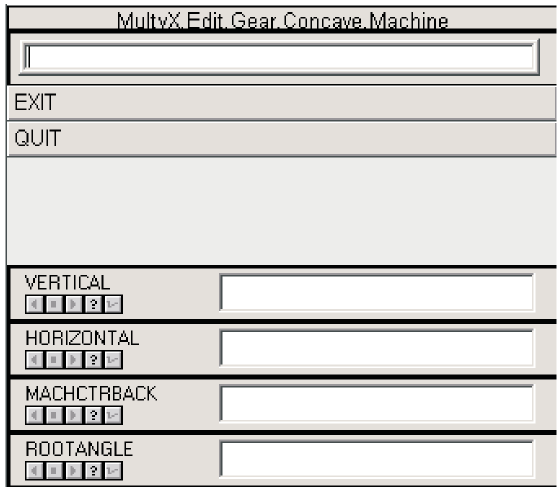


Figure 5.17: Machine settings for the gear

Table 5.5: Machine settings for the gear

Item	Description	Special Analyis File Details
HORIZONTAL	Float, Horizontal setting of the cutter	Concave side-Record#28,Item#2 & Convex side-Record#24,Item#2
VERTICAL	Float, Vertical setting of the cutter	Concave side-Record#28,Item#1 & Convex side-Record#24,Item#1
ROOTANGLE	Float, Angle between the element of the root cone and its axis(Deg)	Concave side-Record#28,Item#4 & Convex side-Record#24,Item#4
MACHCTRBACK	Float, Axial distance from the root apex of the gear to the spindle mounting distance	Concave side-Record#28,Item#3 & Convex side-Record#24,Item#3

5.8 Machine settings for the gear

Figure 5.17 shows the menu for the machine settings of the concave side of gear. The parameters for the convex side are similar to those of the concave side. The ROOTANGLE value is given in radians in the Special Analysis File. The user should convert it in to degrees before entering it in the menu. Table 5.5 explains all the machine parameters associated with the manufacturing of the gear.

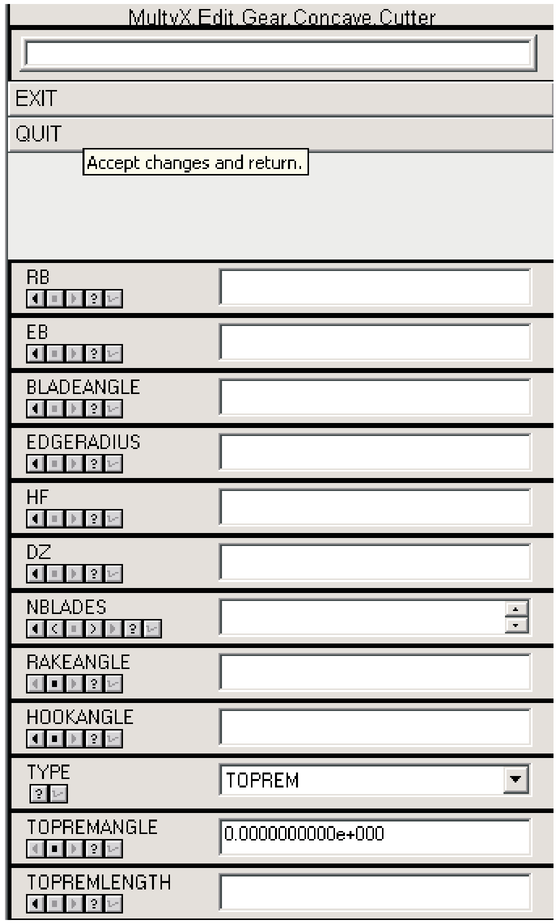


Figure 5.18: Cutter specification for the Pinion and Gear

5.9 Cutter specification for the Pinion and Gear

Figure 5.18 shows the menu for defining the cutters for either side of the pinion or gear. The parameters EB, HOOKANGLE, RAKEANGLE, and TOPREMANGLE are all angular quantites specified in Degrees. They must be converted from Radians to Degrees when they are extracted from the special analysis file.

TYPE is the cutter type. It may be set to STRAIGHT, TOPREM, or CURVED. An extra parameter RHO appears when the cutter TYPE is CURVED. The parameters are explained in Table 5.6.

5.10 Surface Modifications

The Surface modification feature presently is not implemented in the HypoidFaceHobbed program.

Table 5.6: Cutter specification for the Pinion and Gear

Item	Description	Special Analysis File Details							
		Pinion				Gear			
		Concave		Convex		Concave		Convex	
		Rec	Itm	Rec	Itm	Rec	Itm	Rec	Itm
RB	Float, Reference Radius at Pitch Point	52	1	56	1	57	1	53	1
EB	Float, Offset angle of blade.	52	2	56	2	57	2	53	2
HF	Float, Reference height at pitch point.	52	3	56	3	57	3	53	3
DZ	Float	52	4	56	4	57	4	53	4
BLADEANGLE	Float, Cutter blade angle(Deg)-	52	5	56	5	57	5	53	5
NBLADES	Integer, No. of blade groups on cutter head.	52	6	56	6	57	6	53	6
RAKEANGLE	Float, Side rake angle (Deg.)	52	7	56	7	57	7	53	7
HOOKANGLE	Blade slot tilt angle (Deg.)	52	8	56	8	57	8	53	8
EDGERADIUS	Float, Radius of the cutter edge(Tip radius)	16	15	19	15	26	15	22	15
TOPREMANGLE	Float, Toprem angle	17	1	20	1	27	1	23	1
TOPREMLENGTH	Float, Toprem length	17	2	20	2	27	2	23	2
RHO	Float, Spherical radius of cutter blade	15	12	18	12	24	12	21	12

5.11 Modelling the Rim

The RIM command in Figure 5.8 leads us to the Rim data menu for the pinion shown in Figure 5.20. The DORIM flag controls whether the rim model will be generated in addition to the model of the contacting tooth. If it is not generated then the tooth finite element model will be constrained at the base. The rim is specified by sequence of line segments in order to define the rim cross section. The first segment is closest to the tooth, and the last segment is the farthest. Each segment 'i' has two endpoints, A_i and B_i. A_i is the end that is closer to the crossing point as shown in Figure 5.19. Each end point is specified by its radial coordinate r and axial coodinate z. The Rim data parameters are explained in Table 5.7. The RIM menu for the gear is same as that for the pinion.

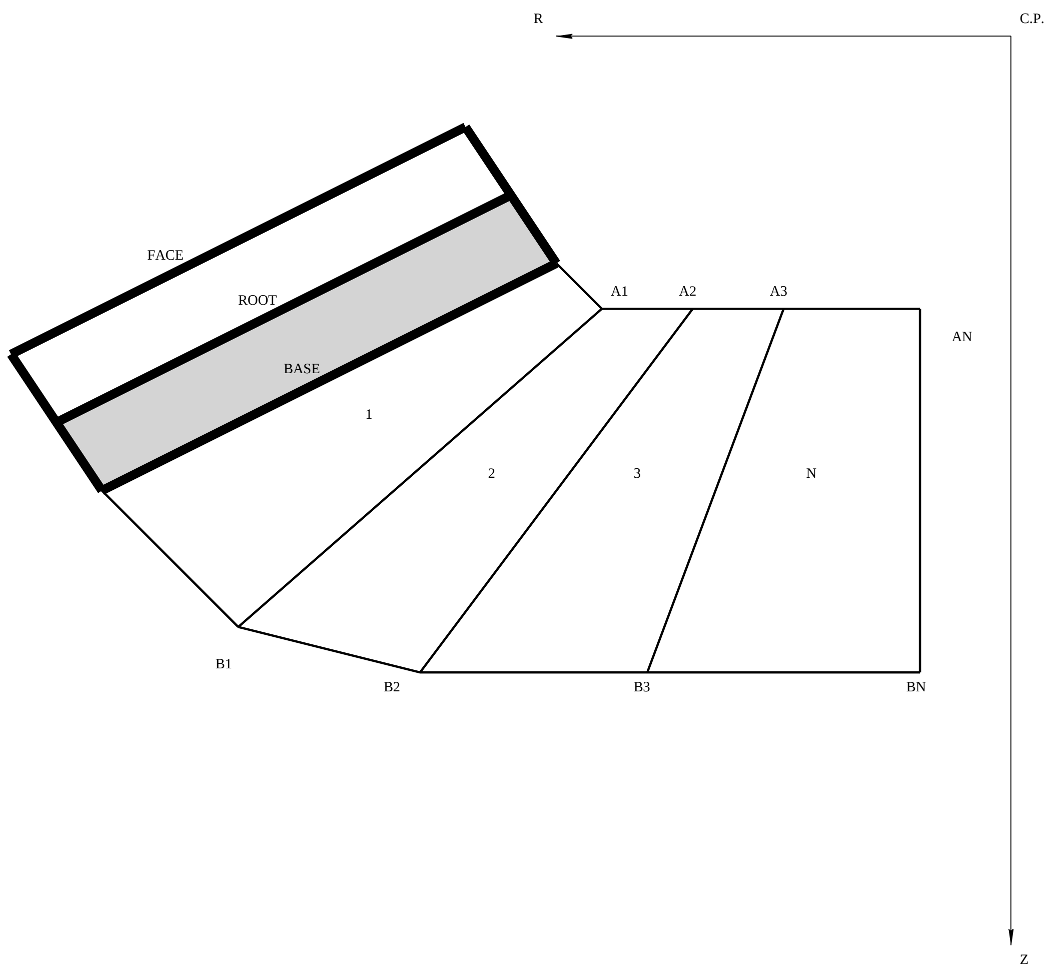


Figure 5.19: The Rim geometry

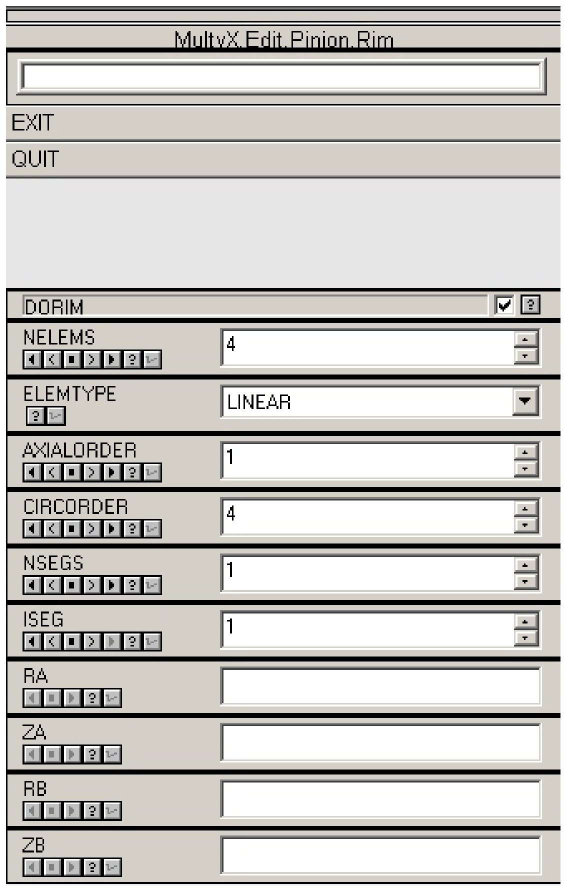


Figure 5.20: The Rim data menu

Table 5.7: The rim parameters

Item	Description
DORIM	Boolean, Whether to build a rim model
NELEMS	Integer, Number of rim elements across the face
ELEMTYPE	Switch, Type of finite element. Available options are LINEAR, QUADRATIC and CUBIC
AXIALORDER	Integer, Polynomial order in the face direction
CIRCORDER	Integer, Fourier series order in the circular direction
NSEGS	Integer, Number of segments used to define the rim
ISEG	Integer, Segment number for which data is being displayed
RA	Float, Vector indexed by ISEG, Radial coordinate at side A.
RB	Float, Vector indexed by ISEG, Radial coordinate at side B.
ZA	Float, Vector indexed by ISEG, Z coordinate at side A.
ZB	Float, Vector indexed by ISEG, Z coordinate at side B.

5.12 Modeling the shaft

When the pinion or gear rides on a flexible shaft, the shaft deflection can also affect the gear contact significantly. In such a situation, it may become necessary to incorporate a finite element model of the shaft.

Presently, the shaft model cannot be combined with a rim model of the type EXTERNALSPLINE or INTERNALSPLINE. It can be combined with the SIMPLE, or WEBBED type of rim model. When the ENABLESHAFT item is checked in the pinion or gear menu (Figures 5.8 and 5.9), the submenu SHAFT appears. This submenu is shown in Figure 5.21.

The shaft is built of a number NSEGS of segments, as shown in Figure 5.22. The same material properties (YOUNGSMOD, POISSON and DENSITY) are used for all the segments in the shaft. The shaft is positioned with respect to the gear or pinion mid-face plane by specifying the offset value TOOTHOFFSET.

A specific segment is selected through the index ISEG. The segment's axial length is specified through the item LENGTH. The shape OUTERSHAPE of the outer surface of the segment can be set to CYLINDRICAL or CONICAL (Figure 5.23). For a cylindrical outer shape, only one outer diameter DOUTER is needed. For a conical outer shape, two diameters D1OUTER and D2OUTER are required. Similarly, the shape INNERSHAPE of the innersurface can be CYLINDRICAL or CONICAL. Accordingly only one diameter DINNER or two diameters D1INNER and D2INNER will need to be specified (Figure 5.24).

If the outer surface of the segment connects to the pinion or gear, then OUTERCONNECTIONTYPE should be set to GEAR. The diameter of the outer surface of the segment must match the inner diameter of the gear or its rim. The axial extent must overlap that of the pinion or gear.

If the segment connects to a bearing, then OUTERCONNECTIONTYPE must be set to BEARINGRIGID (for a rigid bearing) or BEARINGFLEXIBLE (for a flexible bearing). A bearing file name should then be specified through OUTERBRGFILENAME. If the torque flows through the outer surface, then OUTERCONNECTIONTYPE should be made CONSTRAINEDRIGID or CONSTRAINEDFLEXIBLE.

If the outersurface neither connects to the pinion (or gear) or bearing, and is not constrained, then the OUTERCONNECTIONTYPE should be set to FREE.

A similar set of options is available for the inside surface of the shaft segment through the item INNERCONNECTIONTYPE.

At least one segment must have its inner surface or outer surface constrained.

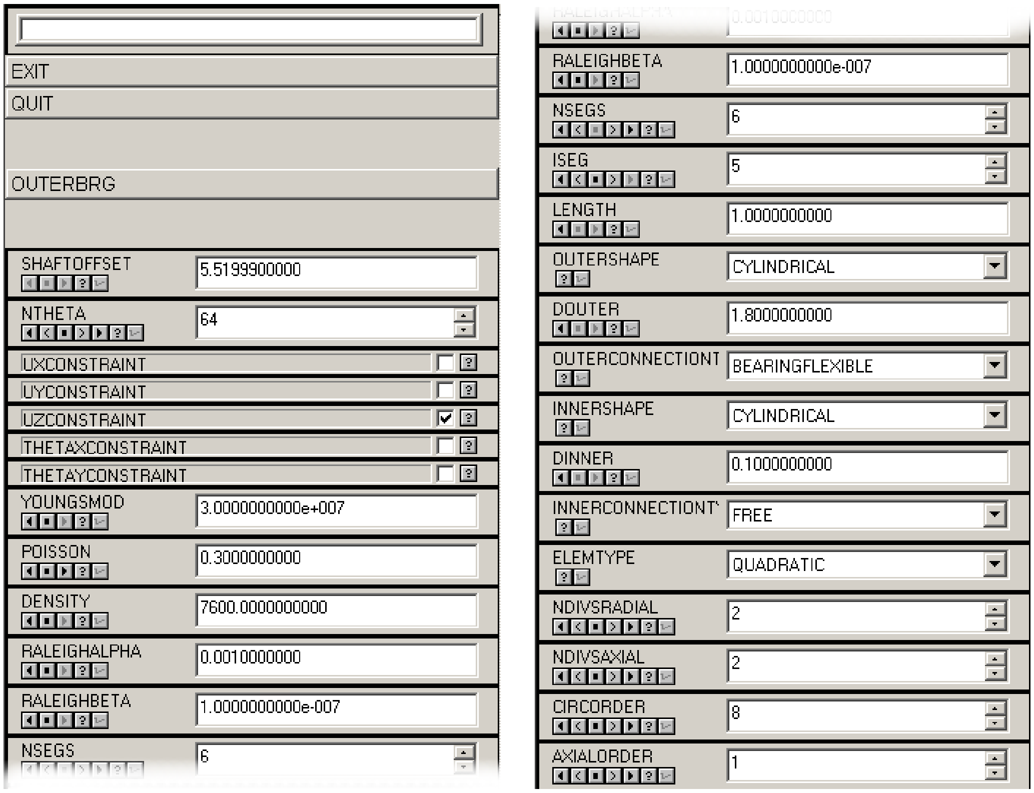


Figure 5.21: The menu for specifying shaft data.

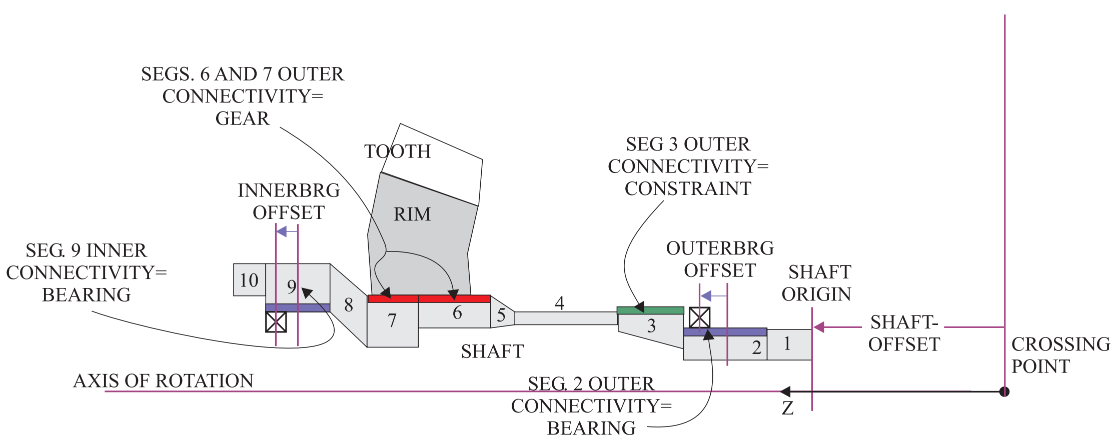
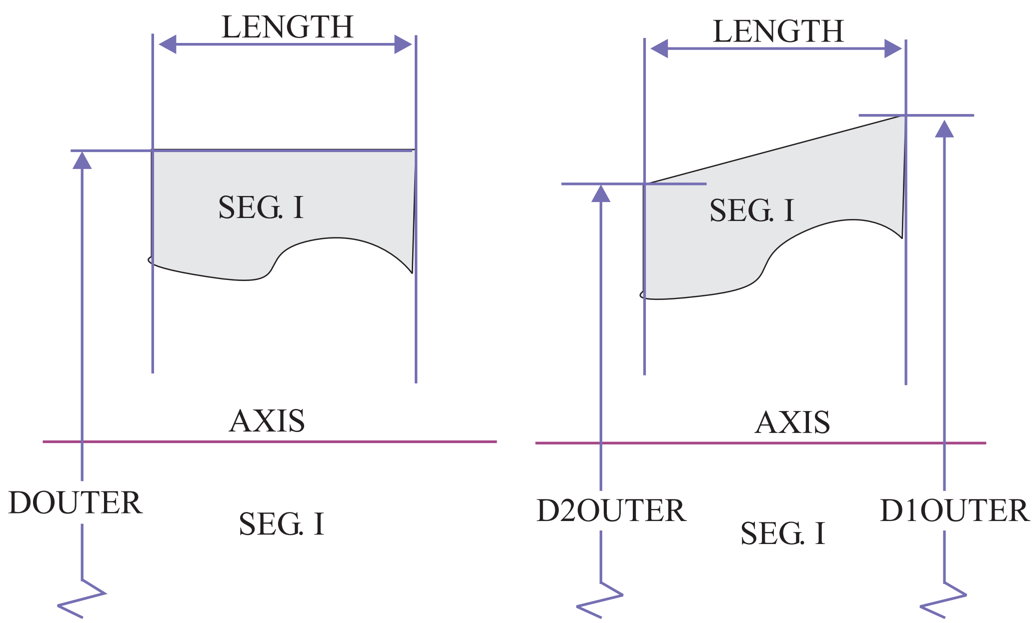
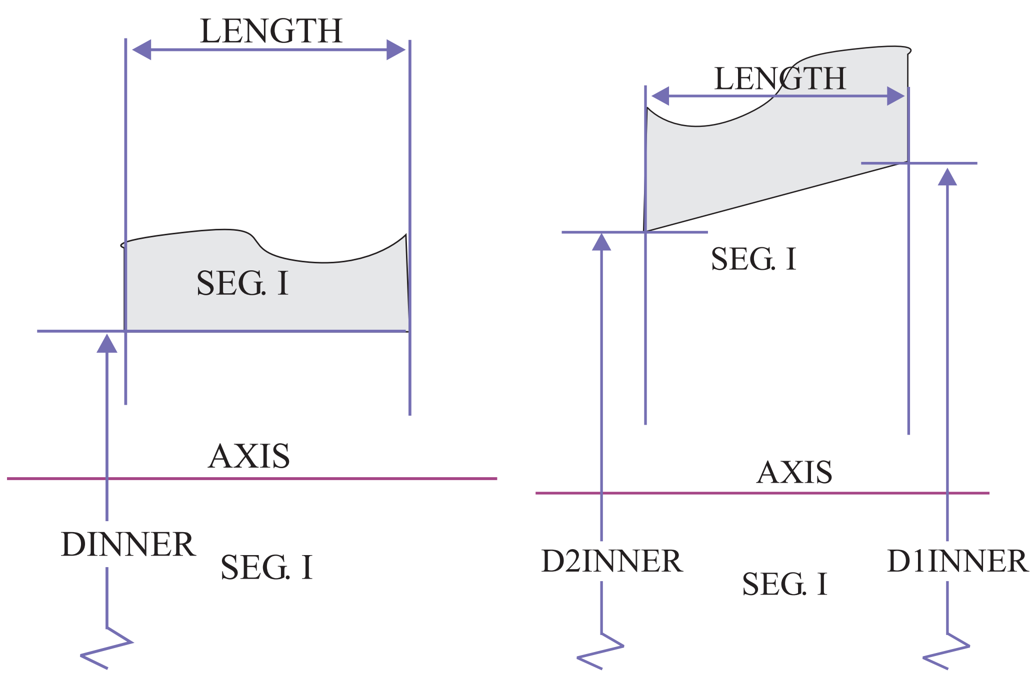


Figure 5.22: The shaft model.



(a) CYLINDRICAL (b) CONICAL

Figure 5.23: The dimensions of the outer surface of a shaft segment.



(a) CYLINDRICAL (b) CONICAL

Figure 5.24: The dimensions of the outer surface of a shaft segment.

Chapter 6

Running an Analysis

The analysis is started by using STARTANAL command of Figure 3.4.

Before starting an analysis, sensor locations have to be set up to measure stress and loads in the model. This is done through the SURFGAGES, FEPROBES and LOADSENSORS commands in the main menu (Figure 3.4). Additional analysis parameters and settings are controlled through the SETUP command.

6.1 Surface gages

A surface gage is used to measure the critical stress along tooth surfaces. The reading of each gage is the most critical stress measured over a user defined range of teeth, profile, face and depth along a specific surface.

Figure 6.1 shows the Surface Gage setup menu. The number of gages NGAGES has to be entered first. Then the gage number for a particular gage can be entered into the GAGE box, and the gage information can be typed into the remaining boxes. For each gage, the BODY item selects which of the individual components in the system the gage is attached to. For a HypoidFaceHobbed gear analysis, there are two bodies, the pinion and the gear.

After the Body is selected, the surface on which the gage should be attached should be selected. The gear teeth typically have four surfaces. SURFACE1 and SURFACE2 cover the entire involute and fillet areas of the two sides Side 1 and Side 2, respectively, of the teeth. FILLET1 and FILLET2 cover only the fillet region of Side 1 and Side 2, respectively.

When there are multiple copies of a surface on a body, each individual copy of that surface is called an instance of that surface, and is given a unique instance number. In the case of gear tooth surfaces, the instance number is the same as the tooth number. The parameters TOOTHBEGIN and TOOTHEND define a range of teeth over which the gages will be placed. The reading of the gage is the stress at the most critical tooth. If the value of TOOTHBEGIN is greater than TOOTHEND, then the search range will wrap around the last tooth.

There are two parameters that identify a point on a surface. We refer to these two parameters as S which varies in the profile direction, and T which varies in the face width direction.

The profile parameter S increases from fillet to the tip on Side 1 of a tooth, and from the tip to the fillet on Side 2, as shown in Appendix A. The parameters SPROFBEGIN and SPROFEND define a range over which the stress will be calculated. These are in surface local units as shown in Appendix A. The GAGE will read out the critical value of stress in this range. The NUMSPROF parameter controls how many search points should be used over this range.

The face parameter T varies from -1.0 to $+1.0$ over the face of the tooth. The face width range parameters TFACEBEGIN, control TFACEEND range over which the search is carried out, and NUMTFACE controls the number of search points within this range.

The DEPTHBEGIN, DEPTHEND and NUMDEPTH parameters extend the search range to a number NUMDEPTH of points ranging in depth from DEPTHBEGIN to DEPTHEND below the surface. This is an expensive computation, and should not be used unless necessary. The surface gage will measure the stress at the critical depth. The depth is in physical length units.

Because finite element stresses computed very close to the highly concentrated contact loads can have a large amount of error, we need a way to screen out points that are too close. The parameter DISTMIN is the minimum allowed distance of a stress calculation point from a contact point. Stresses will not be calculated at any point whose distance from a contact point is less than this value. This distance is in physical length units.

During the analysis, all the surface gage readings are written to a file called `GAGES.DAT`. Each row in this file corresponds to a time instant. The first column in the file contains the value of the time. The remaining columns contain the readings of the surface gages. There are four columns of data for each gage. The first column for a gage contains the critical maximum principal normal stress (s_1) over its search range. The second column contains the value of the critical minimum principal normal stress (s_3). The third column contains the critical maximum shear stress (τ_{max}), and the fourth column contains the critical Von Mises' shear stress (s_{vm}). The columns are separated by tabs.

6.2 Finite element probes

Finite Element Probes can be used to output stresses at a particular point when its element number and local coordinates are known. The Element numbering used in the gear tooth finite element meshes is shown in Appendix A. Figure 6.2 shows the finite element probe input menu. The BODY parameter selects the particular body or component to be probed. Each body can have many finite element meshes. The MESH parameter selects which finite element mesh should be probed. There may be many copies or instances of the finite element mesh. Each copy is given an instance number. In the case of a gear tooth mesh, this instance number is the same as the tooth number. The TOOTH parameter selects the instance number. The ELEM parameter selects the finite element number within the mesh. The XI, ETA and ZETA values are the local coordinates within the finite element. XI, ETA and ZETA vary between -1 and $+1$ over the element. Appendix A shows the orientation of the local coordinate axes for each finite element in the various mesh templates.

The COMPONENT parameter selects which stress component should be measured by the probe. Available options are Maximum principal normal stress (s_1), minimum principal normal stress (s_3), maximum shear stress (τ_{max}), Von Mises' octahedral shear stress (s_{vm}) and the displacement magnitude (u). The data measured by the finite element probes is written to a file called `PROBES.DAT`. The data file has a row for each time instant. The first column contains the value of time. Each subsequent column contains the readout of an individual probe.

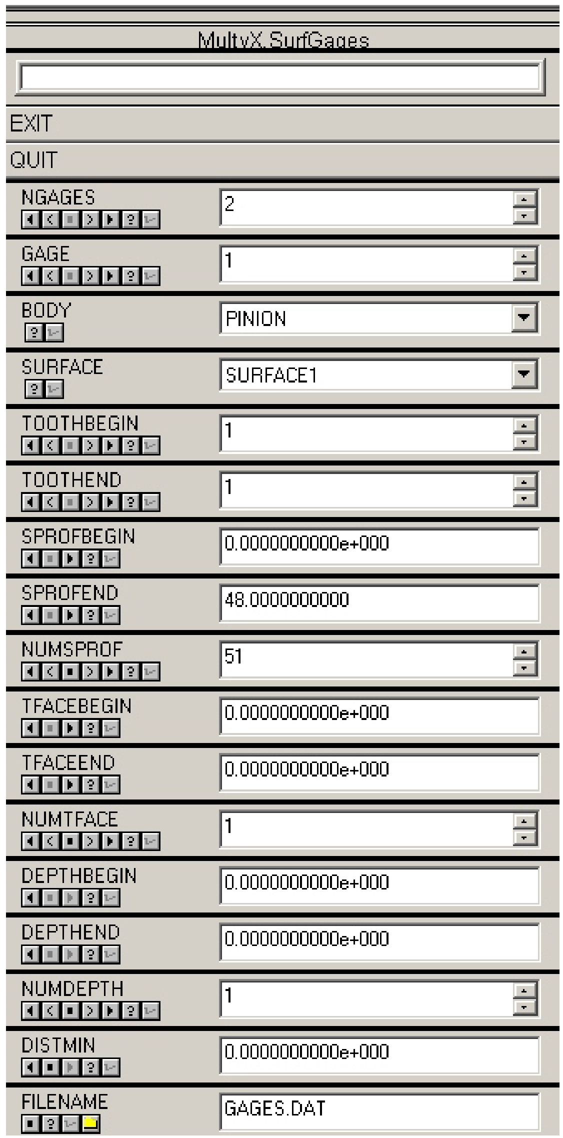


Figure 6.1: The surface gage menu

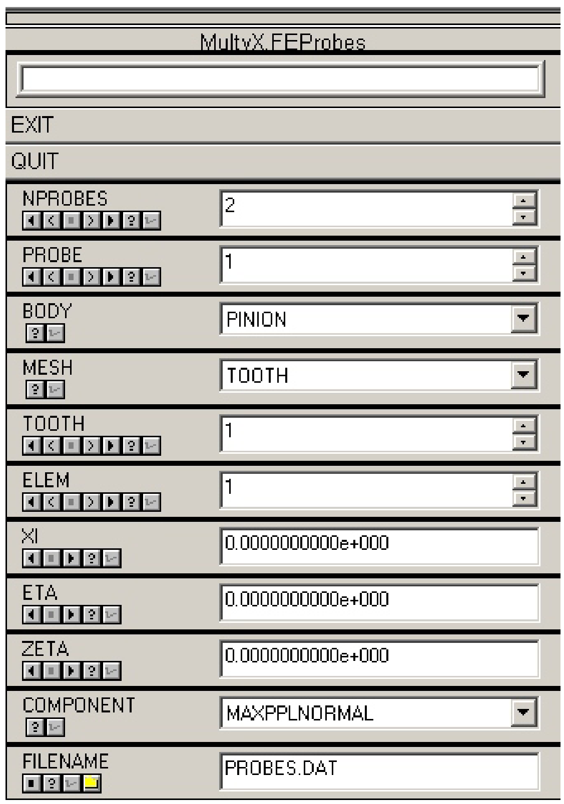


Figure 6.2: The finite element probe menu

6.3 Load sensors

Load sensors are used to measure the contact loads generated at the contact surfaces. Figure 6.3 shows the load sensor menu used to set up the sensors. The SURFPAIR item selects the contact surface pair for which the contact load is of interest. Each surface pairing has two contacting members or bodies. The MEMBER parameter selects one of these two bodies, and the TOOTH item selects the individual surface instance number within that body. The outputs of all the sensors are put into a file called `LOADS.DAT`. This file has one row for each instant of time. The first column contains the time. Each subsequent column contains the reading of one load sensor.

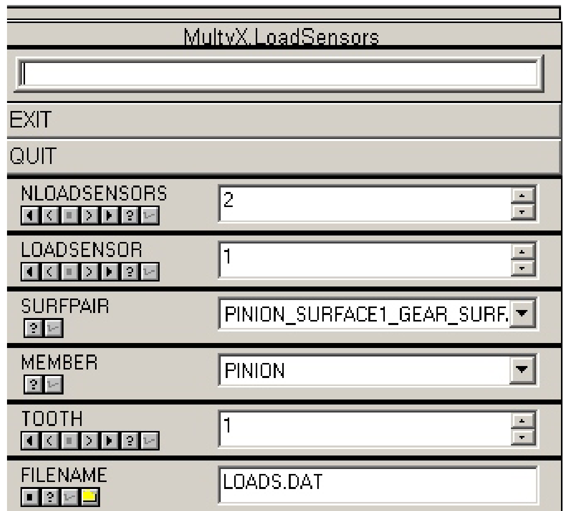


Figure 6.3: The load sensor menu

6.4 Specifying a contact grid

Figure 6.4 shows a computational grid that has been set up in the contact zone of a gear tooth. The entire face width of the tooth is divided into $2N + 1$ slices. N is a user selectable quantity (NFACEDIVS in Figure 6.9). If ζ is a parameter that goes from -1 at one end of the face width of a tooth to $+1$ at the other end, then the thickness of each slice in the ζ parameter space is $\Delta\zeta = 2/(2N + 1)$. For each slice $j = -N : +N$, a cross section of the tooth is taken at the middle of the slice, and a point is located on this slice that approaches the surface of the mating tooth the closest. This selection is carried out using the undeformed geometry. If the separation between the two gears at this closest point is larger than a user selectable separation tolerance (SEPTOL in Figure 6.9), then the entire gear slice is eliminated from further consideration. Otherwise, a set of grid cells identified by the grid cell location indices $(i, j), i = -M : M$ is set up centered around this closest point of slice j. The number M (NPROFDIVS in Figure 6.9) is user selectable. The dimension of the grid cells in the profile direction Δs (DSPROF in Figure 6.9) is also user selectable. Here s is the curve length parameter measured along the profile.

The number M is referred to as the number of grid cells in the profile direction (NPROFDIVS), and N is referred to as the number of grid cells in the face width direction (NFACEDIVS). Δs is referred to as the width of the grid cell in the profile direction (DSPROF). The width of the grid is $(2M + 1)\Delta s$. Choosing the correct width is crucial in obtaining correct contact pressures. Using too wide a grid for a fixed M can result in loss of resolution, because only the center grid cell will end up carrying all the load (Figure 6.5). If the grid is too narrow, then the contact zone will get truncated, causing artificially high contact pressures at the edges of the grid (Figure 6.6). If the grid size is correct, a variation of contact pressure similar to that in Figure 6.7 should be obtained.

Figure 6.8 shows an example of a contact grid set up on a pair of contacting teeth.

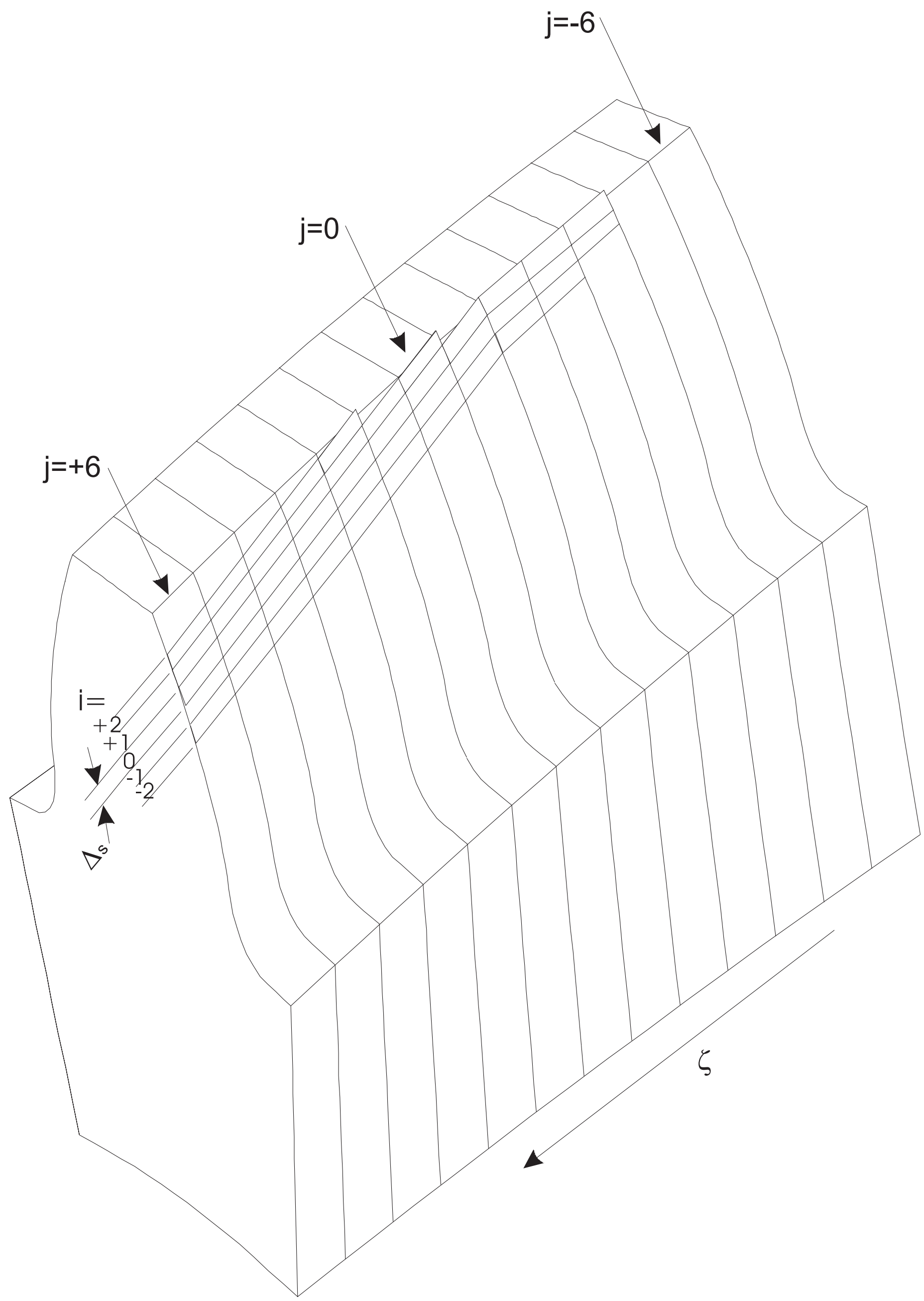


Figure 6.4: Computational grid in the contact zone of the gears

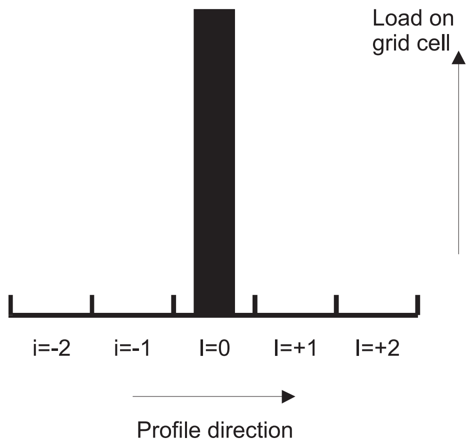


Figure 6.5: Contact pressure distribution across the width of contact obtained when the contact grid is too wide.

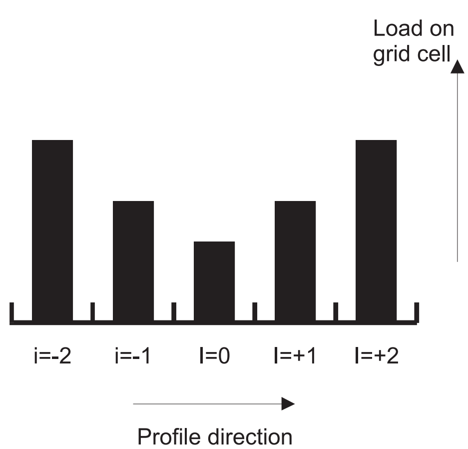


Figure 6.6: Contact pressure distribution across the width of contact obtained when the contact grid is too narrow.

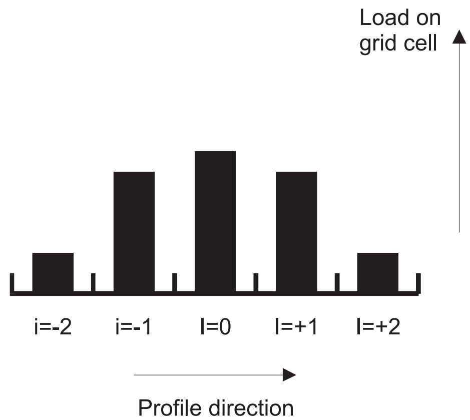


Figure 6.7: Contact pressure distribution across the width of contact obtained when the contact grid is correct.

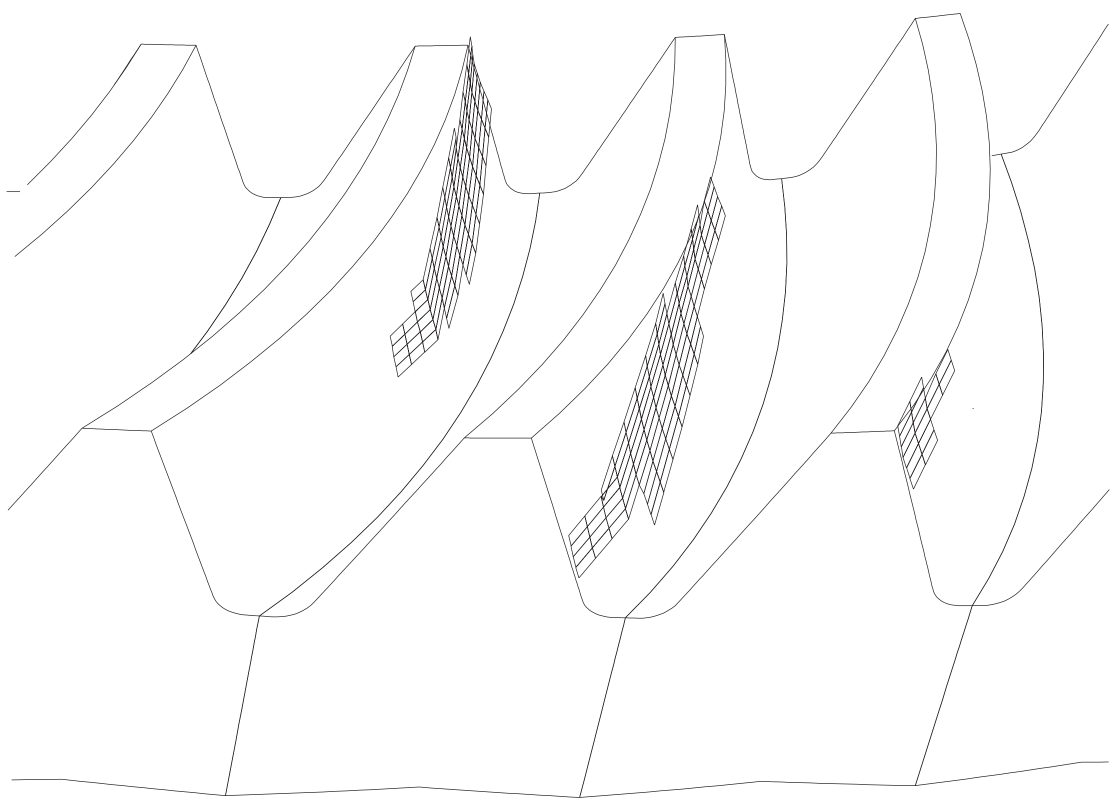


Figure 6.8: An example of a contact grid set up on a pair of contacting teeth.

6.5 The setup menu

Figure 6.9 shows the analysis setup menu accessed by using the SETUP command at the main menu. The parameters SEPTOL, NPROFDIVS, NFACEDIVS and DSPROF are the grid specification parameters described earlier. The initial state of the system can be specified as the undeformed state by enabling the ZEROINITIAL flag. The time at which to start the analysis is specified in the INITIALTIME box. If the ZEROINITIAL flag is not checked, then a restart file has to be specified, from which the deformed state and the value of time will be loaded. The analysis time is divided into a user-specified number NRANGES of time ranges. The time step DELTATIME, solution method SOLMETHOD and the number of time steps NTIMESTEPS can be specified separately for each time range.

It is possible to control the operating speed in each time range by specifying a speed factor at the beginning of the range by using the parameter STARTSPEEDFACTOR. A speed factor of 1.0 implies that the system is at its nominal speed. The speed factor at the end of a time range is the same as the speed factor at the beginning of the next time range. The speed at the end of the last range is always assumed 1.0. The speed is assumed to vary as a linear function of time within a time range.

The torque in a time range can be controlled by setting the STARTTORQUEFACTOR and ENDTORQUEFACTOR for each range. Again, a factor of 1.0 means that the system is operating at its nominal torque. The torque is assumed to vary as a linear function of time within a time range.

The SAVEPERIODICALLY option saves the state of the system in a restart file after every NSTEPSSAVE number of steps. The state is saved in the restart file named in the SAVEFILE-NAME box. This restart file can be used to restart another analysis. The OUTPUTRESTART option saves the state of the system in a restart file at the end of the analysis. The file named in the OUTPUTFILENAME box is used. This file can also be used to start a subsequent analysis.

Finally a finite element post-processing data file can be emitted once every NSTEPSWRITE number of time steps by enabling the POSTPROCWRITE option. The file used is selected in the POSTFILENAME box. The post-processing file can be used subsequently to make drawings and stress contour diagrams of the deformed system.

6.6 Other output files

Several tabular output files are created during the analysis.

The displacements and reaction forces generated by the reference frames of the individual bodies in the system are saved in data files during analysis. These data files are named after the bodies. The file `PINIONRES.DAT` contains the results for the pinion, `GEARRES.DAT` contains results for the gear. Each data file has one row for each instant of time analyzed. The first column contains the time. The next 6 columns contain the six components of reference frame deflection, u_x, u_y, u_z, θ_x, θ_y, and θ_z. The last 6 columns contain the 6 components of reference frame reaction, F_x, F_y, F_z, M_x, M_y, and M_z.

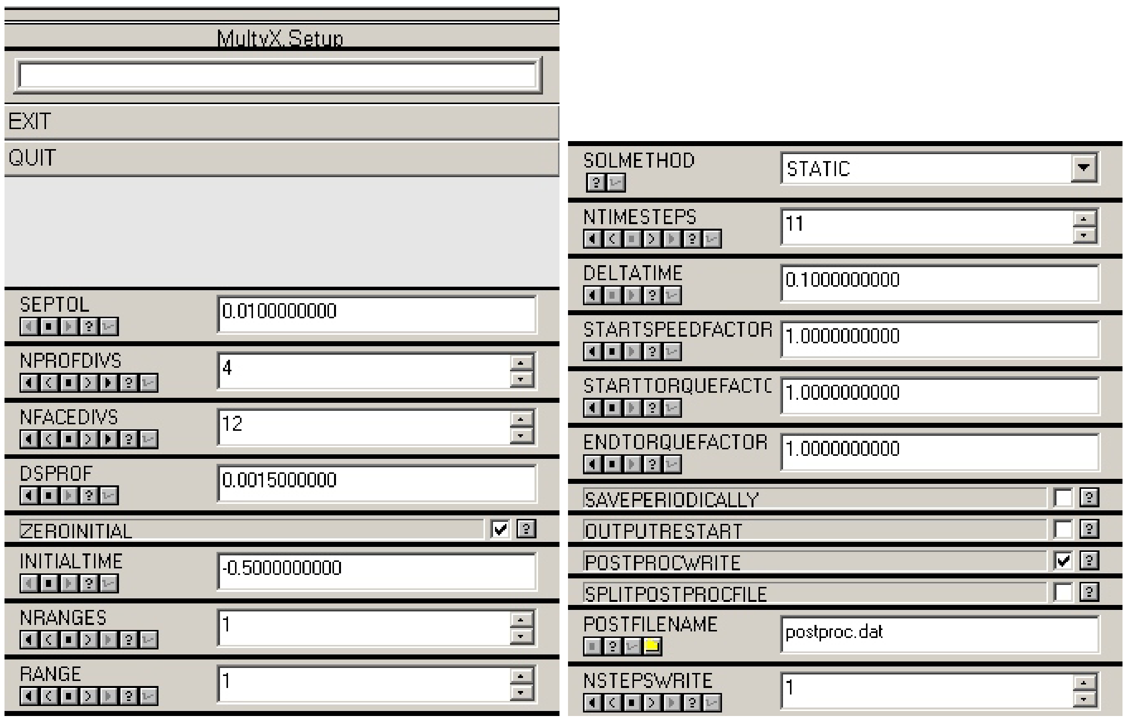


Figure 6.9: The setup menu

Chapter 7

Pre- and Post-processing

The PREPROC command in the main menu leads to the pre-processing menu shown in Figure 7.1. The POSTPROC command leads to the dialog box shown in Figure 7.2, where Multyx asks for the name of the post-processing data file created in the analysis step. When a valid name is entered, the post-processing menu shown in Figure 7.3 comes up.

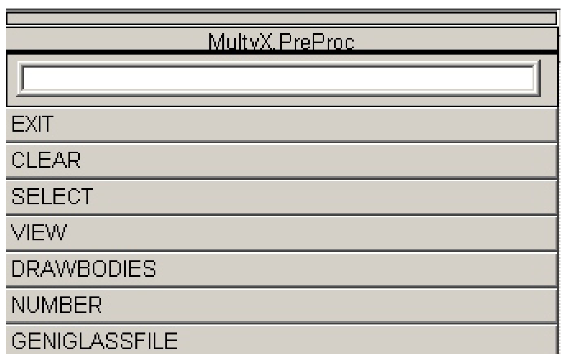


Figure 7.1: The pre-processing menu.

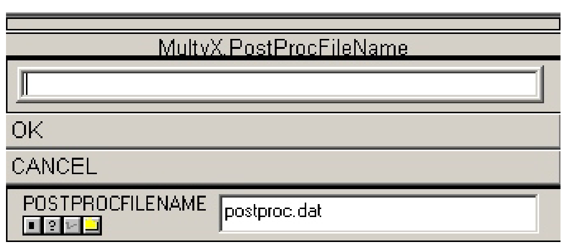


Figure 7.2: The post-processing file name dialog box.

The pre-processing menu and the post-processing menu are used to make drawings of the system and its components. The CLEAR command clears the graphics screen. The DRAWBODIES draws all the selected bodies using the current view settings. The DRAWBODIES command does not clear the screen before it makes the drawing. In the post-processing menu, the FIRSTPOSN, PREVPOSN, NEXTPOSN, and LASTPOSN commands allow the user to move from

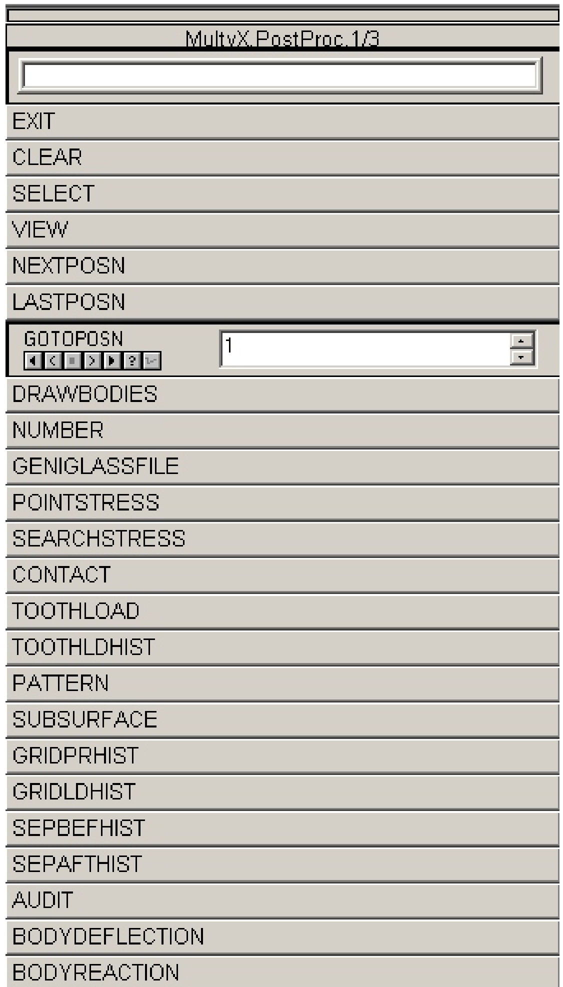


Figure 7.3: The post-processing menu.

one time step saved in the post-processing file to another. Entering a position number directly in the GOTOPOSN box takes the user directly to that time step.

7.1 Selecting bodies

The object selection menu which appears when the SELECT command is invoked from the pre- and post-processing menus is shown in Figure 7.4. The objects that should be drawn are selected from this menu.

Figure 7.4: The body selection menu.

7.2 View parameters

The VIEW menu controls the appearance of the drawings. In the pre-processing view menu shown in Figure 7.5, the user can enter any value of time into the TIME box. The next drawing will show the system as it would appear at this instant of time. The resolution level controls the degree of detail with which the drawing is made. The ELEMENTS checkbox controls whether or not the individual finite elements should be drawn. The COLORS option controls whether or not the bodies will be filled with color. In pre-processing mode, all bodies are painted Gray. The OUTLINE box controls whether or not an outline drawing of the body will be made.

The view menu in post-processing mode (Figure 7.6) has a few additional parameters. There is a CONTOURS option to draw stress contours. If the COLORS or CONTOURS option is selected, then the menu also asks for the values of the lowest contour level MINSTRESS and the highest contour level MAXSTRESS. The colors used in the drawing are based on the stress level. If the LOADS option is selected, then the contact loads acting on the components will be drawn using the scale factor entered in the LOADSCALE box (Figure 7.7).

If the LOADS option is not checked (Figure 7.6), then an additional box EXAGGERATION appears where an exaggeration factor can by entered for deformed geometry plots. An exaggeration factor of 0.0 will draw the bodies in their undeformed state.

The axes of rotation are aligned with the screen axes. So if you enter a particular angle for any of the LEFTROTATE, RIGHTROTATE, UPROTATE, DOWNROTATE, CWROTATE, CCWROTATE items then the model is rotated by that angle with respect to the screen axes. The value in all the rotate items is always going to be 0 degrees after a change in the rotate angle is applied. For instance if you enter 30 in the LEFTROATE box and press enter (on Keyboard) you will see 0 in the box. This means that the program has incorporated that change of 30 degrees rotation in the model even though you are not able to visualise that change. To see the new rotated drawing you will have to clear the one present on your screen and click on the DRAWBODIES button in the pre and the postprocessing menus.

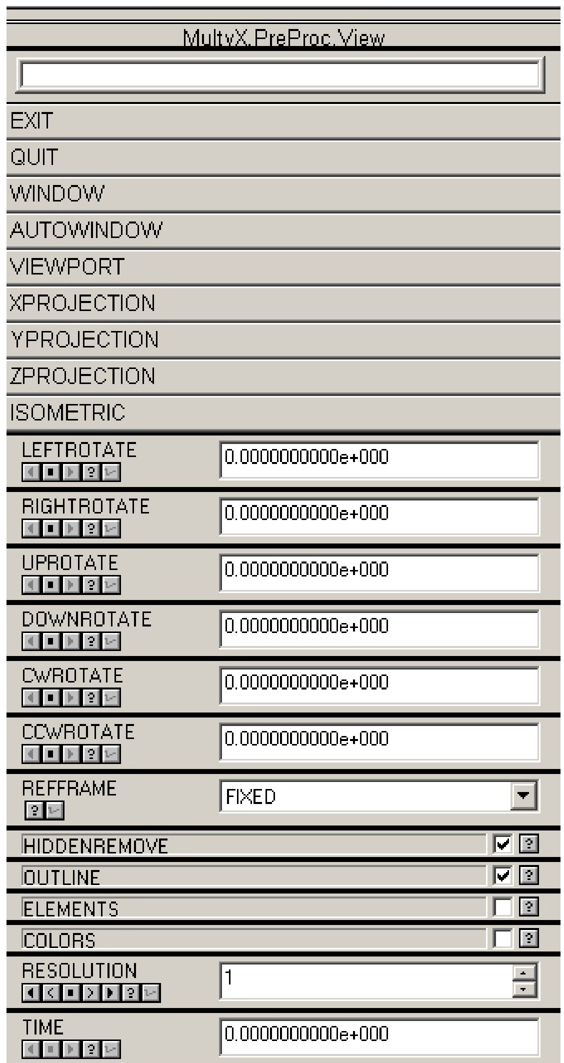


Figure 7.5: The view menu in pre-processing mode.

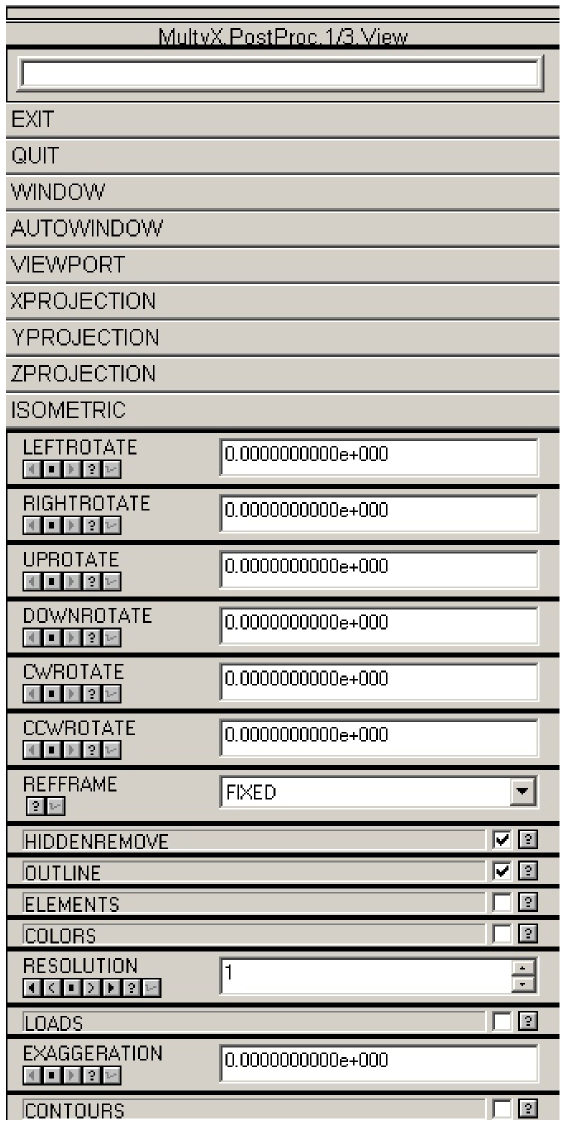


Figure 7.6: The view menu in post-processing mode with the LOADS option disabled.

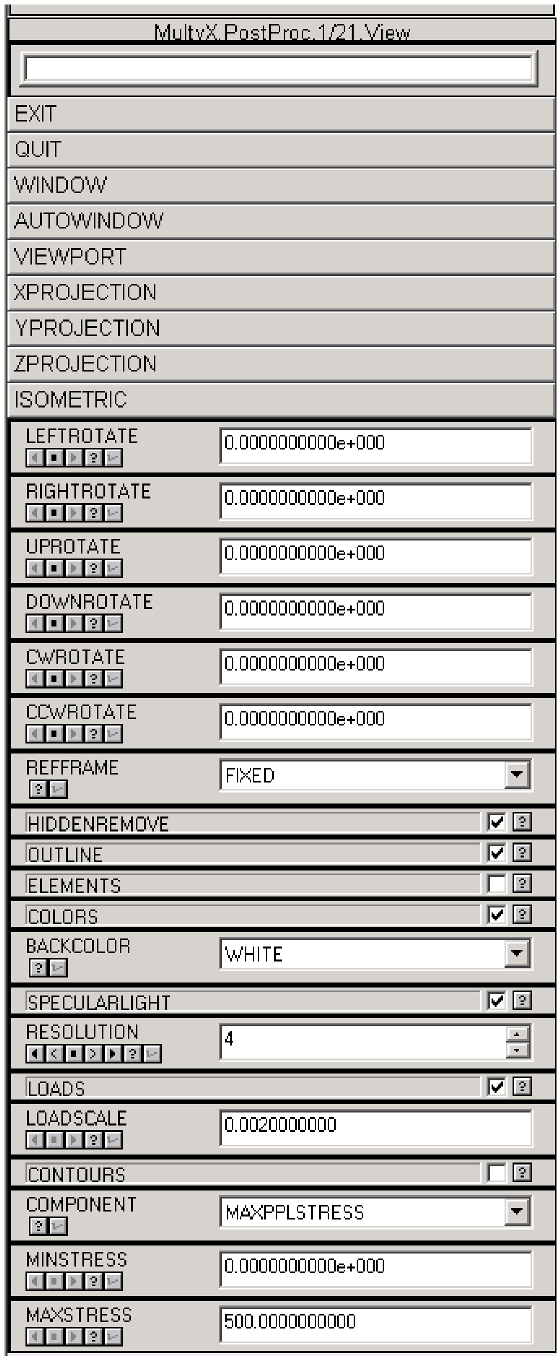


Figure 7.7: The view menu in post-processing mode with the LOADS option enabled.

7.3 The DRAWBODIES command

After an appropriate view and objects have been selected, the DRAWBODIES command in the pre- and post-processing menus (Figures 7.1 and 7.3) will generate a drawing. Figures 7.8 and 7.9 show examples of drawings generated Multyx in the post-processing mode.

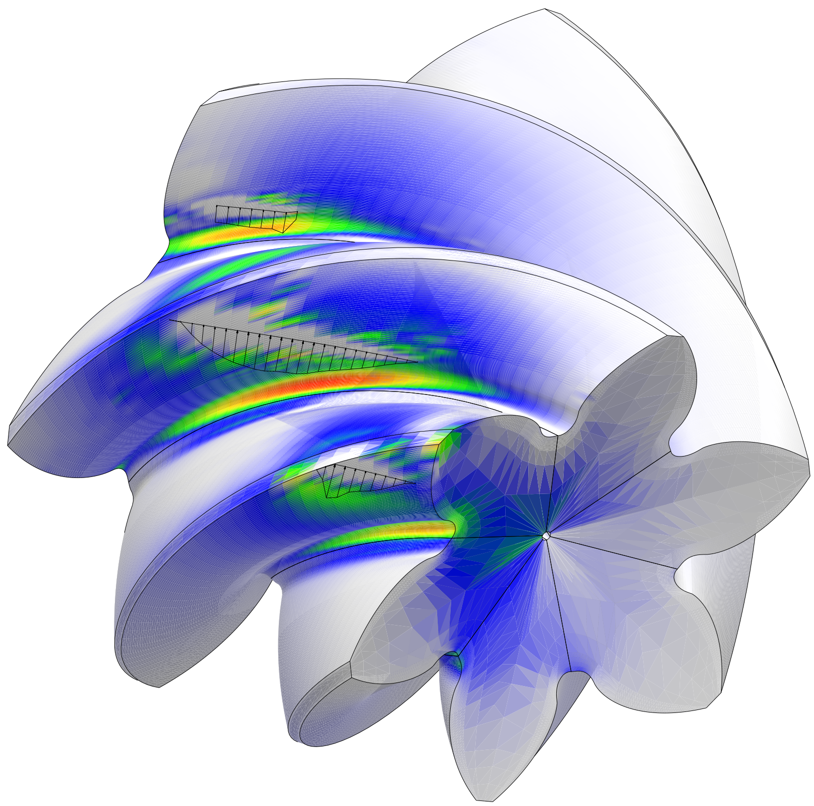


Figure 7.8: An example of drawing made in the post-processing mode.

7.4 The NUMBER command

The NUMBER command in the pre- and post-processing menus (Figures 7.1 and 7.3) lead to the numbering menu shown in Figure 7.10. This menu is used to to generate tooth and surface numbering, as shown in Figure 7.11.

7.5 The TOOTHLOAD command

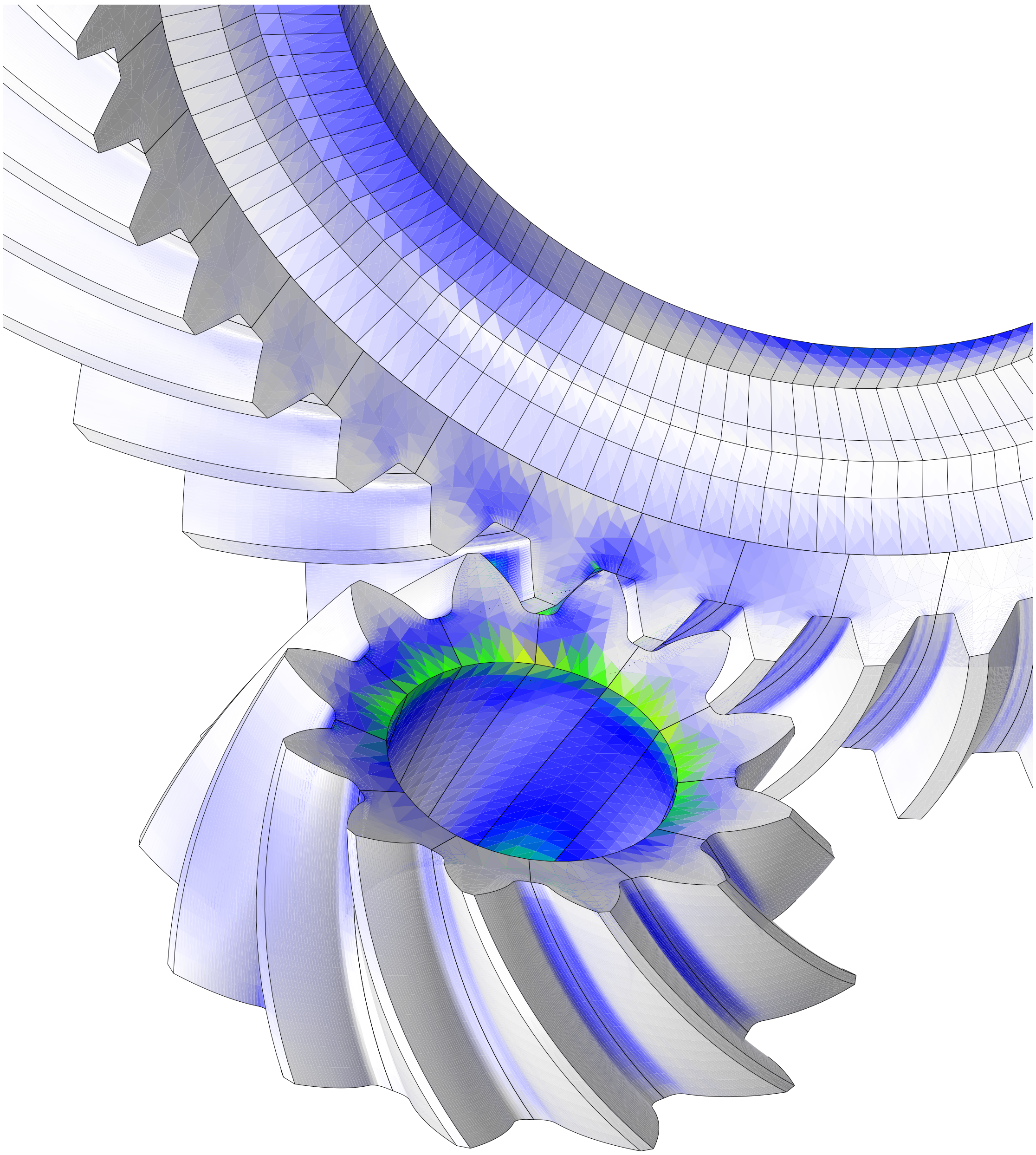


Figure 7.9: An example of drawing made in the post-processing mode.

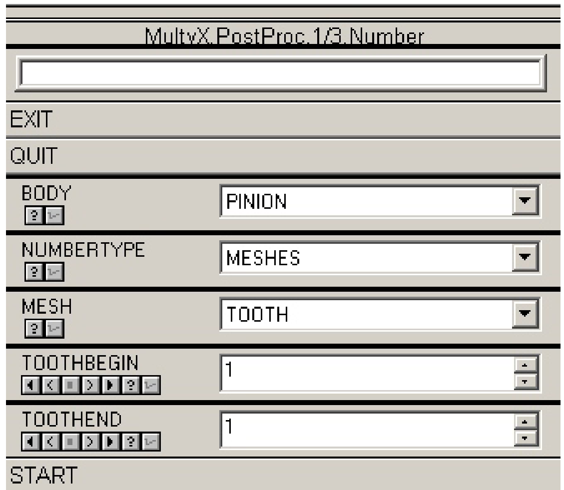


Figure 7.10: The NUMBER menu.

The TOOTHLOAD command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.12. This menu is used to to generate a graph of tooth load vs. time. The SURFACEPAIR item selects the contact surface pair for which the load is of interest. Each surface pair has two contacting members or bodies. The MEMBER parameter selects one of these two bodies, and the TOOTHBEGIN and TOOTHEND items select a range of instance numbers (or tooth numbers) within that body. If TOOTHBEGIN is greater than TOOTHEND, then the range wraps around the last tooth of the surface. This range must contain 7 teeth or less.

BEGINSTEP and ENDSTEP are used to select a range of time steps for which results have been stored in the post-processing file. Figure 7.13 shows a graph of tooth load vs. time generated by the TOOTHLOAD command.

The OUTPUTFILENAME item is used to write the tooth load data into an ASCII file. The name of the ASCII file is entered into the item OUTPUTFILENAME. If the APPEND box is checked, and if this file already exists, then the data is appended at the end of the file. Otherwise a new file is created.

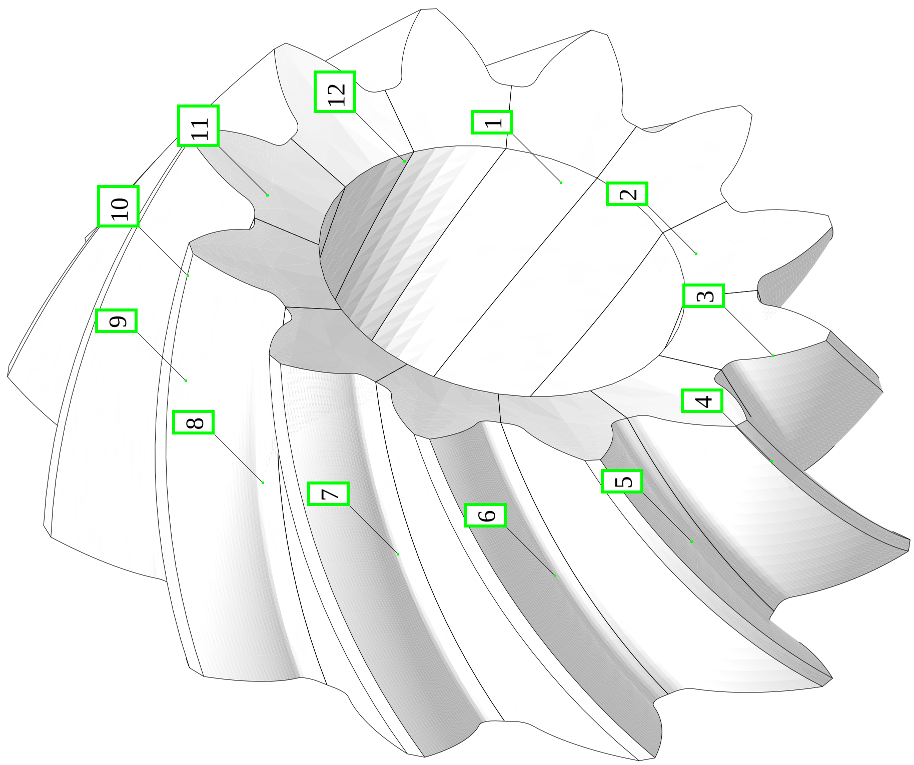


Figure 7.11: Surface numbering superimposed on a pinion drawing using the NUMBER command.

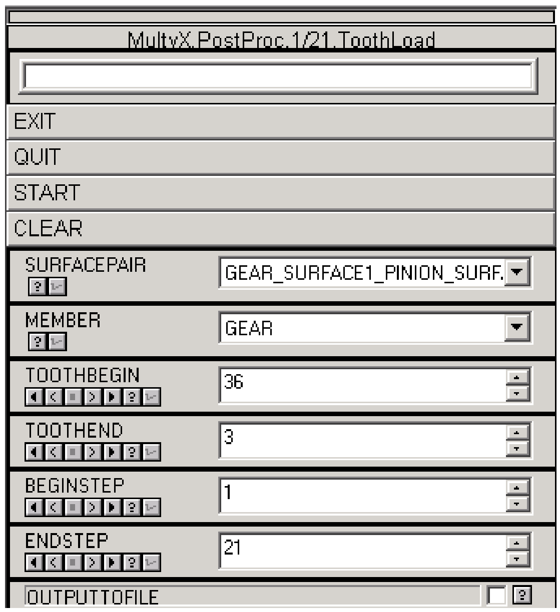


Figure 7.12: The TOOTHLOAD menu.

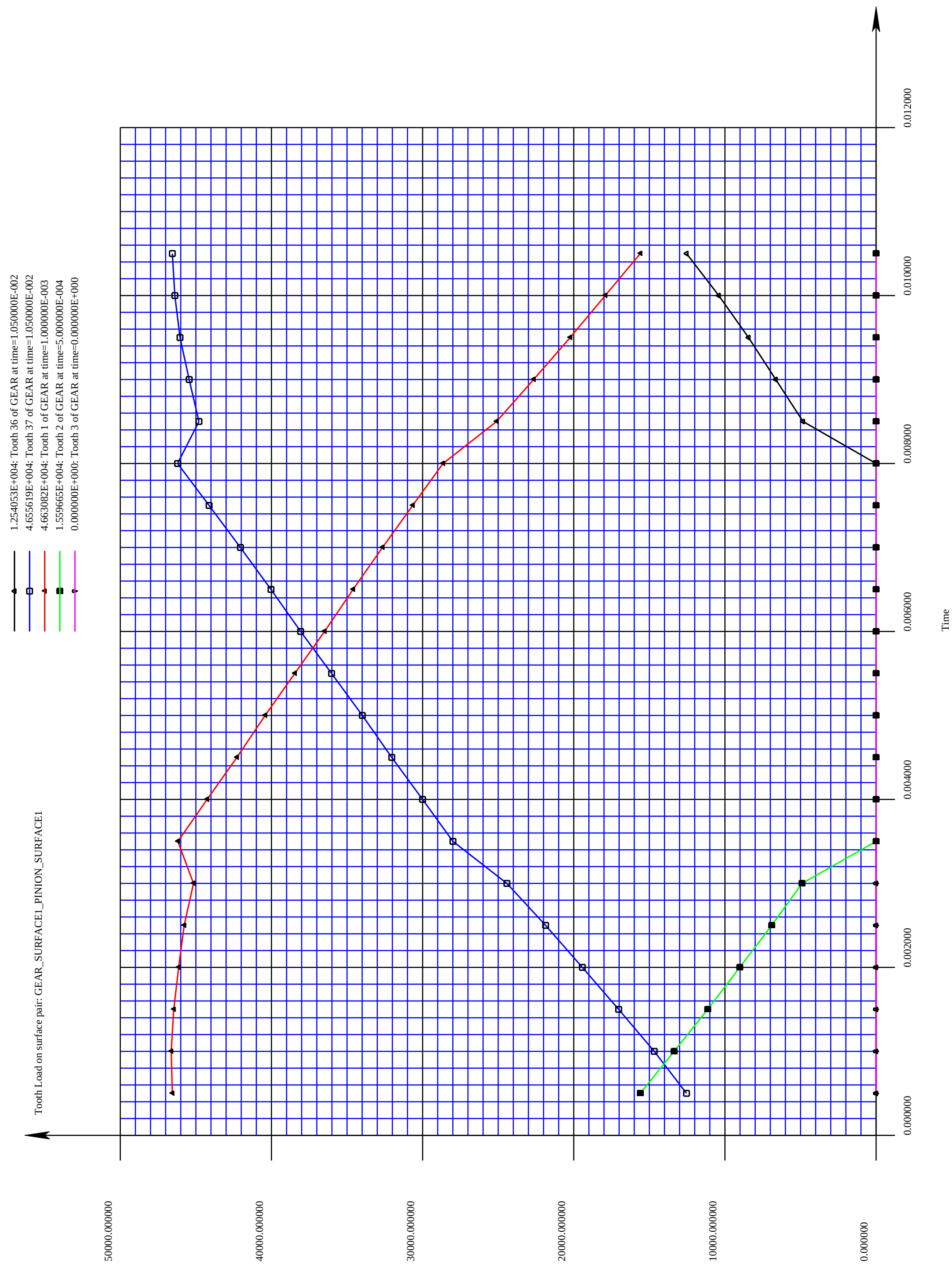


Figure 7.13: The tooth load vs. time graph generated by the TOOTHLOAD menu.

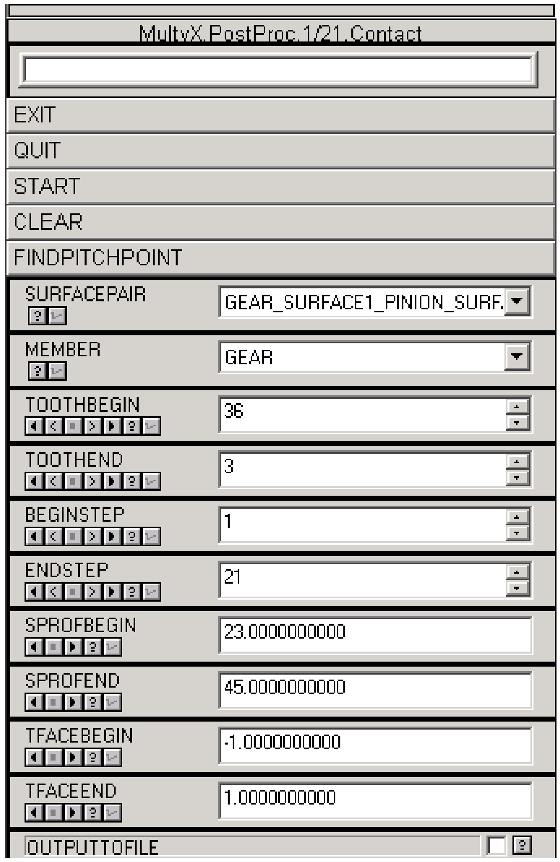


Figure 7.14: The CONTACT menu.

7.6 The CONTACT command

The CONTACT command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.14. This menu is used to generate a graph of contact pressure vs. time.

The SURFACEPAIR item selects the contact surface pair for which the pressure is of interest. Each surface pair has two contacting members or bodies. The MEMBER parameter selects one of these two bodies, and the TOOTHBEGIN and TOOTHEND items select a range of instance numbers (or tooth numbers) within that body. If TOOTHBEGIN is greater than TOOTHEND, then the range wraps around the last tooth of the surface. This range must contain 7 teeth or less. BEGINSTEP and ENDSTEP are used to select a range of time steps for which results have been stored in the post-processing file. The items SPROFBEGIN, SPROFEND, TFACEBEGIN and TFACEEND are used to restrict the search to a part of the contact surface. Contact occurring outside this range is not considered for display in this graph.

Figure 7.15 shows the graph of contact pressure vs. time over the entire surface of the gear tooth. When the contact runs all the way to the tip of the gear and/or pinion, very high contact pressures. This high contact pressure near the edges can be filtered out by restricting the search range using the SPROFBEGIN and SPROFEND commands.

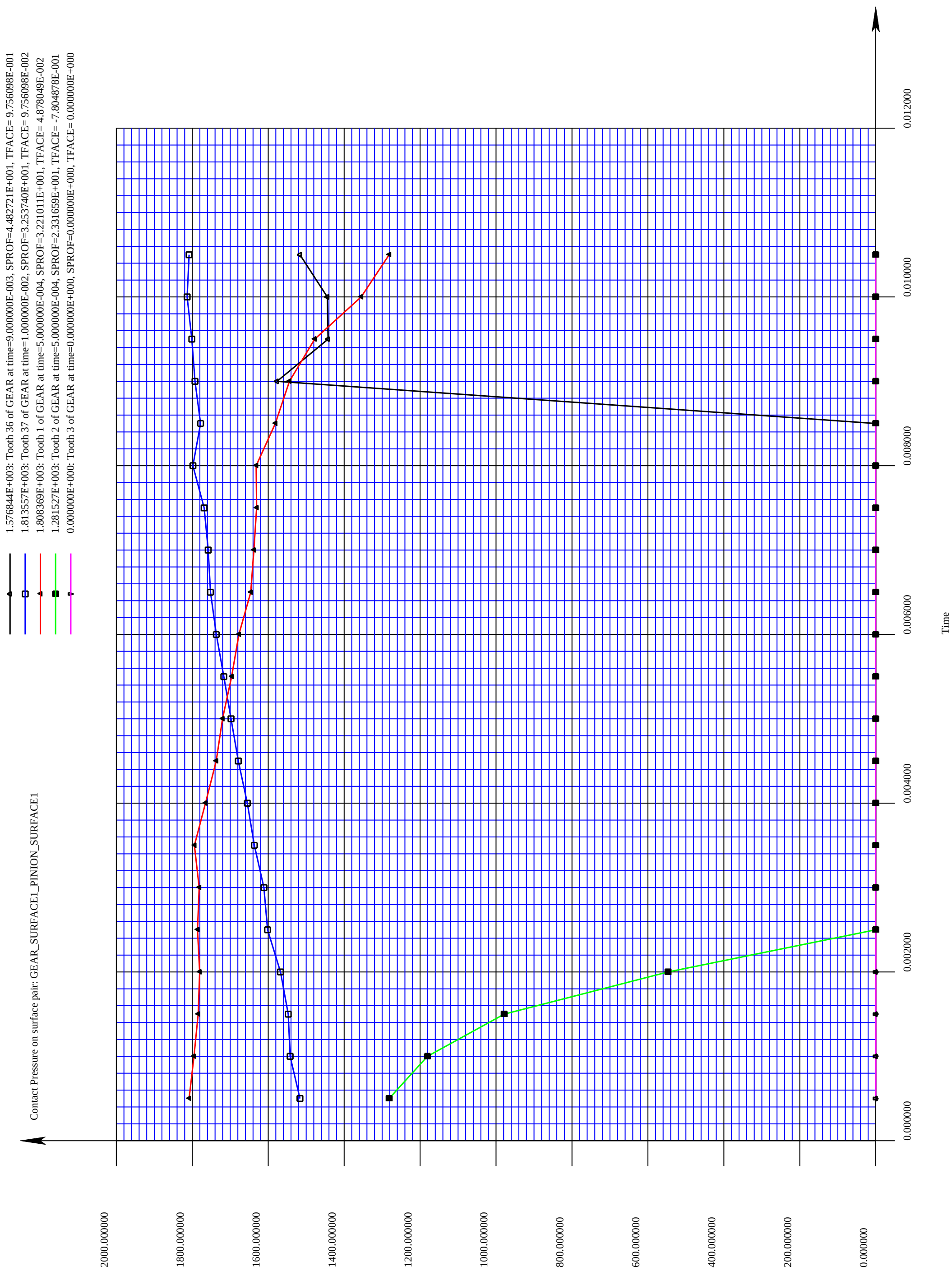


Figure 7.15: The tooth contact pressure vs. time graph generated by the CONTACT menu.

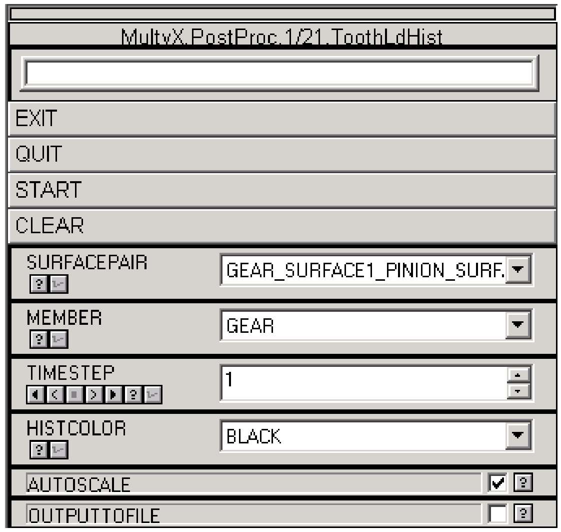


Figure 7.16: The TOOTHLDHIST menu.

7.7 The TOOTHLDHIST command

The TOOTHLDHIST command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.16. This menu is used to to generate a histogram of tooth loads at the different teeth in the pinion or gear at a particular time step. The SURFACEPAIR item selects the surface pair, and the MEMBER parameter selects one of the two bodies in this pair. The time step number is selected by the TIMESTEP item. If the AUTOSCALE box is checked, then the vertical scale is automatically computed. Otherwise the user can specify a maximum load value to be used for scaling the vertical axis. The color of the histogram is specified in the HISTCOLOR item. An example of a tooth load histogram is shown in Figure 7.17.

7.8 The SUBSURFACE command

The SUBSURFACE command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.18. This menu is used to to generate a graph of subsurface stresses vs. depth under the most critical point in the contact zone. The SURFACEPAIR item selects the surface pair, and the MEMBER parameter selects one of the two bodies in this pair. The time step number is selected by the TIMESTEP item

The items TOOTHBEGIN and TOOTHEND are used to select a range of surface instances (tooth numbers). There can be at most 7 teeth in this range.

The items DEPTHBEGIN and DEPTHEND define a depth range, and NUMDEPTH specifies the number of points over this range. Very close to the surface, the subsurface stresses have a large error because of the concentrated nature of the load as shown in Figure 7.19. So DEPTHBEGIN should never be set to zero.

The stress component is selected in the COMPONENT box. Options available are MAXPPLNORMAL (the maximum principal normal stress s_1), MINPPLNORMAL (the minimum principal normal stress s_3), MAXSHEAR (the maximum shear stress τ_{max}) and VONMISES (the Von Mises' octahedral shear stress s_{VM}).

Figure 7.20 shows an example of a graph of sub-surface stress vs. depth.

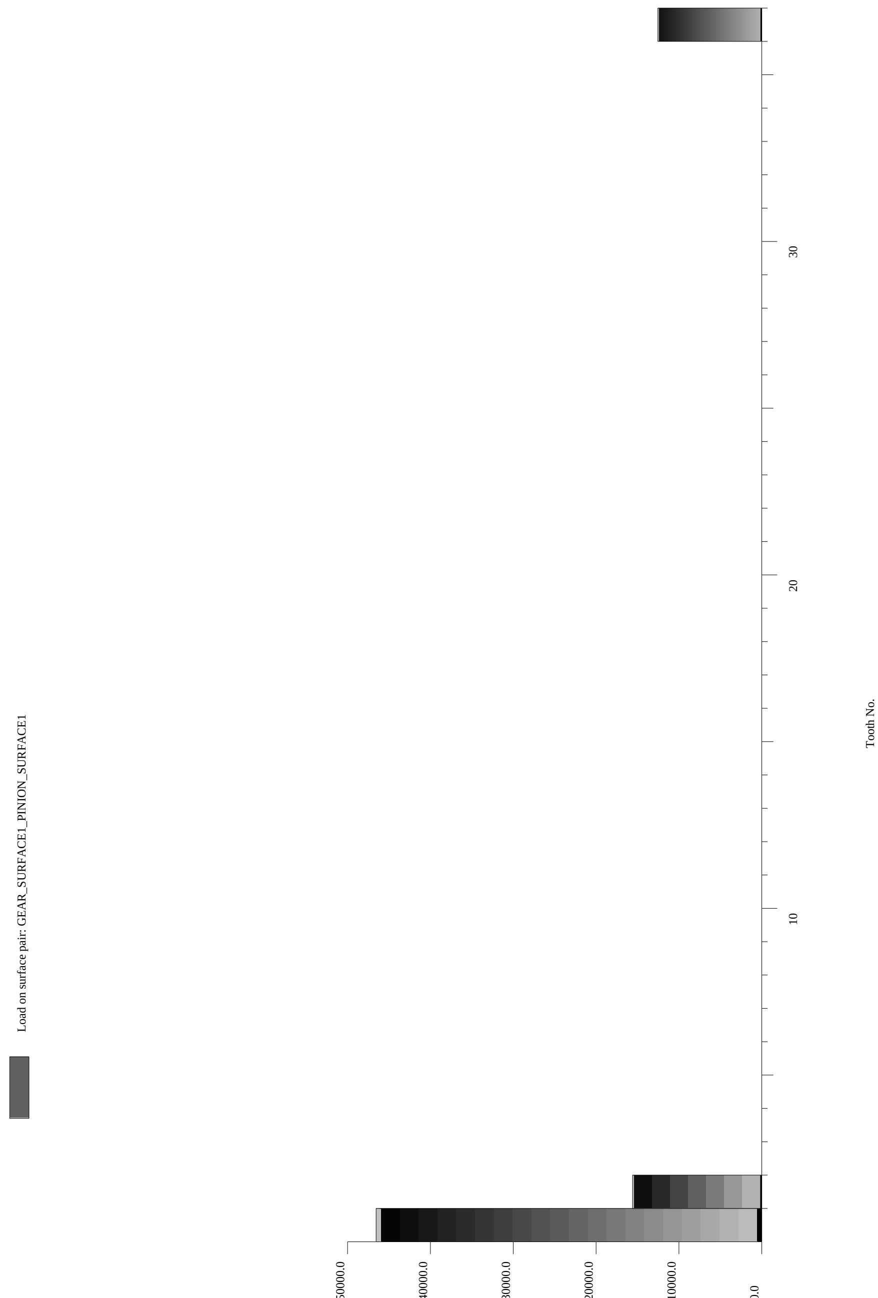


Figure 7.17: The tooth load histogram generated by the TOOTHLDHIST menu .

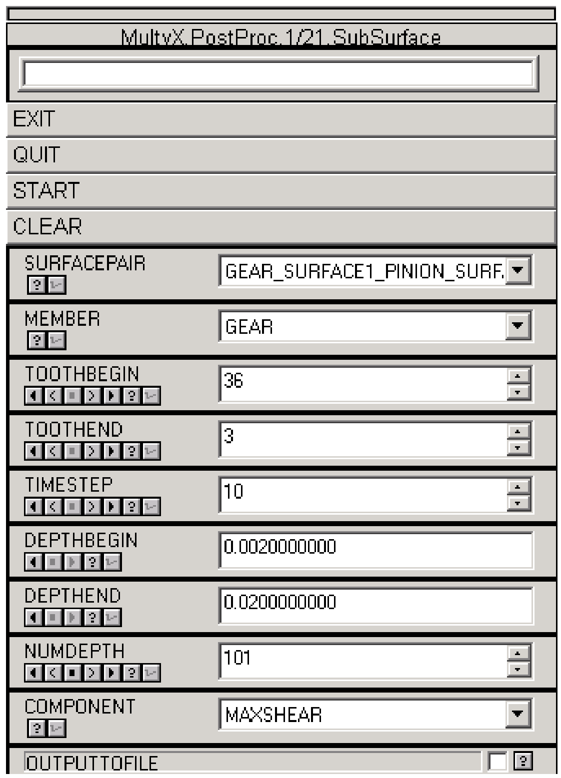


Figure 7.18: The SUBSURFACE menu.

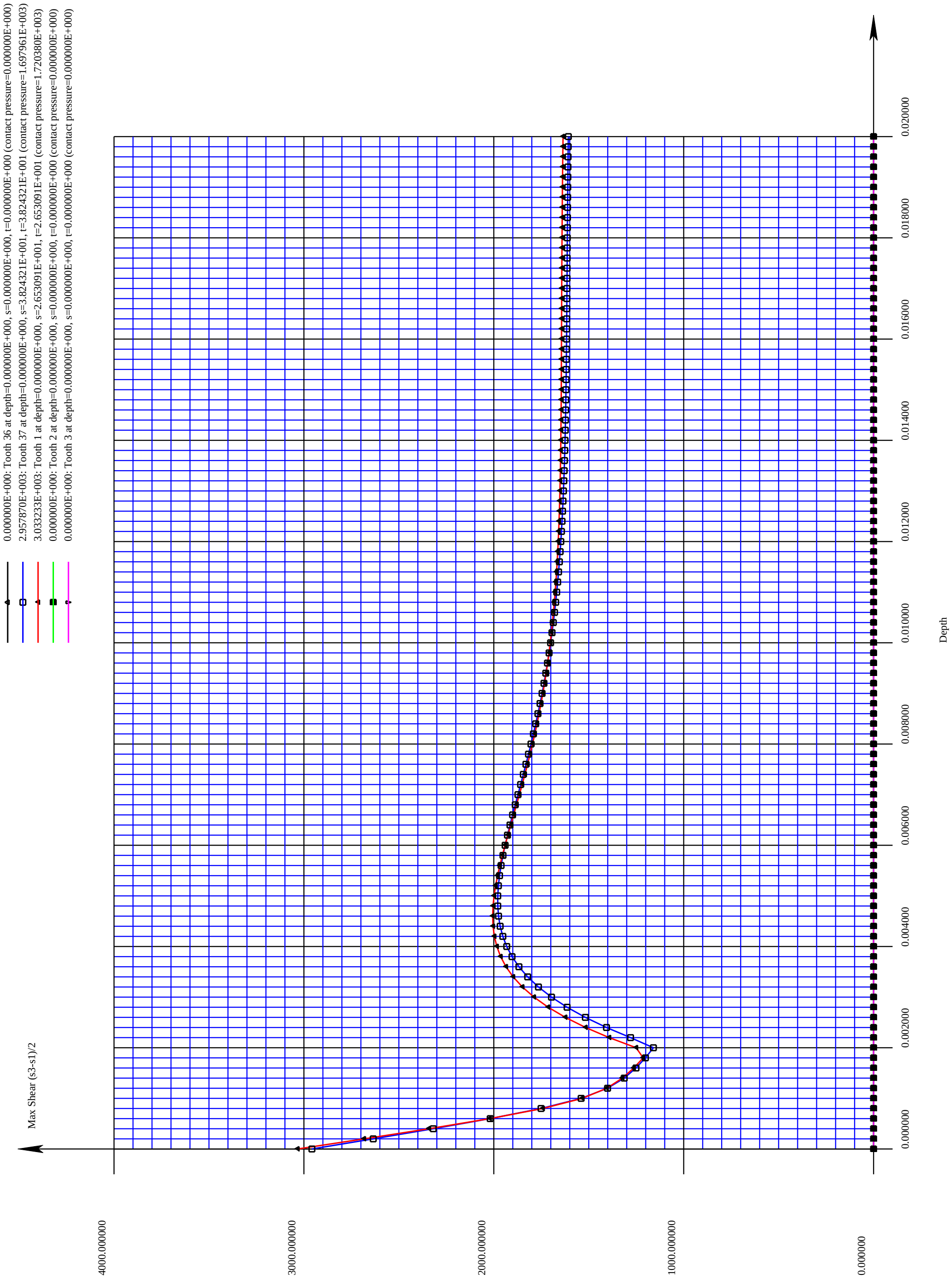


Figure 7.19: The subsurface shear graph generated by the SUBSURFACE menu showing large errors when DEPTHBEGIN = 0

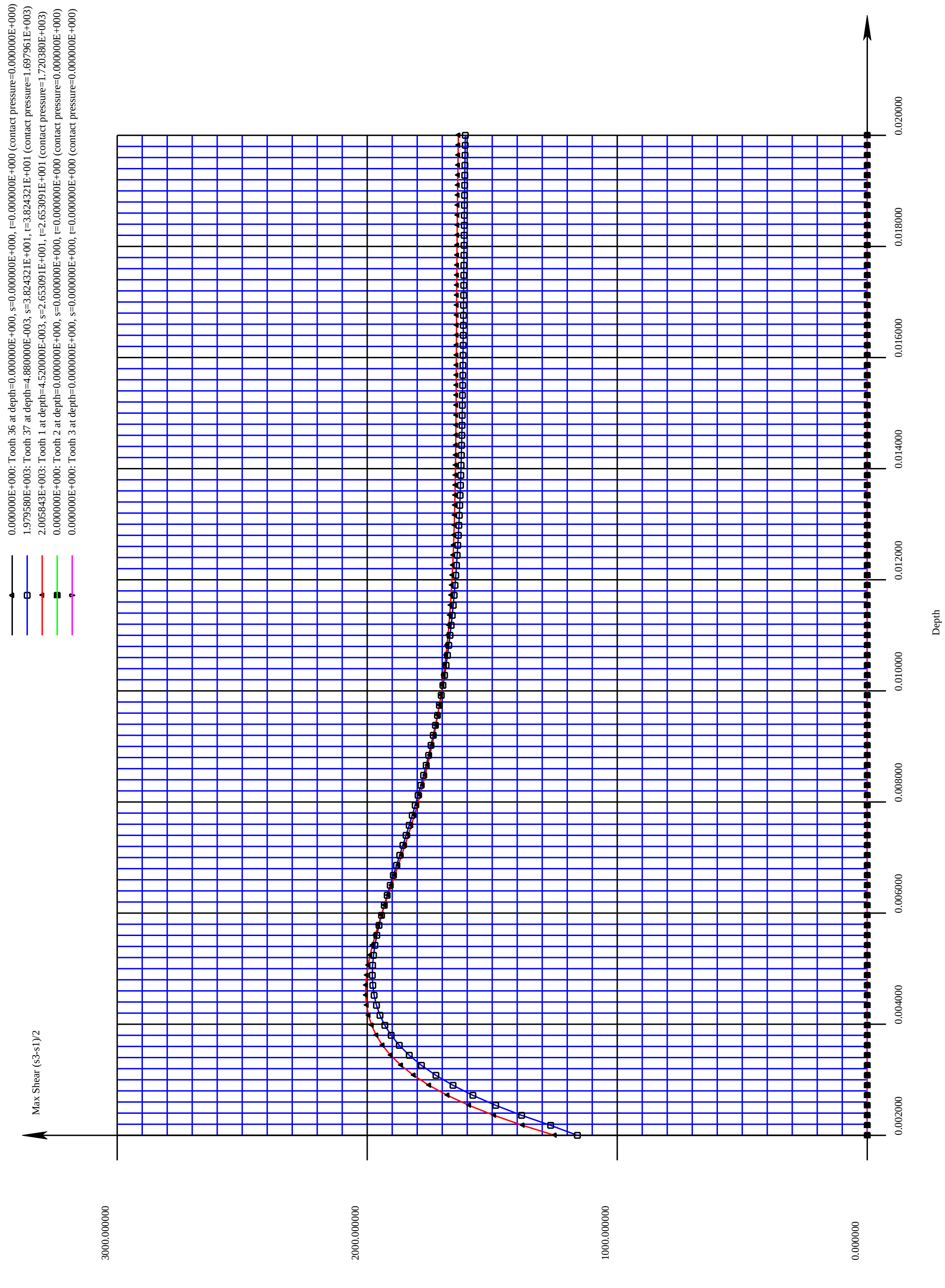


Figure 7.20: The subsurface shear graph generated by the SUBSURFACE menu

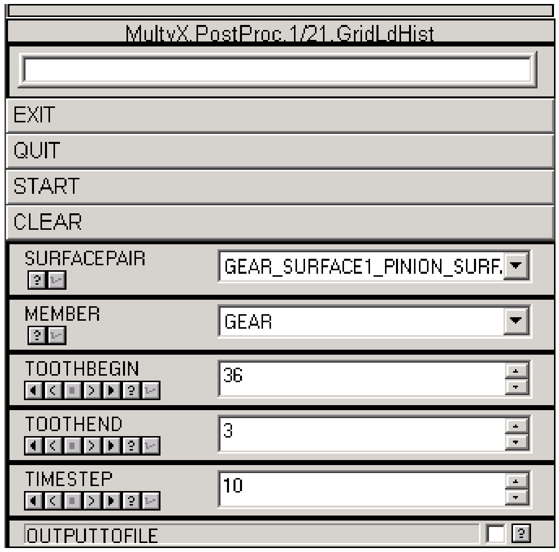


Figure 7.21: The GRIDLDHIST menu.

7.9 The GRIDLDHIST command

The GRIDLDHIST command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.21. This menu is used to generate a histogram of the distribution of contact load over individual contact grid cells. This figure is useful in determining whether the contact grid cell has been properly sized, and whether it has adequate resolution.

The SURFACEPAIR item selects the surface pair, and the MEMBER parameter selects one of the two bodies in this pair. The items TOOTHBEGIN and TOOTHEND are used to select a range of surface instances (tooth numbers). There can be at most 7 teeth in this range. The item TIMESTEP selects a time step number.

Figure 7.22 shows an example of a grid load histogram.

7.10 The GRIDPRHIST command

The GRIDPRHIST command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.23. This menu is used to to generate a histogram of the distribution of contact pressure over individual contact grid cells. This command is very similar to the GRIDLDHIST command. The only difference is that it uses contact pressure instead of contact load.

Figure 7.24 shows an example of a grid pressure histogram.

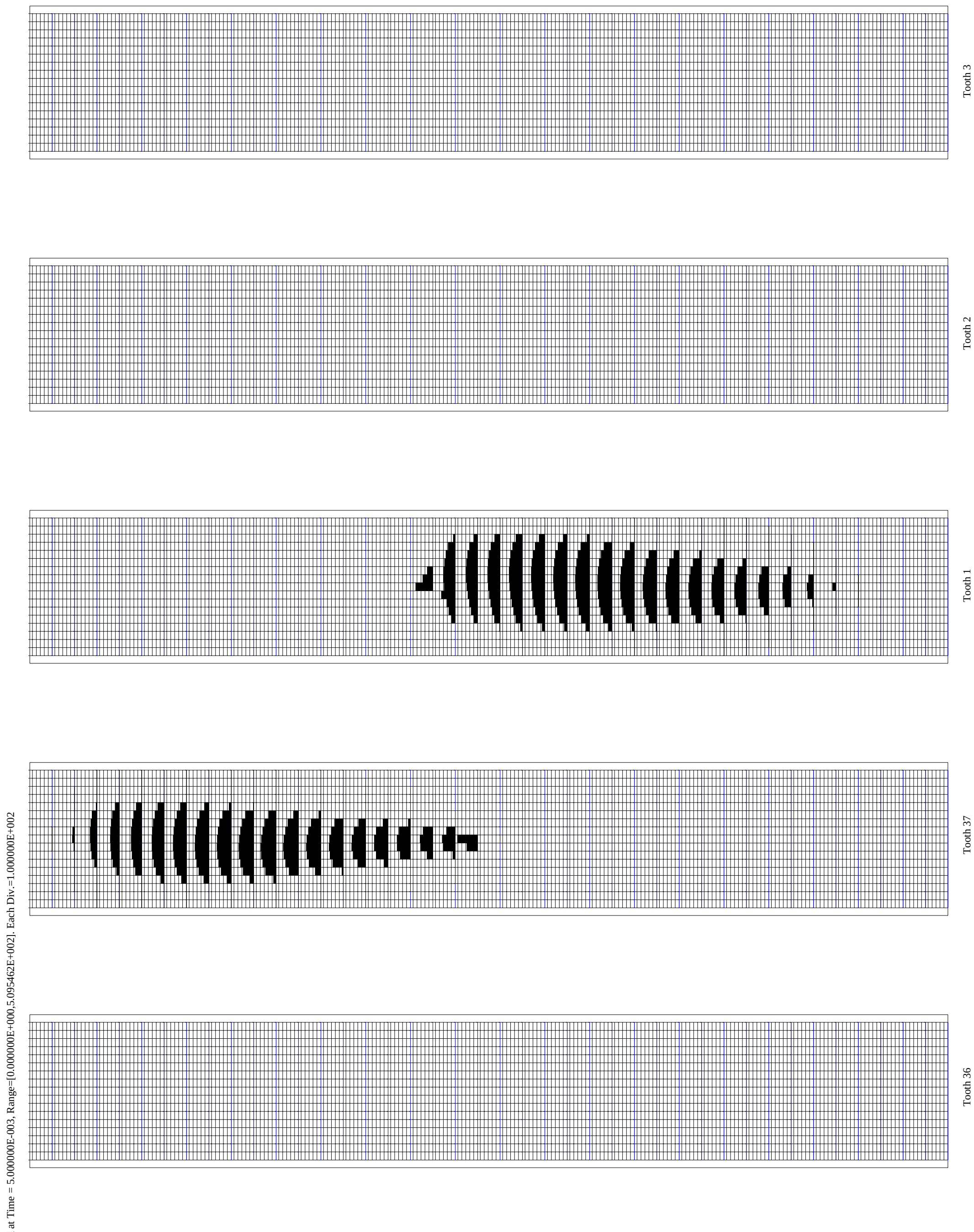


Figure 7.22: The grid load histogram generated by the GRIDLDHIST menu.

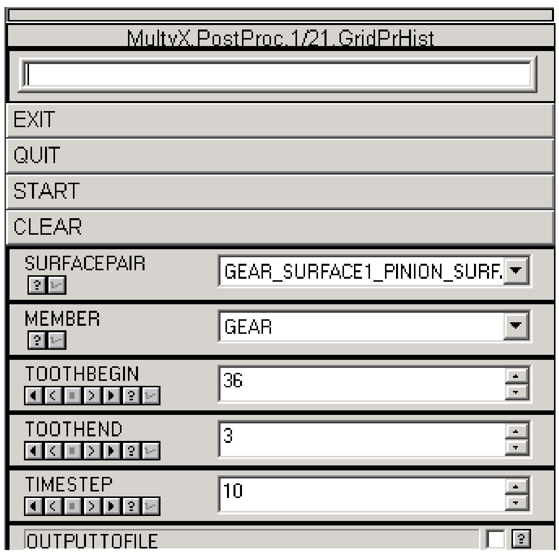


Figure 7.23: The GRIDPRHIST menu.

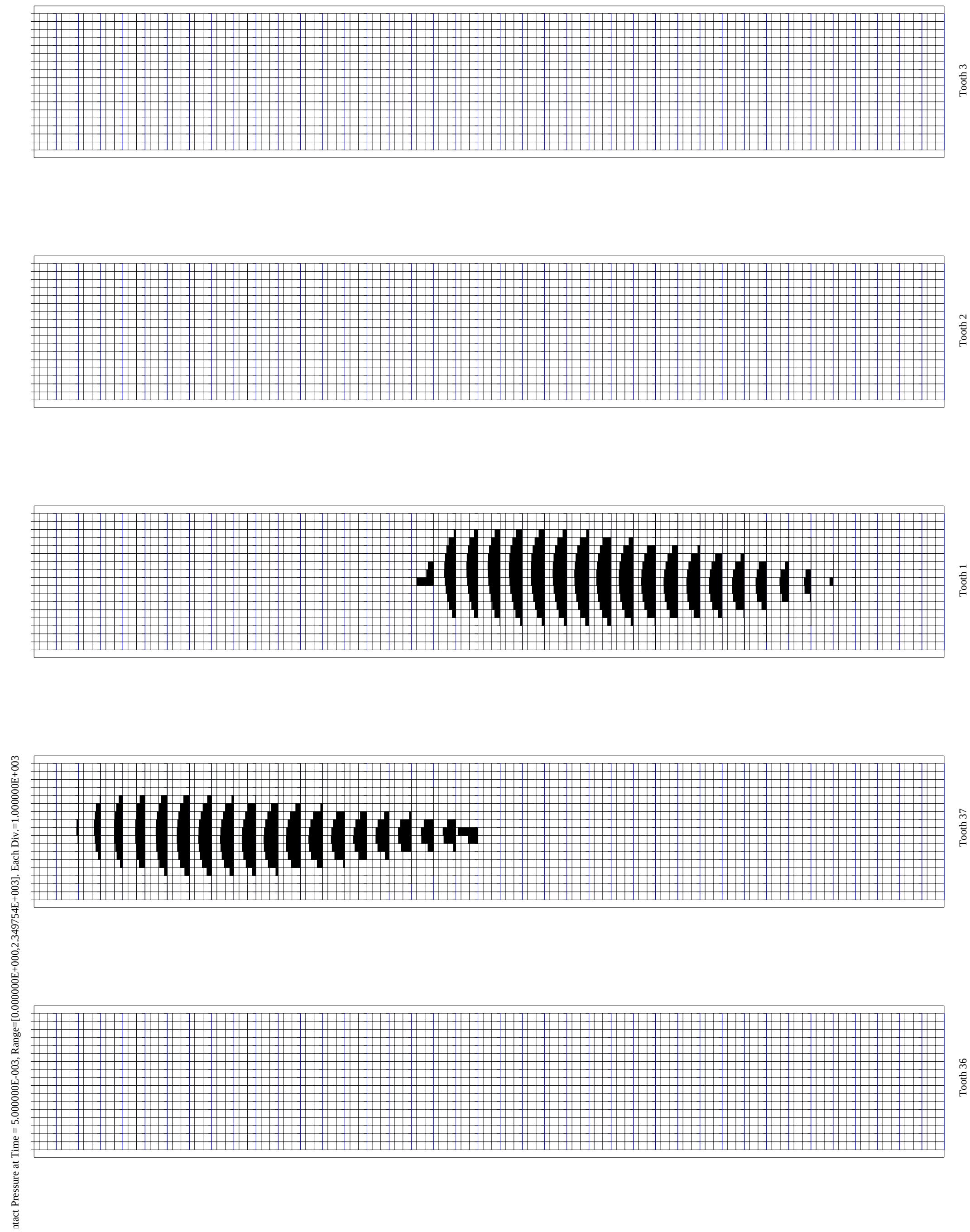


Figure 7.24: The grid pressure histogram generated by the GRIDPRHIST menu.

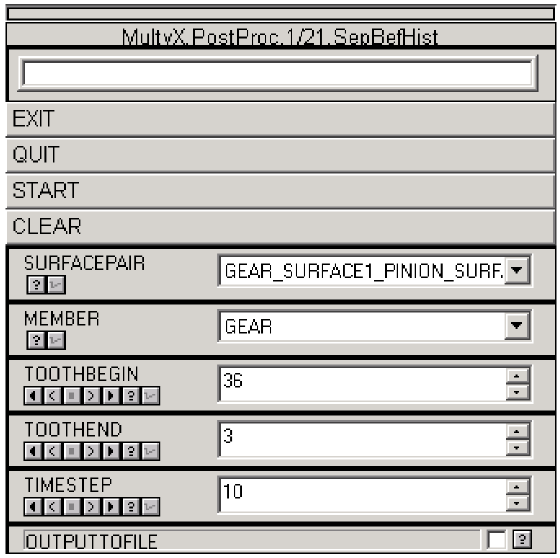


Figure 7.25: The SEPBEFHIST menu.

7.11 The SEPBEFHIST command

The SEPBEFHIST command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.25. This menu is used to to generate a histogram of the distribution of normal separation over individual contact grid cells, in the unloaded and undeformed state. The SURFACEPAIR item selects the surface pair, and the MEMBER parameter selects one of the two bodies in this pair. The items TOOTHBEGIN and TOOTHEND are used to select a range of surface instances (tooth numbers). There can be at most 7 teeth in this range. The item TIMESTEP selects a time step number.

Figure 7.26 shows an example of a histogram of separation in the unloaded state. Negative separation values are possible in this histogram.

7.12 The SEPAFTHIST command

The SEPAFTHIST command in the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.27. This menu is used to to generate a histogram of the distribution of normal separation over individual contact grid cells, in the loaded and deformed state. The SURFACEPAIR item selects the surface pair, and the MEMBER parameter selects one of the two bodies in this pair. The items TOOTHBEGIN and TOOTHEND are used to select a range of surface instances (tooth numbers). There can be at most 7 teeth in this range. The item TIMESTEP selects a time step number.

Figure 7.28 shows an example of a histogram of separation in the loaded state. These separation values must be either zero or positive.

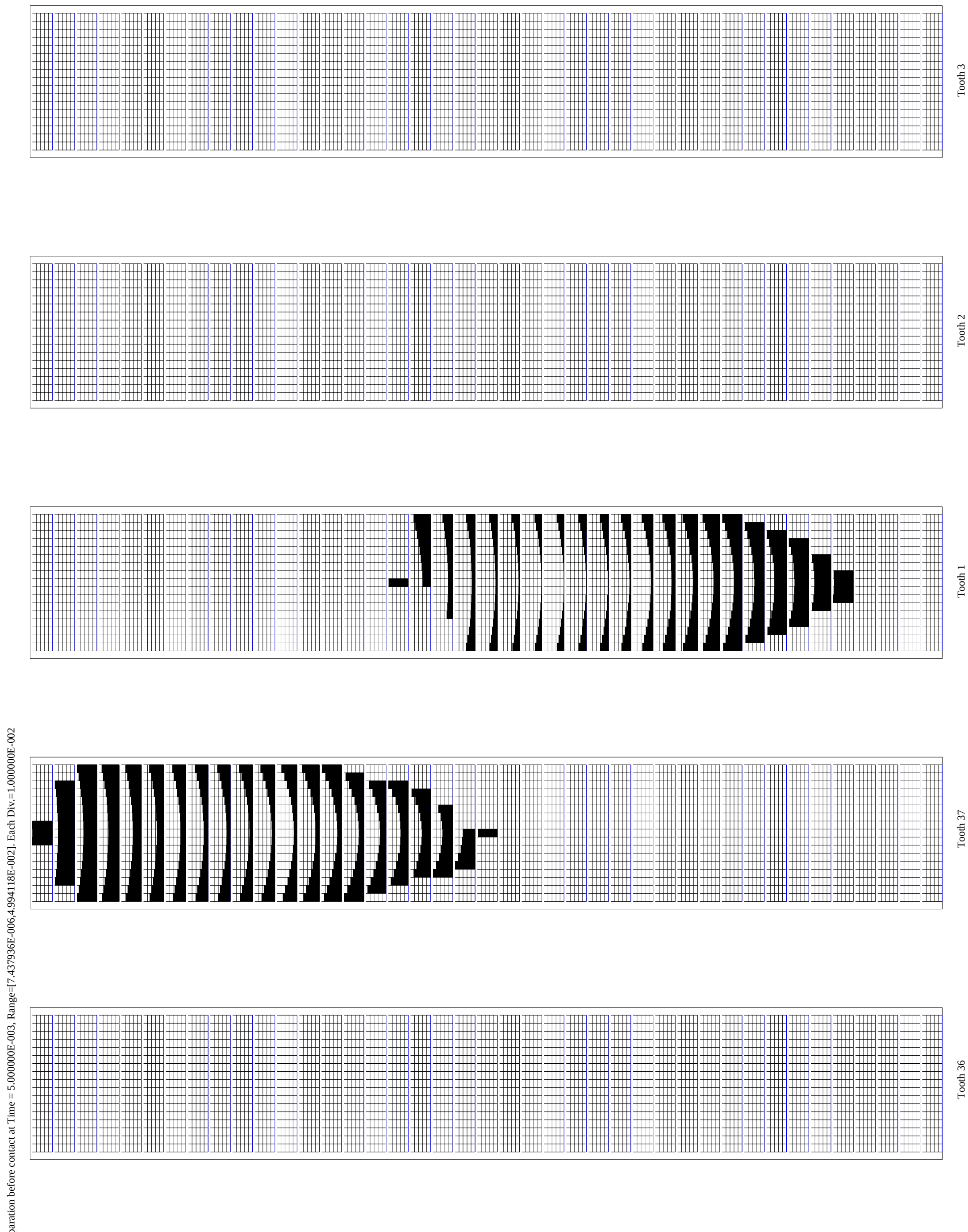


Figure 7.26: The histogram of grid separation before contact, generated by the SEPBEFHIST menu.

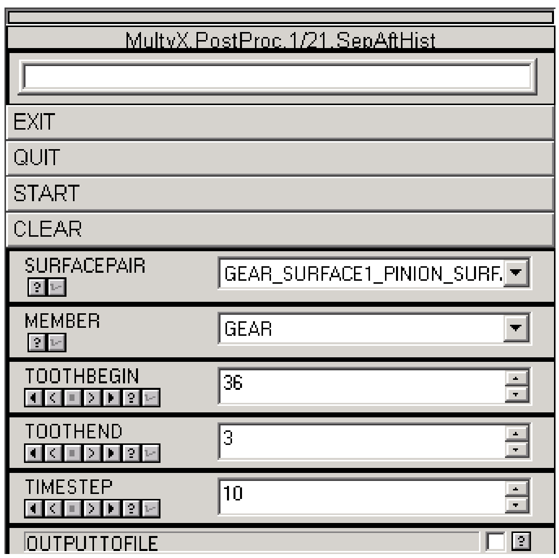


Figure 7.27: The SEPAFTHIST menu.

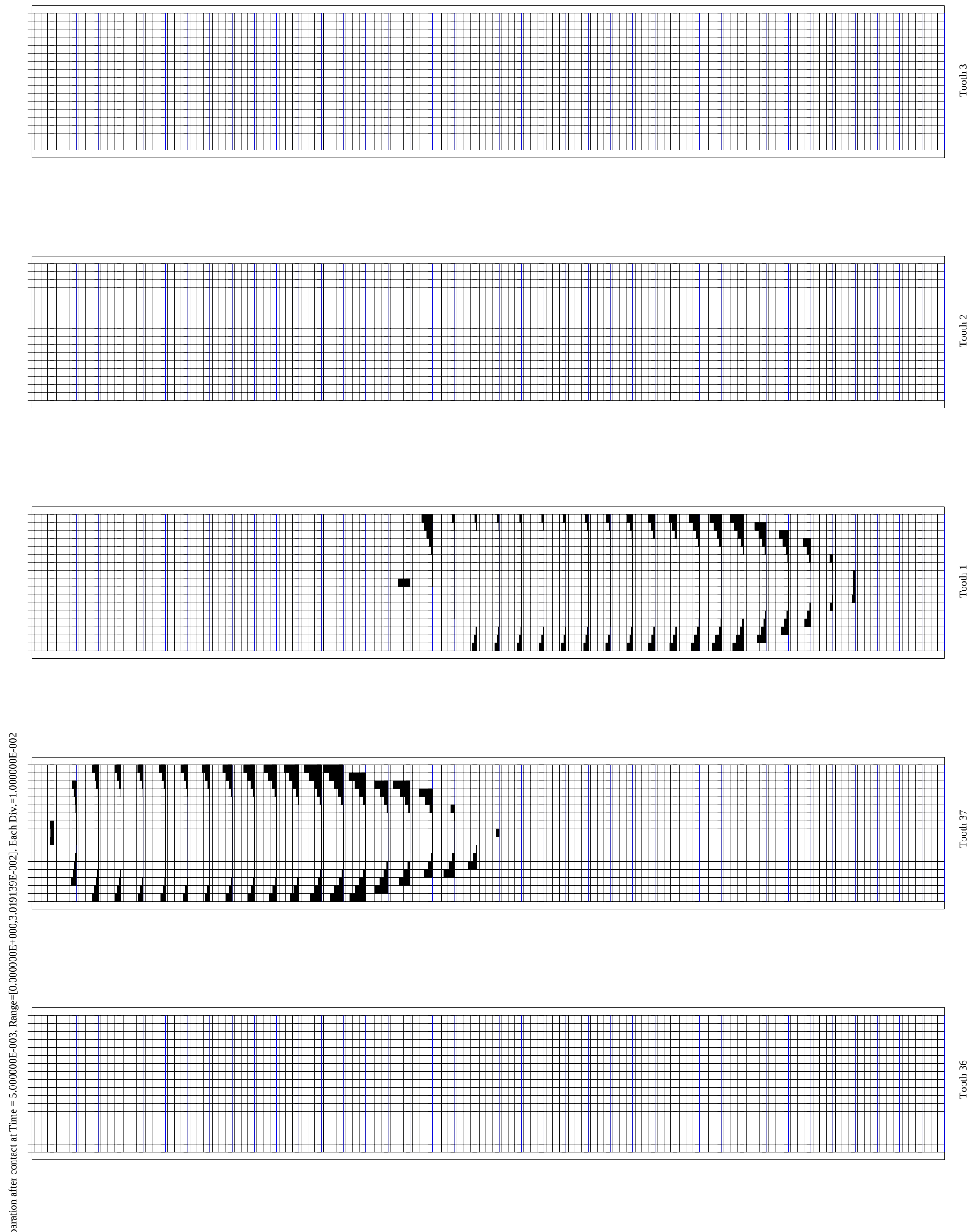


Figure 7.28: The histogram of grid separation after contact, generated by the SEPAFTHIST menu.

7.13 The SEARCHSTRESS command

The SEARCHSTRESS command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.29. This menu is used to locate the most critical stresses in the system.

The COMPONENT box is used to select the stress component of interest. Available choices are MAXPPLSTRESS (the maximum principal normal stress s_1), MINPPLSTRESS (the minimum principal normal stress s_3), MAXSHEAR (the maximum shear stress τ_{max}), and VONMISES (the Von Mises' octahedral shear stress s_{VM}).

Depending on selection in the XAXIS box, the stress can be displayed as a function of time (TIME), profile (SPROF), face (TFACE) or depth (DEPTH).

The stress values are computed over a range of time steps (specified by BEGINSTEP and ENDSTEP), teeth (specified by TOOTHBEGIN and TOOTHEND), location along the profile (specified by SPROFBEGIN, SPROFEND and NUMSPROF), location along the face (specified by TFACEBEGIN, TFACEEND and NUMTFACE), and depth (specified by DEPTHBEGIN, DEPTHEND and NUMDEPTH).

If the number of teeth in the range defined by TOOTHBEGIN and TOOTHEND is 7 or less, and if the SEPTEETH box is checked, then a separate graph is drawn for each tooth. Otherwise a single graph is drawn showing the most critical stress among all the teeth in the range.

Searching for stresses in the depth direction is a very compute intensive operation, so the number of points in the depth direction should be kept at 1 if possible. If a graph of stress vs. depth is desired, then the range of the other parameters should be restricted as much as possible.

File output is controlled by the OUTPUTTOFILE, FILENAME and APPEND items. Figure 7.30 shows an example of stress as a function of time, Figure 7.31 shows stress as a function of profile position. Sharp oscillations can be seen in this graph in the vicinity of the concentrated contact loads. Figure 7.32 shows a graph of stress vs. face.

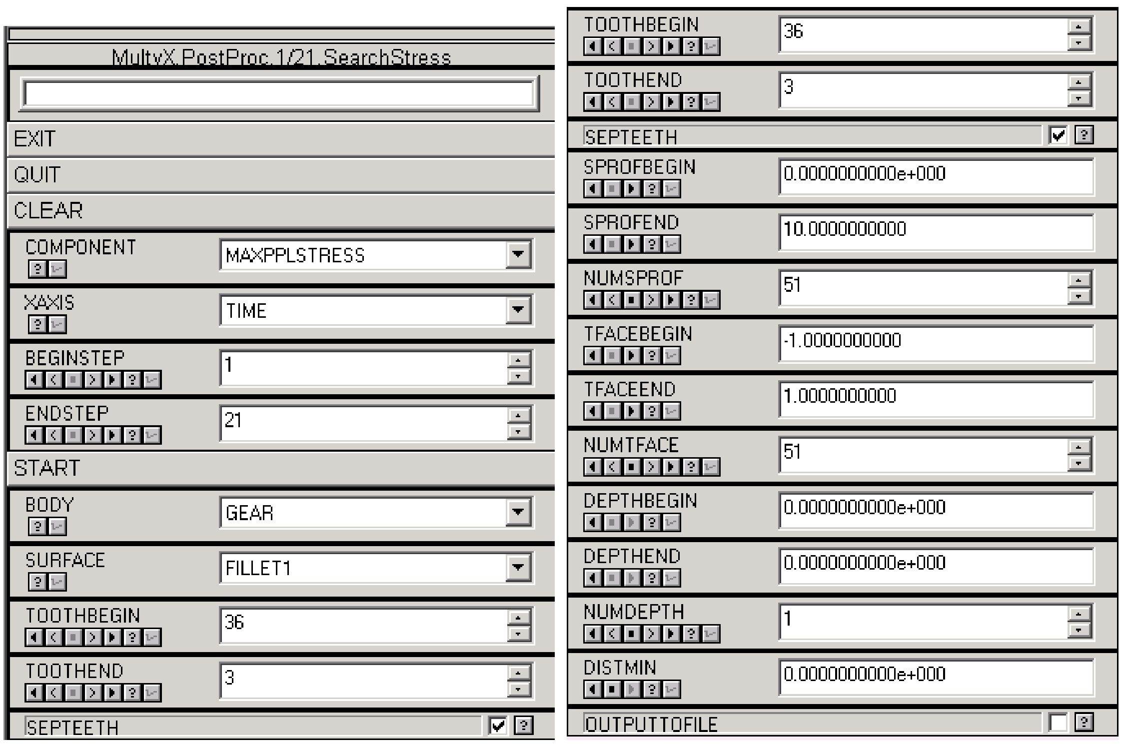


Figure 7.29: The SEARCHSTRESS menu

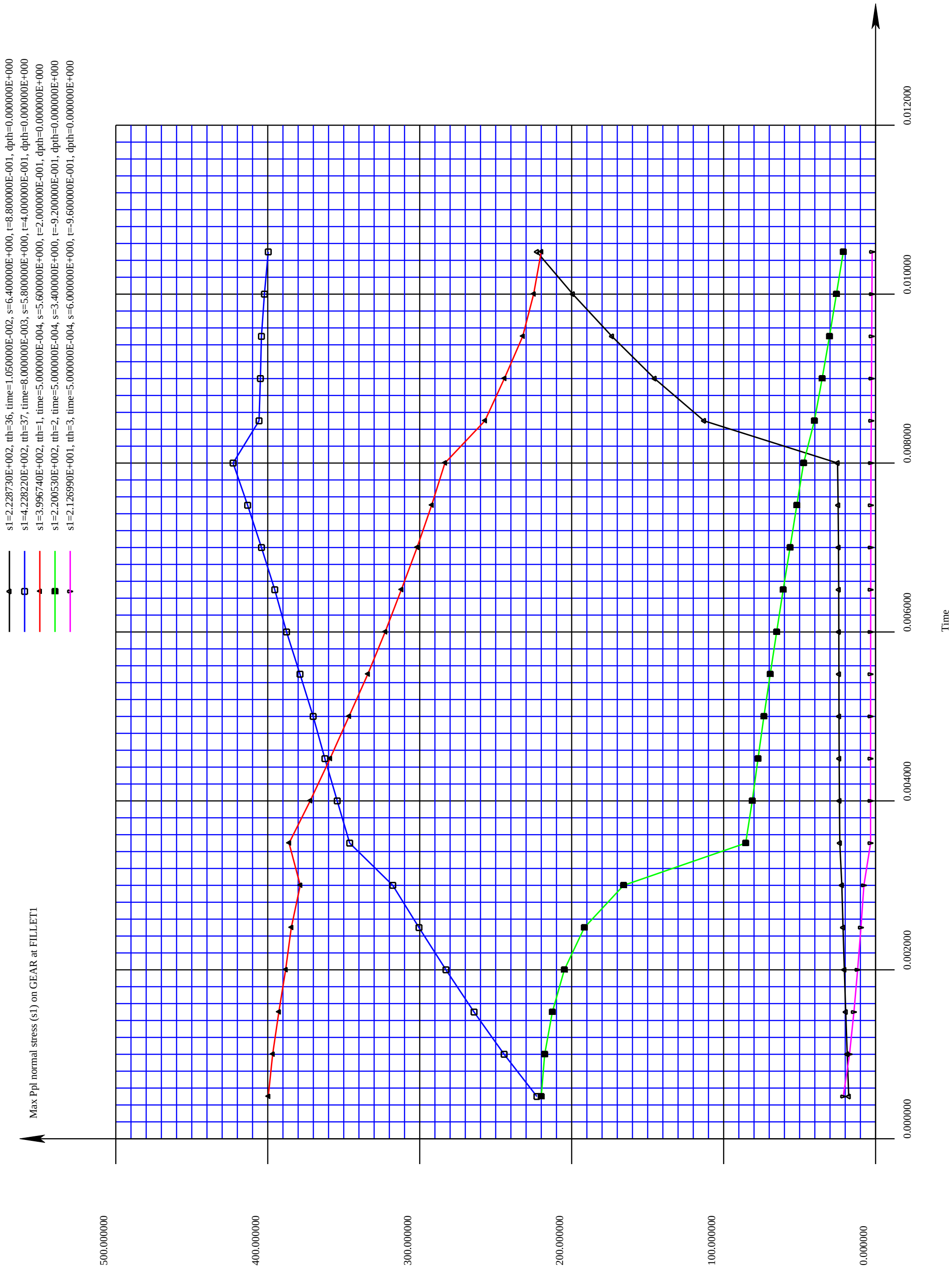


Figure 7.30: The graph of root stress vs. time generated by the SEARCHSTRESS menu.

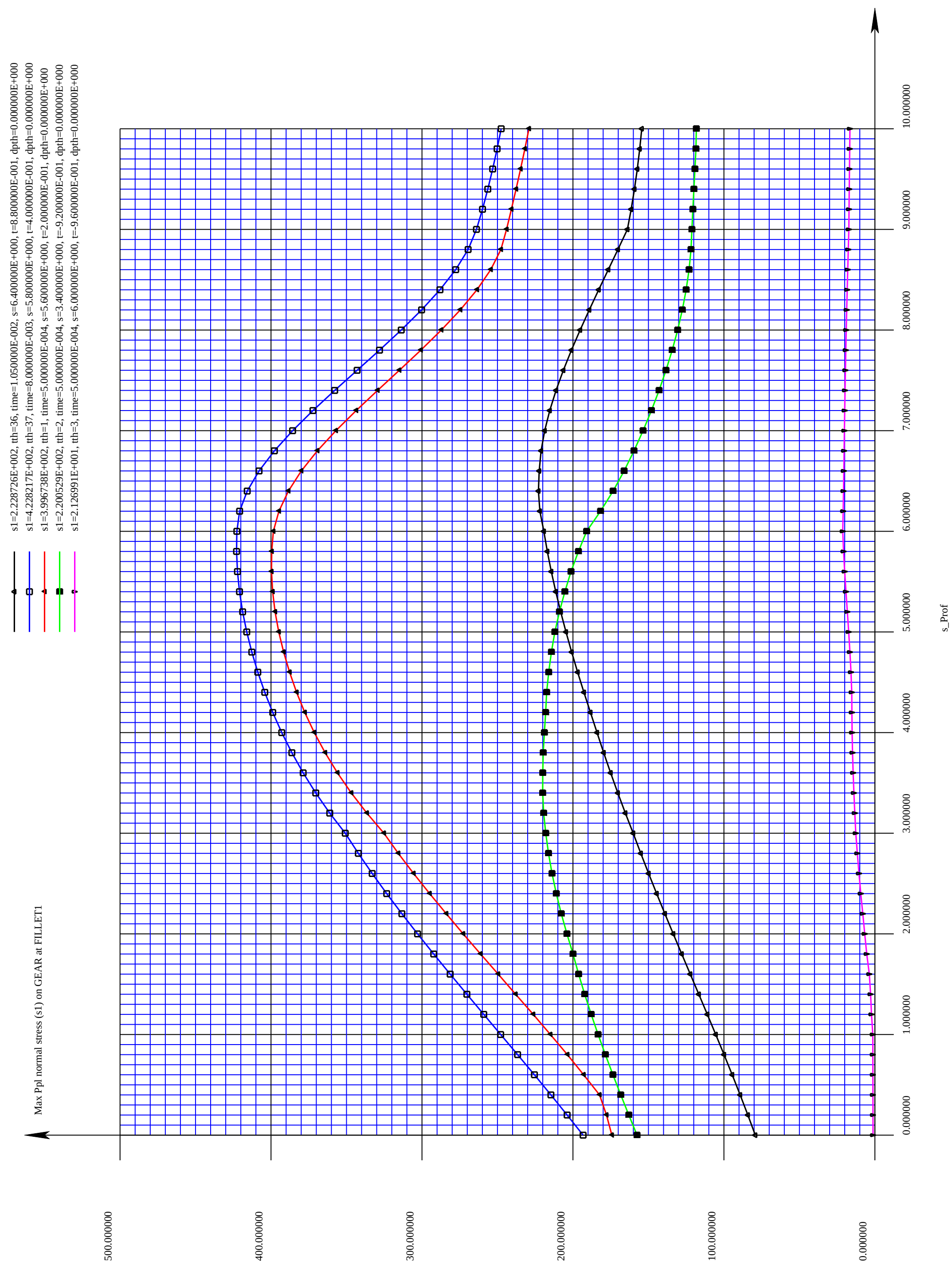


Figure 7.31: The graph of root stress vs. profile generated by the SEARCHSTRESS menu.

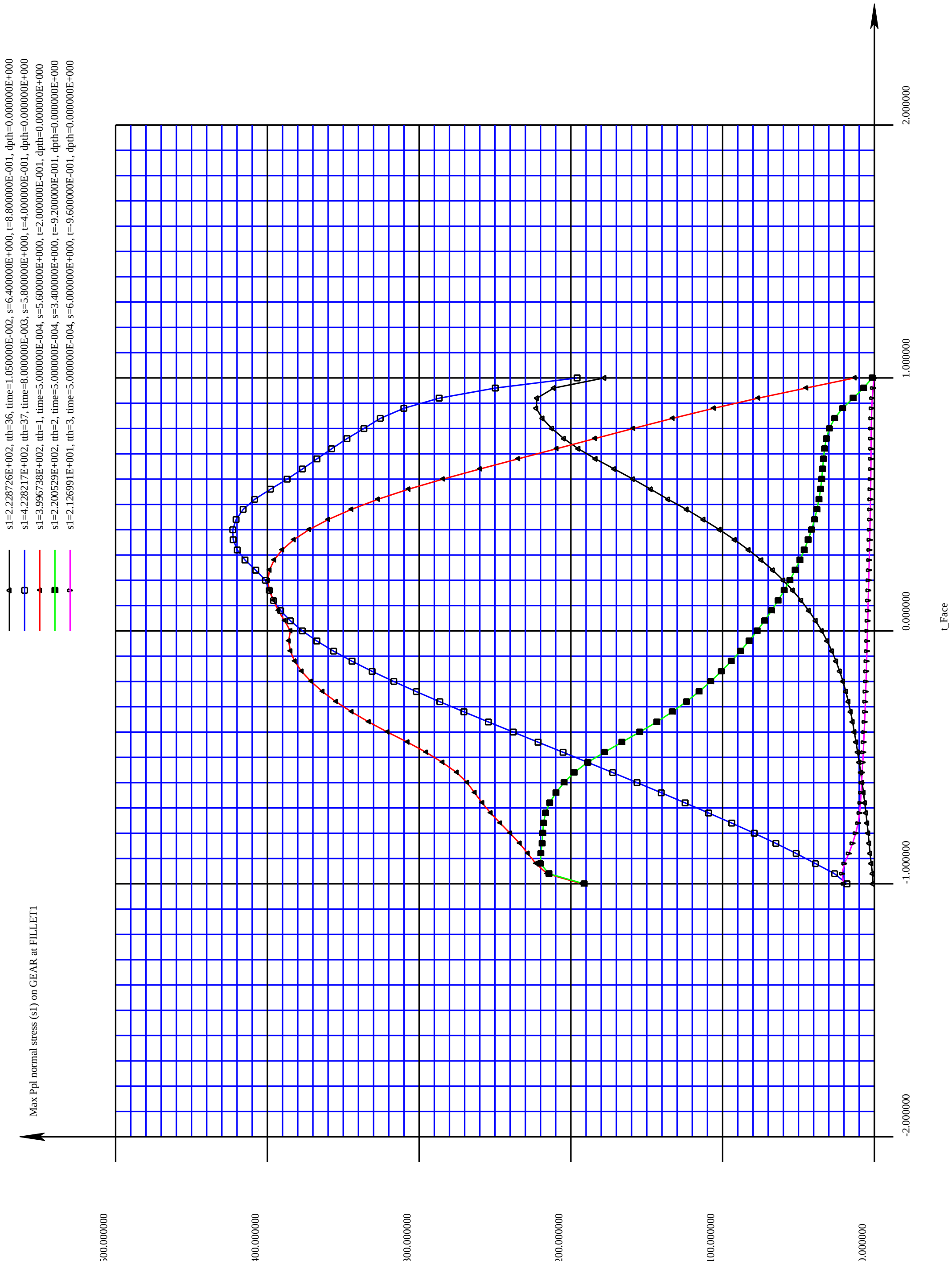


Figure 7.32: The graph of root stress vs. face generated by the SEARCHSTRESS menu.

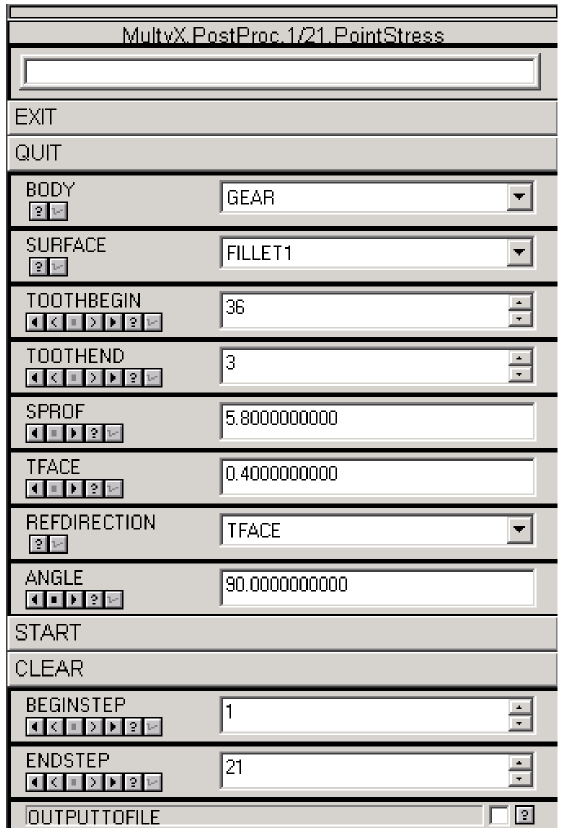


Figure 7.33: The POINTSTRESS menu.

7.14 The POINTSTRESS command

The POINTSTRESS command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.33. This menu is used to track normal stresses in a specific direction at a specific point on a surface.

The surface is selected by specifying the body in the BODY box and a surface in the SURFACE box. A range of teeth with up to 7 teeth is selected through the TOOTHBEGIN and TOOTHEND items. A profile and face location on this surface is specified through the SPROF and TFACE parameters.

The direction is specified by an angle in the item ANGLE. This angle is the angle between the normal direction of interest and the profile direction (if the REFDIRECTION option is SPROF) or the face direction (if the REFDIRECTION option is TFACE). The angle is measured using the right hand rule about the outward normal to the surface.

The range of time steps is specified by the BEGINSTEP and ENDSTEP items. File output is controlled by the OUTPUTTOFILE, FILENAME and APPEND items.

Figure 7.34 shows an example of the graph generated by this menu.

7.15 The PATTERN command

The PATTERN command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.35. This menu is used to create a drawing of the contact pattern on a tooth.

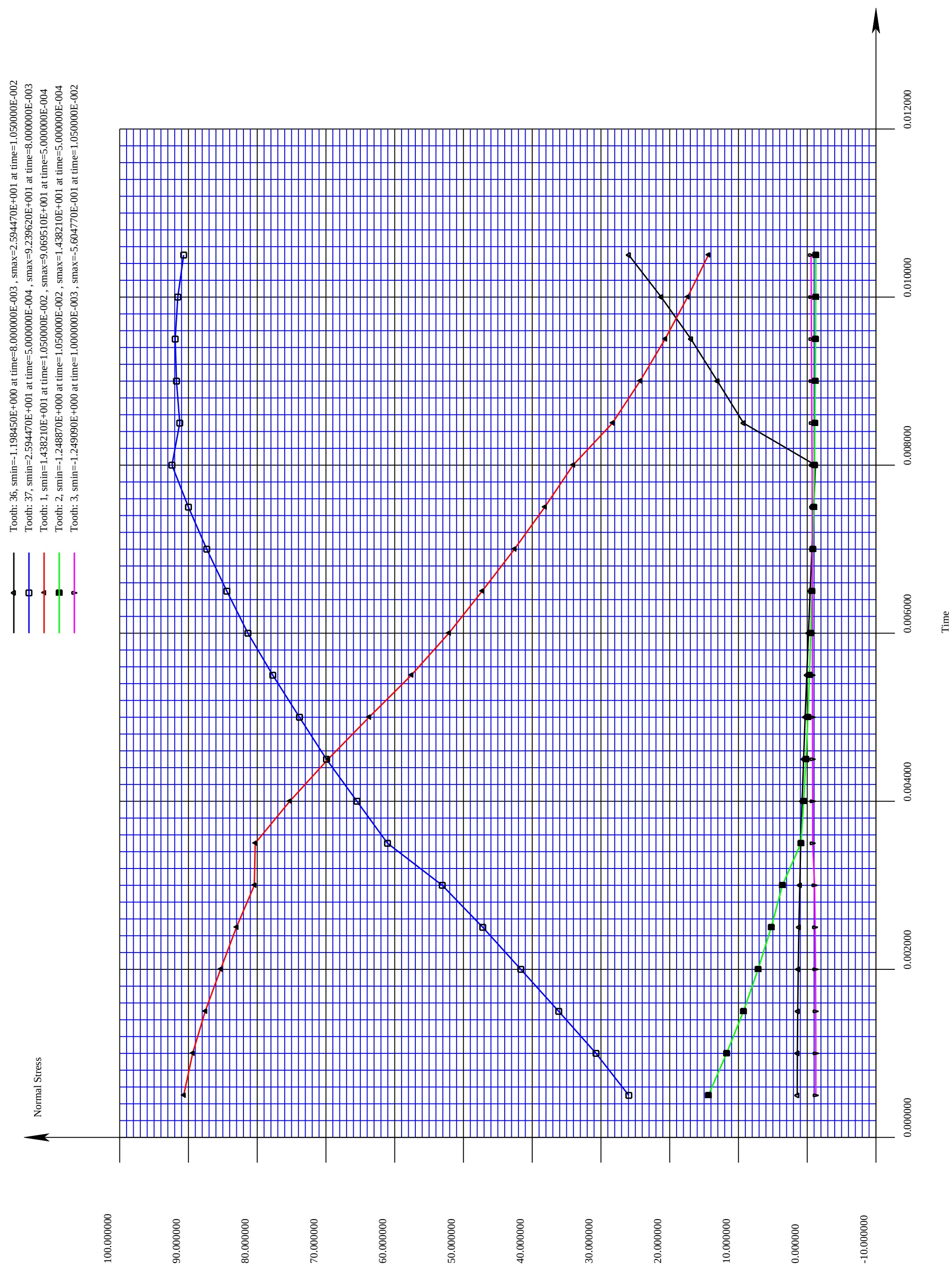


Figure 7.34: The graph of root stress vs. face generated by the POINTSTRESS menu.

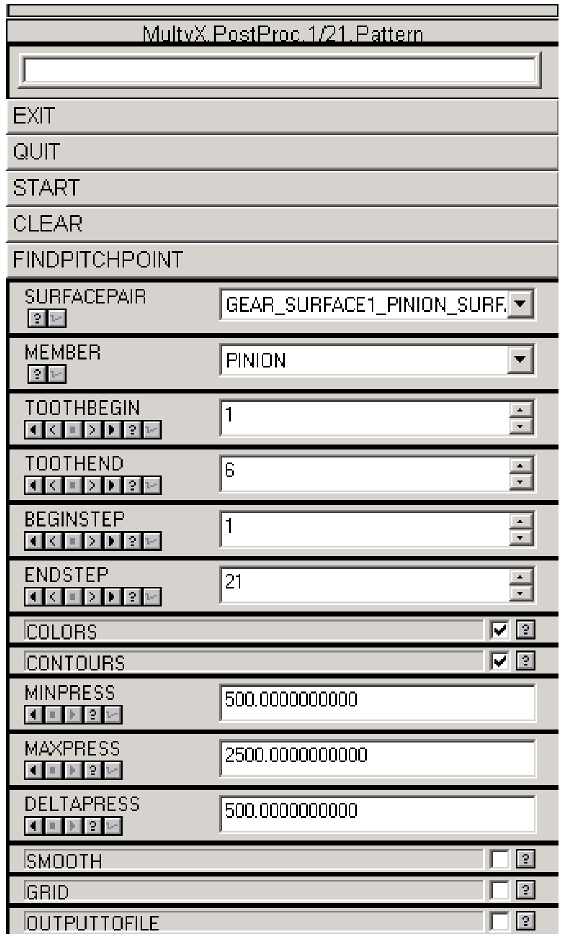


Figure 7.35: The PATTERN menu.

The surface is selected by specifying the body in the BODY box and a surface in the SURFACE box. A range of teeth with up to 7 teeth is selected through the TOOTHBEGIN and TOOTHEND items. The range of time steps is specified by the BEGINSTEP and ENDSTEP items.

The contact pattern can be displayed in color if the COLORS box is checked, or with contour lines if the CONTOURS box is checked. If both options are selected, then a contact pattern like the one shown in Figure 7.36 will be created.

The contact pattern drawing is not three-dimensional. It is a projection of the contact surface in the $r - z$ coordinate plane.

If the SMOOTH box is checked, then the contact pressures will be smoothed by fitting a polynomial surface to the raw data.

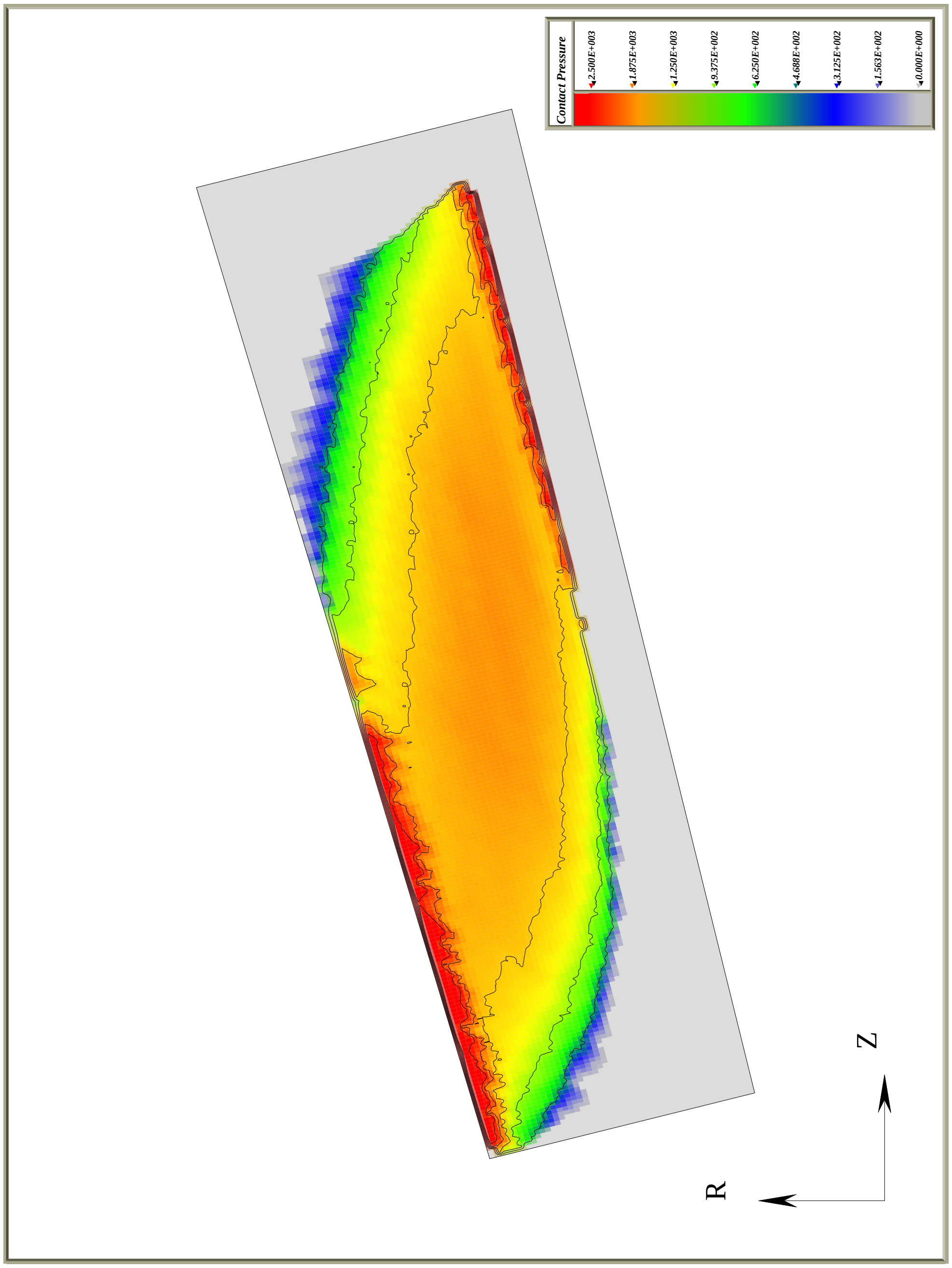


Figure 7.36: The contact pattern generated by the PATTERN menu.

7.16 The AUDIT command

Frequently the user needs to obtain the force and moment balance for the individual bodies in the system. The AUDIT command of the post-processing menu (Figure 7.3) generates an equilibrim 'audit' of all the forces and moments acting on each body. Figure 7.37 shows the AUDIT sub-menu. The list of bodies for which this audit is to be generated is selected through a sub-menu accessed through the SELECT button in this menu. The range if time steps is specified in the BEGINSTEP and ENDSTEP boxes.

The START button then displays the audit statement in the Information window. It can also be sent to an ASCII file by using the OUTPUTTOFILE, FILENAME and APPEND boxes.

A sample equilibrium audit for the pinion shaft is shown below:

```
Time=0.0005
Body no.1:PINION (Origin at:[-44.45,0,-3.332995954e-031])
===============================================================

Contact forces:
    Exerted by:GEAR
    Total  :f [45354.60452,56418.0549,-17953.91639],
            mo[-4915821.122,1600000,-7356082.6]
            m [-4915821.122,801948.4166,-9863865.14]
Total contact force=f [45354.60452,56418.0549,-17953.91639]
                    mo[-4915821.122,1600000,-7356082.6]
                    m [-4915821.122,801948.4166,-9863865.14]


Bearing forces:
Total bearing force=f [0,0,0],
                    mo[0,0,0]
                    m [0,0,0]


Total internal force (inertial+press+body):f [0,0,0],
                                           mo[0,0,0]
                                           m [0,0,0]
Total mass & damping force                :f [0,0,0],
                                           mo[0,0,0]
                                           m [0,0,0]
Total contact  force                      :f [45354.60452,56418.0549,-17953.91639],
                                           mo[-4915821.122,1600000,-7356082.6]
                                           m [-4915821.122,801948.4166,-9863865.14]
Total bearing  force                      :f [0,0,0],
                                           mo[0,0,0]
                                           m [0,0,0]
Total reaction force                      :f [-45354.60452,-56418.0549,17953.91639],
                                           mo[4915821.122,-1600000,7356082.6]
                                           m [4915821.122,-801948.4166,9863865.14]
===============================================================================
Residual force (error)                    :f [-1.455191523e-011,0,-3.637978807e-012],
                                           mo[-1.862645149e-009,3.492459655e-009,2.793967724e-009]
                                           m [-1.862645149e-009,3.330751497e-009,2.793967724e-009]
```

The forces (and moments) are broken down into contact forces, bearing forces, internal forces, mass and damping forces and reaction forces. The reaction forces are the forces exerted by the reference frame constraints.

Two values for the moments are displayed. In the above example, `mo` refers to the moments computed about the origin of the pinion shaft body. `m` stands for the moment computed about the origin of the fixed reference frame. The moments about the fixed reference frame are more useful in comparing the action and reaction acting on different bodies.

Regardless of the origin about which the moments are computed, the X Y and Z components of each force and moment always refer to the fixed reference frame.

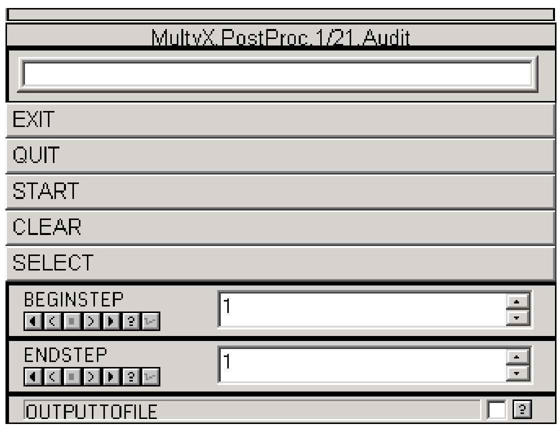


Figure 7.37: The AUDIT menu.

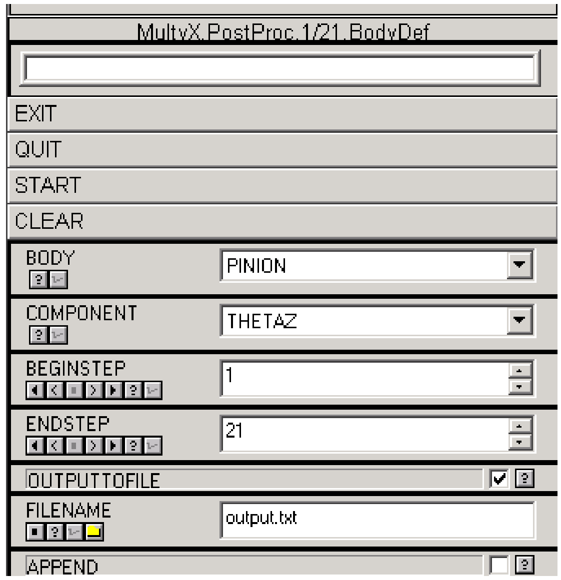


Figure 7.38: The BODYDEFLECTION menu.

7.17 The BODYDEFLECTION command

The BODYDEFLECTION command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.38. This menu is used to generate a graph (Figure 7.39) of a component of the rigid body type motion of a body as a function of time. The six components of motion that can be graphed are the 3 translation motions u_x, u_y and u_z, and the three rotation components θ_x, θ_y and θ_z. These components are calculated in the reference frame attached to the body. The rotation components are displayed in Radians. The θ_z component of the body deflection is used to study the transmission error.

7.18 The BODYREACTION command

The BODYREACTION command of the post-processing menu (Figure 7.3) leads to the menu shown in Figure 7.40. This menu is used to generate a graph (Figure 7.41) of a component of the body frame reaction as a function of time. The six force components that can be graphed are the three forces F_x, F_y and F_z, and the three moments M_x, M_y and M_z. These components are calculated in the reference frame attached to the body. The moments are computed about origin of this reference frame.

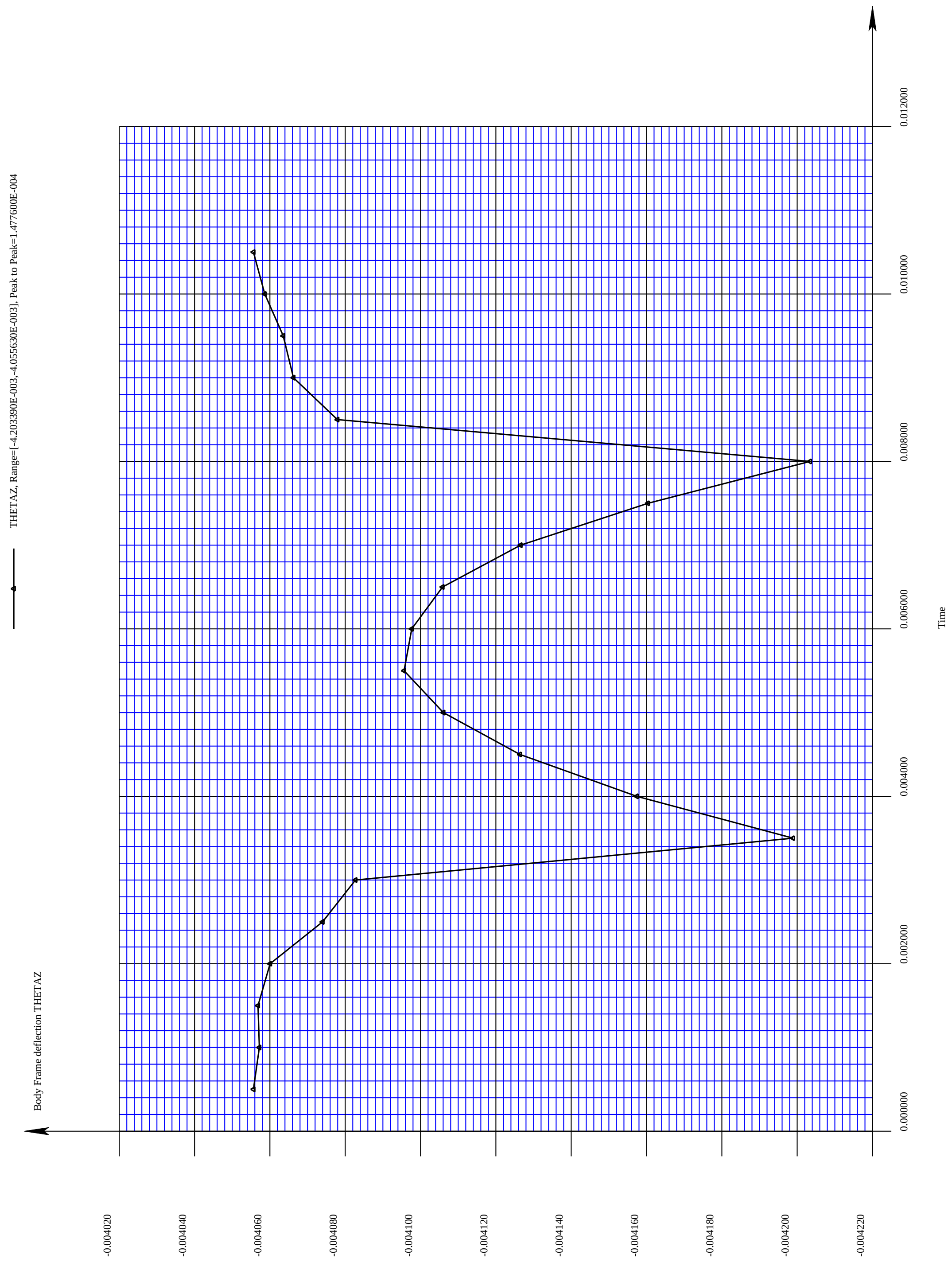


Figure 7.39: The graph generated by the BODYDEFLECTION menu.

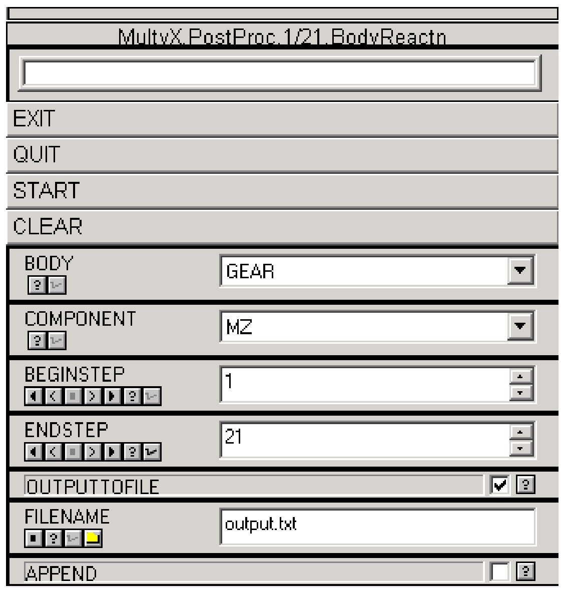


Figure 7.40: The BODYREACTION menu.

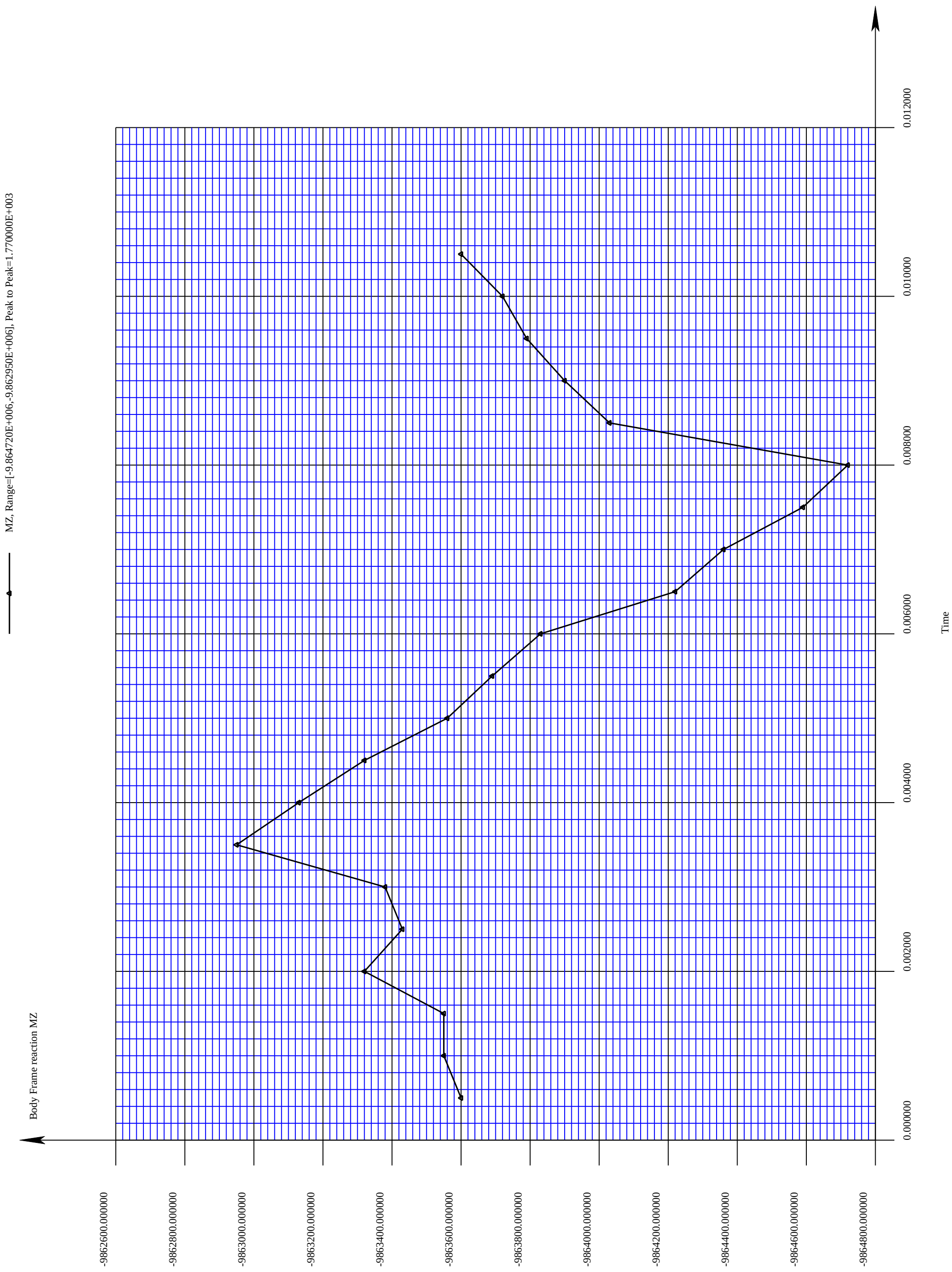


Figure 7.41: The graph generated by the BODYREACTION menu.

Chapter 8

Pre and Post processing using IglassViewer

IglassViewer is a very powerful tool for pre and postprocessing gear models and results. Several features have been added to the Multyx program so as to enhance the compatability with IglassViewer. Thus it can be considered as a program which enables the user to view pre and postprocessing files generated by an external code. Note that the IglassViewer graphics window is independent of the guide graphics window. The advantage of using IglassViewer over guide program for pre and postprocessing is that it is more faster, efficient and more simple to operate. Also, you can visualise the models in their dynamic mode which is not possible using the Guide program. Following sections gives a detailed explanation of the procedure for creating the pre and postprocessing iglass files and also the various functions associated with the iglass program.

8.1 Generating an Iglass file for preprocessing

The GENIGLASSFILE command in Figure 7.1 will lead to a menu shown in Figure 8.1 using which you can generate a preprocessing file for Iglass. The filename is specified in the IGLASSFILENAME menu. The time at which the user wants to visualise the model can be specified in the TIME menu. The user can also visualise the model at a sequence of time steps by entering the number of steps in the NTIMESTEPS menu. The DELTATIME menu is the value of time increment between successive writes to the iglass file. The POPUPIGLASS menu if turned on will automatically open up the Iglass graphical window after the Igass file is generated. If it is not turned on, only the data file for iglass will be created, and iglass will have to be started manually. Using the SELECT menu in Figure 8.1 the user can select the bodies to be displayed in the Iglass graphical window. Click on the START button in Figure 8.1 to generate the Iglass preprocessing file. After the file is generated and if the POPUPIGLASS menu is turned on a separate Iglass window will open showing the reference axes and the gear bodies (selected in the SELECT menu). An example of the Iglass preprocessing window for a Hypoid gear pair is shown in Figure 8.2. As shown in Figure, it has 3 menus- View, Bodies and Attributes. The Attributes menu is used more commonly in the postprocessing mode. The 'Exit' button in each menu will close the Iglass graphics window.

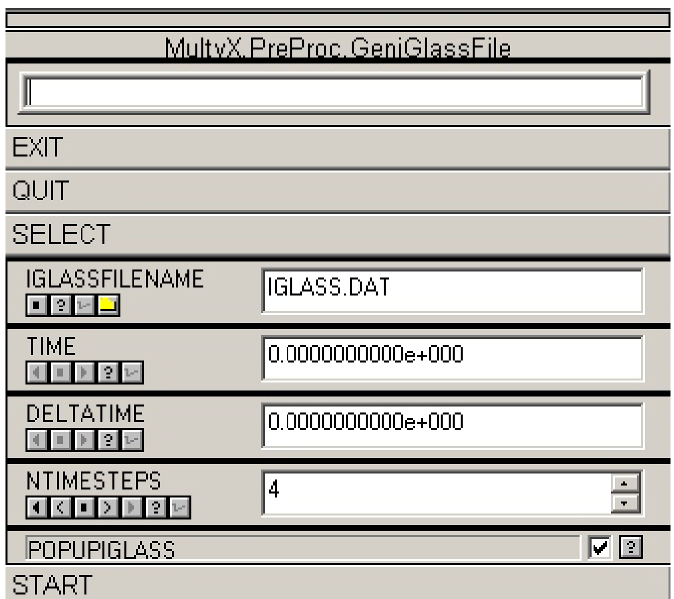


Figure 8.1: The generate Iglass file menu

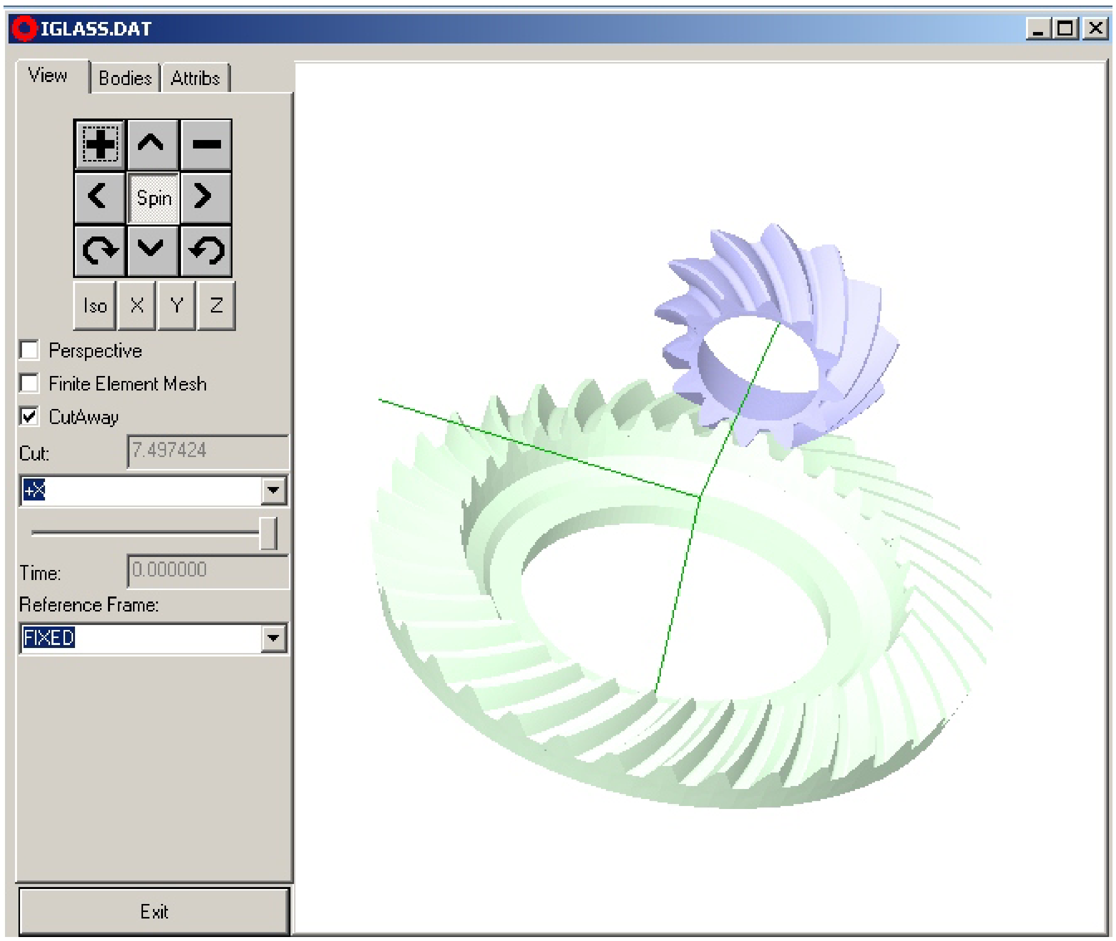


Figure 8.2: An example of an Iglass preprocessing window.

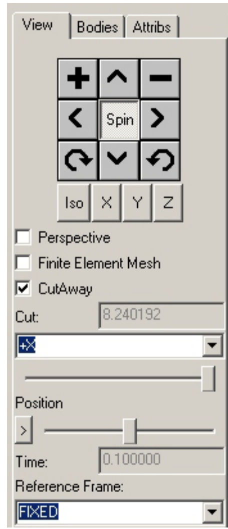


Figure 8.3: Iglass preprocessing view menu

8.2 View menu

The View menu is shown in Figure 8.3. Table 8.1 shows the common tasks performed by some of the buttons displayed in the Iglass window.

Apart from all the features shown in Table 8.1 you can also rotate the model using the left mouse button. Drag the left mouse button in the direction you want to rotate the model in the iglass graphics window. Also the model can be moved in the graphics window in any directions you want using the right mouse button. Drag the right mouse button in the direction you want to move the model in the iglass graphics window.

8.2.1 Finite element mesh

The finite element mesh model can be visualised if the 'Finite Element Mesh' item is selected. Figure 8.4 shows the finite element mesh model of the gear bodies in iglass preprocessing.

8.2.2 Cutting plane

Using the cutting plane switch shown in Figure 8.5 you can visualise the model along a section. This feature is especially useful in pre and post processing of complicated models with a large number of internal gears. The cutting plane can be selected along the +ve and -ve X, Y and Z axes. Using the button below the cutplane switch you can select the cutting plane at various points along the axis chosen by the cut plane switch option.

8.2.3 Selecting the time step

User can visualise the model at a particular timestep in iglass pre-processing using the 'Position' slider shown in Figure 8.6. Each position corresponds to the DELTATIME selected in the generate iglass file menu. The corresponding time can be seen in the 'Time' item shown in Figure 8.7.

Table 8.1: Common buttons in Iglass pre and postprocessing window

Button	Purpose
+	Zoom In
−	Zoom Out
^	Move the model upwards (If Spin is turned OFF)
v	Move the model downwards (If Spin is turned OFF)
>	Move the model towards right (If Spin is turned OFF)
<	Move the model towards left (If Spin is turned OFF)
^	Rotate the model upwards (If Spin is turned ON)
v	Rotate the model downwards (If Spin is turned ON)
>	Rotate the model towards right (If Spin is turned ON)
<	Rotate the model towards left (If Spin is turned ON)
↻	Rotate the model clockwise (If Spin is turned ON)
↺	Rotate the model counterclockwise (If Spin is turned ON)
Iso	View the model in an isometric view
X	View the model in the $Y - Z$ plane
Y	View the model in the $X - Z$ plane
Z	View the model in the $X - Y$ plane

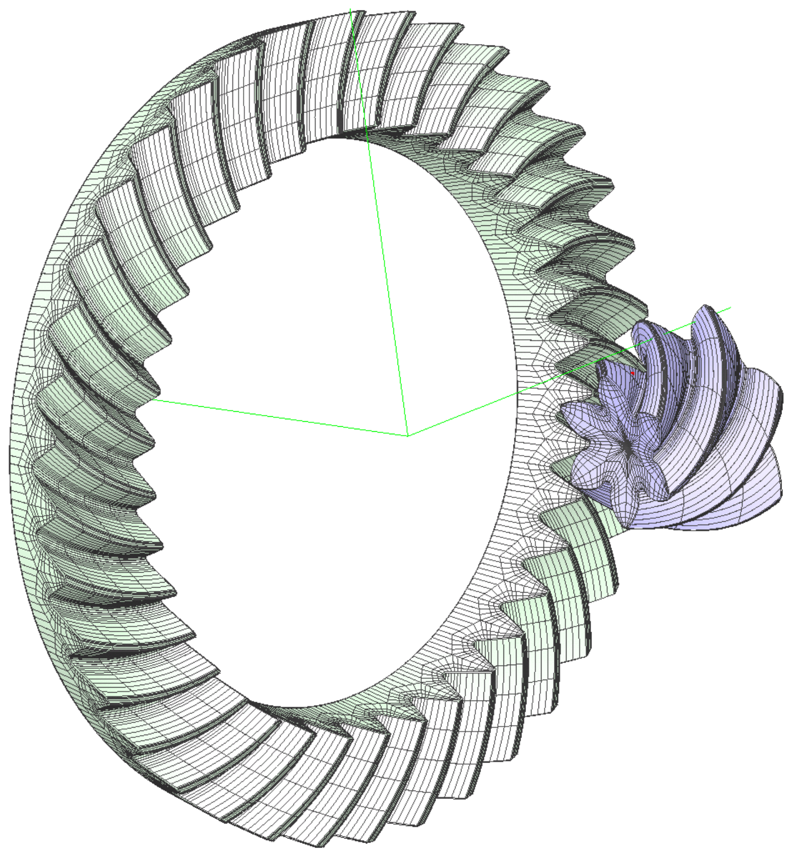


Figure 8.4: Finite element mesh model of the gear bodies

Figure 8.5: The cutting plane switch.

Figure 8.6: The position slider.

Figure 8.7: The time menu.

Figure 8.8: The reference frame switch.

8.2.4 Reference frames

The default reference frame is the FIXED reference frame. Both, the pinion and the gear appear to move when observed from the FIXED frame. The model will align itself to this reference frame when the iglass window pops up. The reference frame can be aligned to a body member using the reference frame switch shown in Figure 8.8. If you select the GEAR as the reference frame the reference frame origin will coincide with the origin of the gear. The gear appears stationary when observed from the GEAR reference frame, and the pinion orbits around it. If the PINION option is selected then the reference frame origin aligns itself to the origin of the pinion.

8.3 The Bodies menu

The 'Bodies' menu is shown in Figure 8.9. The body member can be turned on or off by clicking on the member name in the Bodies menu. User can view the tooth and the rim sector separately for each gear body.

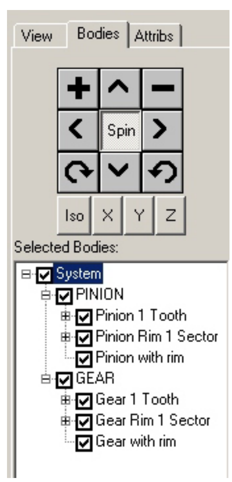


Figure 8.9: Iglass preprocessing Bodies menu

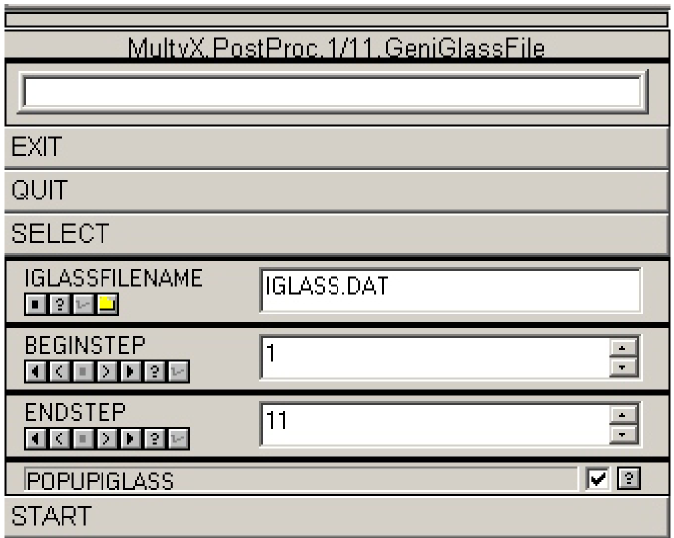


Figure 8.10: The generate iglass file menu for post processing.

8.4 Post processing using iglass

The GENIGLASSFILE command in Figure 7.3 leads to the generate iglass file menu shown in Figure 8.10 for post processing in iglass. BEGINSTEP and ENDSTEP menus shown in Figure 8.10 define the range for which you want to check for results. Note that these menus are independent of the GOTOPOSN menu shown in Figure 7.3.

An example of an iglass post processing window is shown in Figure 8.11.

8.5 Features specific to iglass post processing

The position switch shown in Figure 8.12 is used to run the simulation of the model in the post processing iglass window. You can look at the simulation at a particular time step by dragging the slider along the scale. The 'Defmn'(deformation) slider shown in Figure 8.13 is used to view the deformed shaped of the gear bodies. The 'Rigid Defl' and the 'F.E.Defl' shows the rigid body deflection and the finite element deflection of the bodies. The magnification scale of deformation can be adjusted using the slider. The load slider shown in Figure 8.14 is used to look for the load patterns on a tooth over the range of time step selected in the BEGINSTEP and ENDSTEP menus. The magnification scale of loading can be adjusted using the slider. The directions of the bearing forces and moments can be visualised using the 'Brg Frc' and 'Brg Mom' sliders shown in Figure 8.15. The magnification scale of the forces and the moments can be adjusted using the respective sliders.

The 'Attribs' menu is shown in Figure 8.16. The attribute menu shown in Figure 8.17 is used to check for contours for different component of results. The available options are DISPLVECTOR, MAXPPLNORMAL, S2PPLNORMAL, MINPPLNORMAL, MAXSHEAR, VONMISES and ERRORESTIMATE. The DISPLVECTOR will pop up a component switch using which the contour for displacement vector in the X, Y and Z directions can be displayed. MAXPPLNORMAL, S2PPLNORMAL, MINPPLNORMAL, MAXSHEAR, VONMISES menus show their respective stress contours. The ERRORESTIMATE menu is used to display the stress error estimate. This error estimate is computed from the magnitude of the inter-element stress discontinuity.

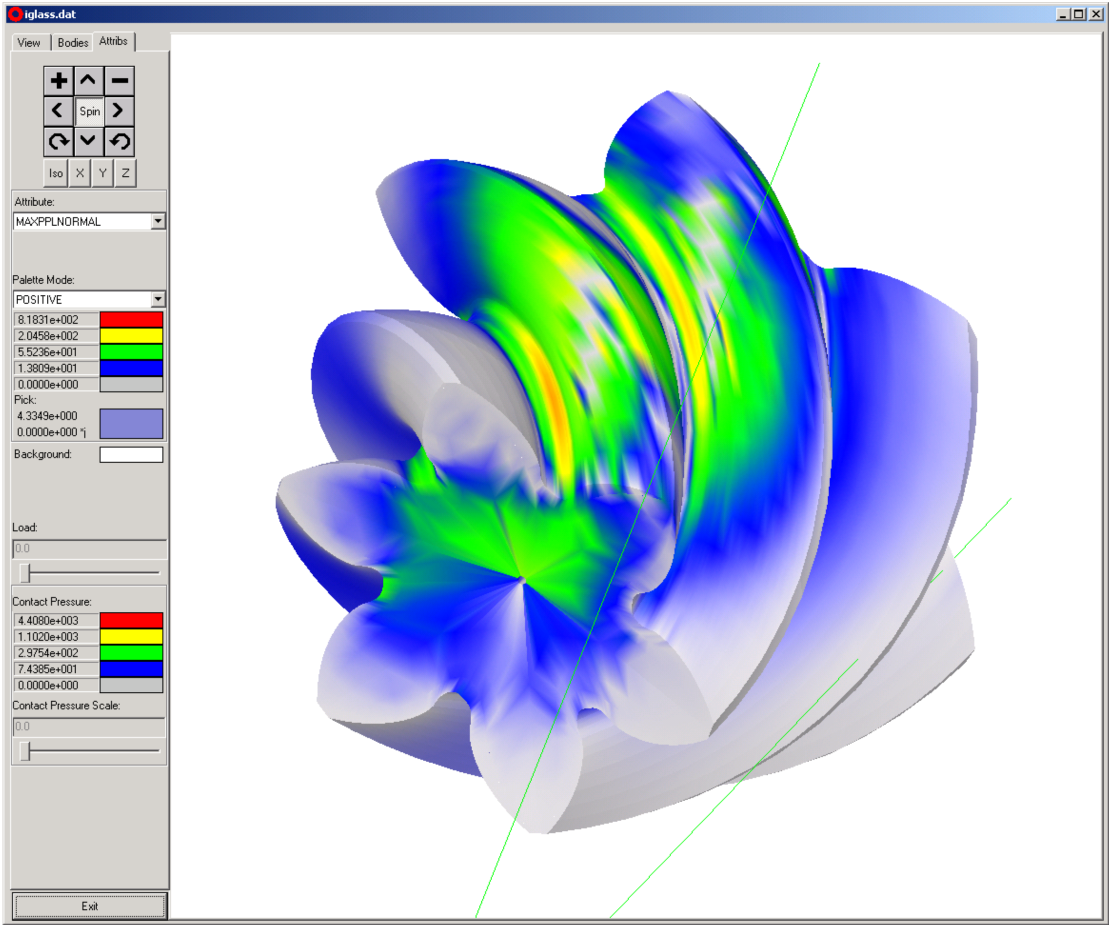


Figure 8.11: Iglass in post processing mode.

Figure 8.12: The position slider.

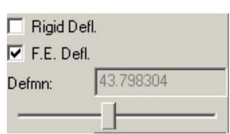


Figure 8.13: The deformation slider.

Figure 8.14: The load slider.

Figure 8.15: The bearing forces and moments sliders.

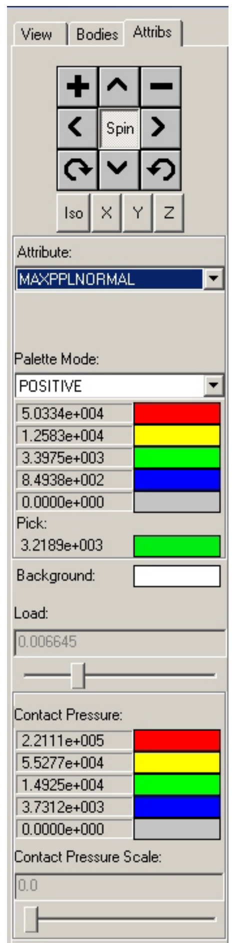


Figure 8.16: The iglass postprocessing attribute menu.

Figure 8.17: The attribute switch.

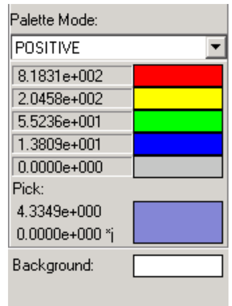


Figure 8.18: The palette switch.

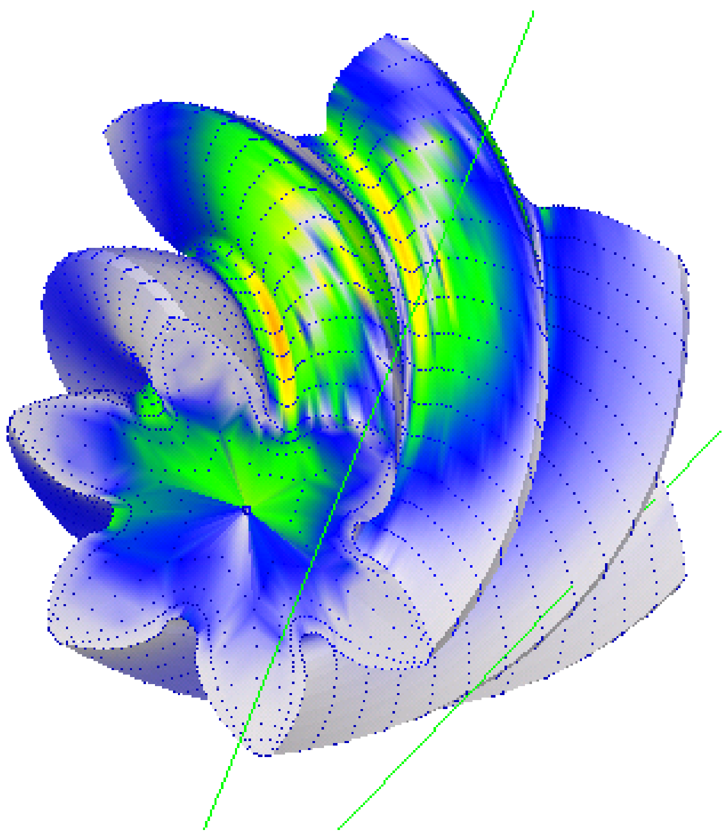


Figure 8.19: Picking a nodal point to examine stresses.

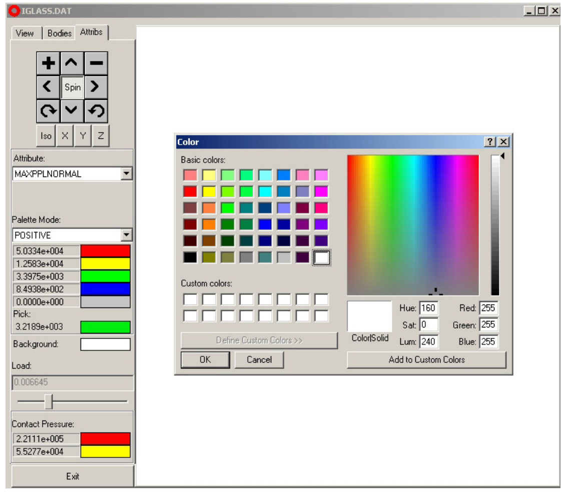


Figure 8.20: The background color popup window switch.

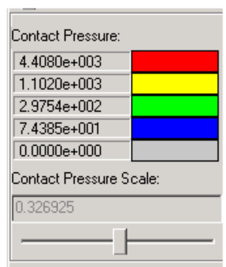


Figure 8.21: The Contact pattern legend.

The colors for minimum and maximum stress contours can be controlled using the palette mode menu shown in Figure 8.18. A POSITIVE mode will align the scale from 0 (minimum stress) to a maximum positive value (maximum stress). A NEGATIVE mode will align the scale from 0 to a negative value. The BOTH type mode will align the scale from the maximum negative value (minimum stress) to a maximum positive value (maximum stress). So as to find the stress at a node, double click on the gear body. The finite element nodes are now visible as shown in figure 8.19. Clicking once on the node will show the stress at that nodal point in the 'pick' item of the Palette menu.

Double clicking on the 'Background' button will popup the 'Color' window shown in Figure 8.20 using which you can change the background color of the iglass graphics window.

The Contact pattern legend shown in Figure 8.21 is used to read the contact pressure on the contacting surfaces. Figure 8.22 shows the contact pressures on the pinion teeth.

The EXIT button will take you out of the iglass post processing window.

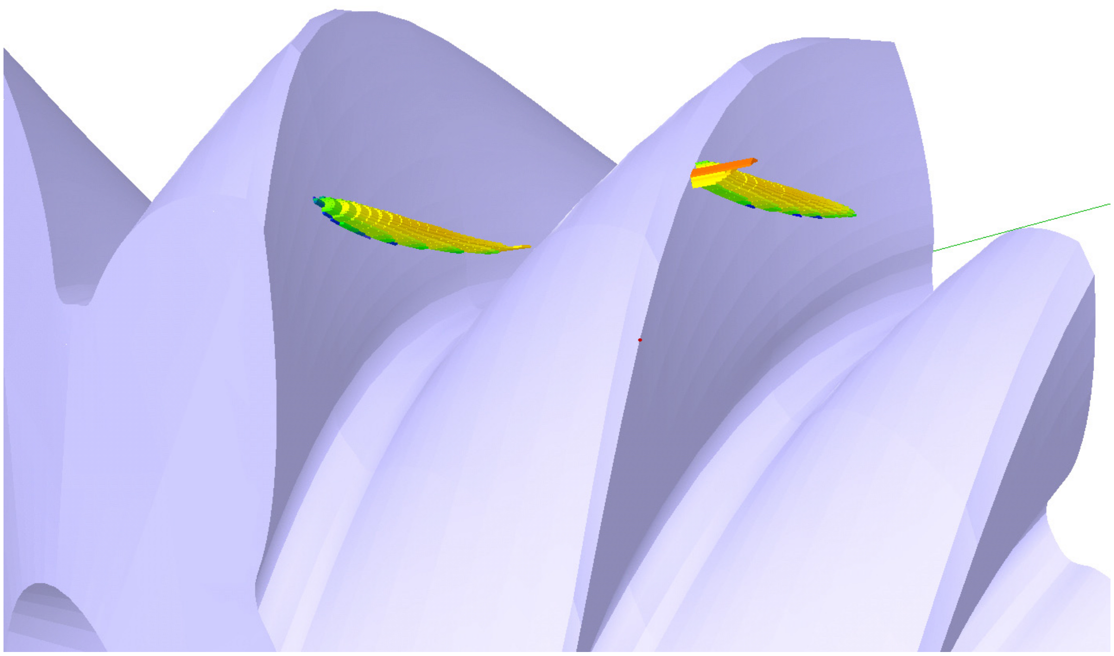


Figure 8.22: The contact pressure distribution on the pinion teeth.

Appendix A

Tooth Mesh Templates

The finite element meshes in the *HypoidFaceHobbed* package are created with very little input from the user. The user does not need to provide any of the node numbering and element connectivity information to the model generator. This information is read by the program from pre-existing files called 'template' files.

Figures A.1 through A.4 show the element connectivity and element numbering scheme used in the four standard templates. The orientation of the element coordinate system is indicated by the notch in one of the corners of each element. The range of the surface profile coordinate s for the two contact surfaces is also shown.

The element orientation for the rim sector is shown in Figure A.5.

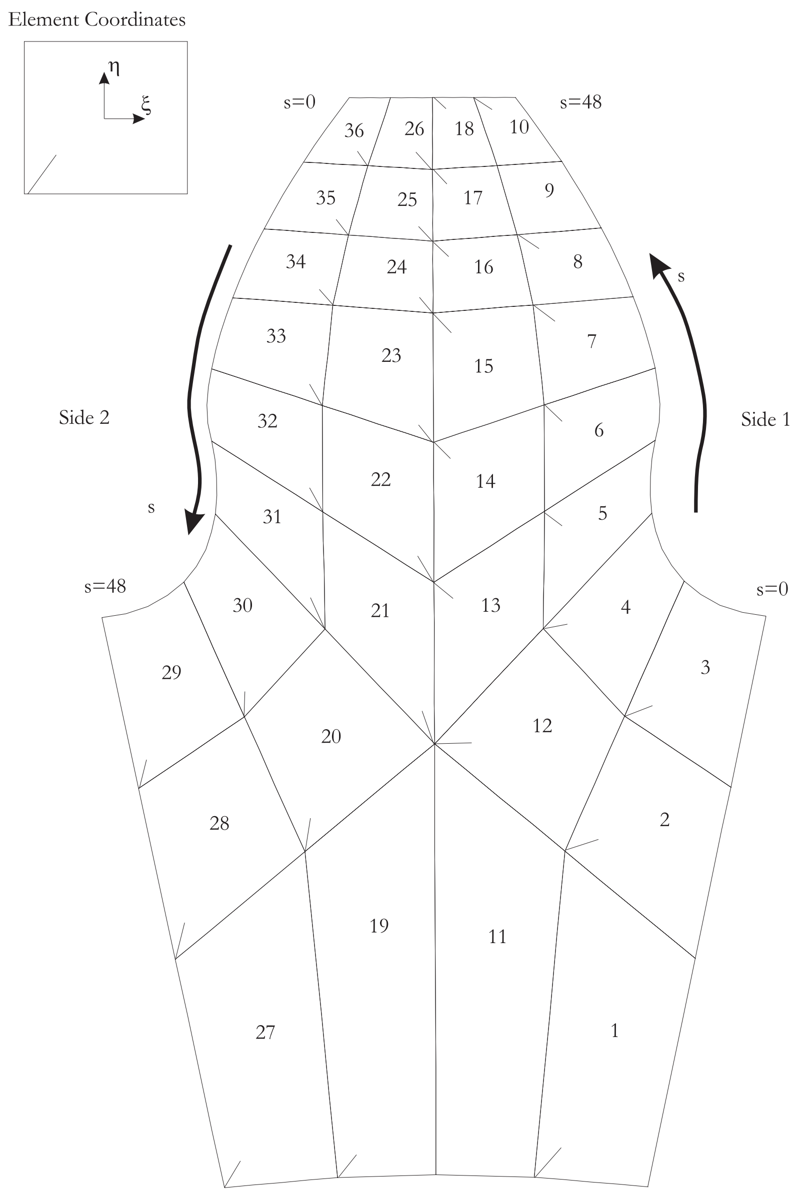


Figure A.1: The MEDIUM.TPL template file.

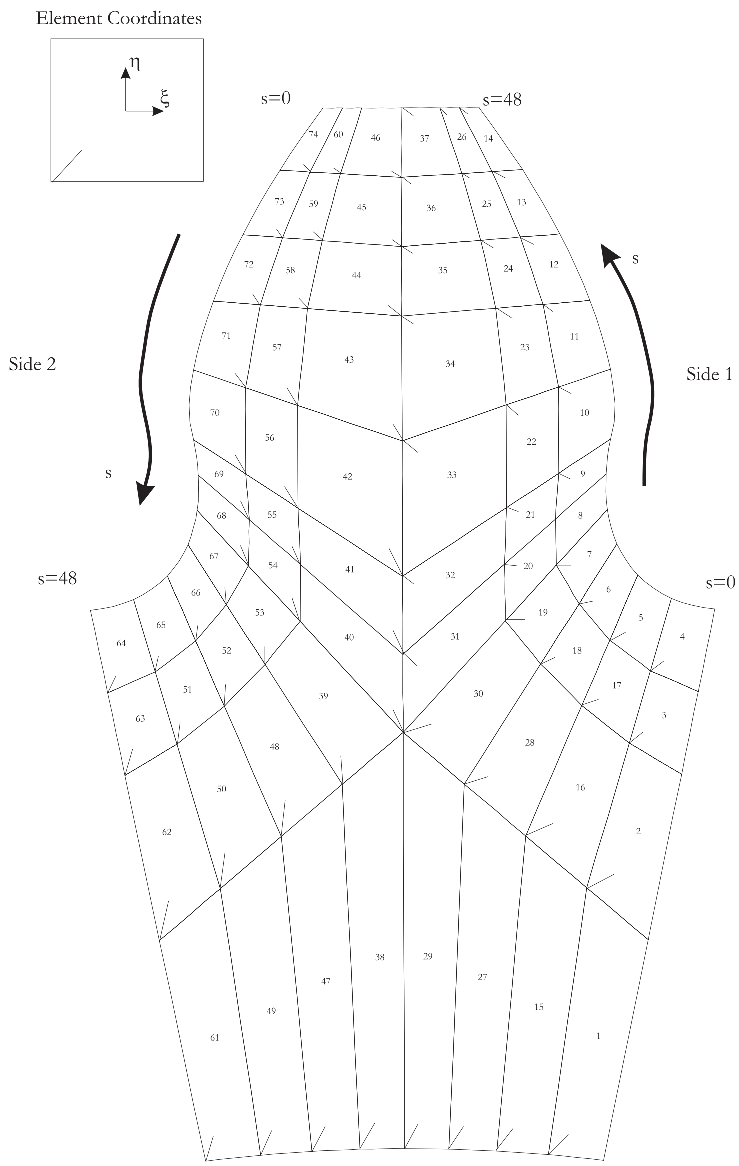


Figure A.2: The FINEROOT.TPL template file.

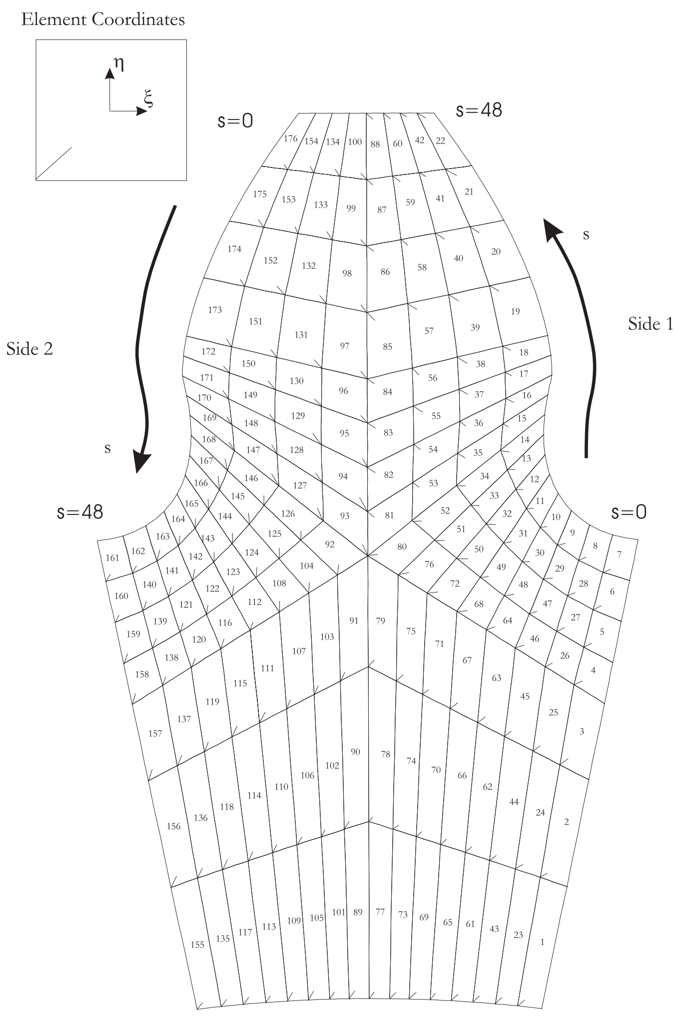


Figure A.3: The FINEST.TPL template file.

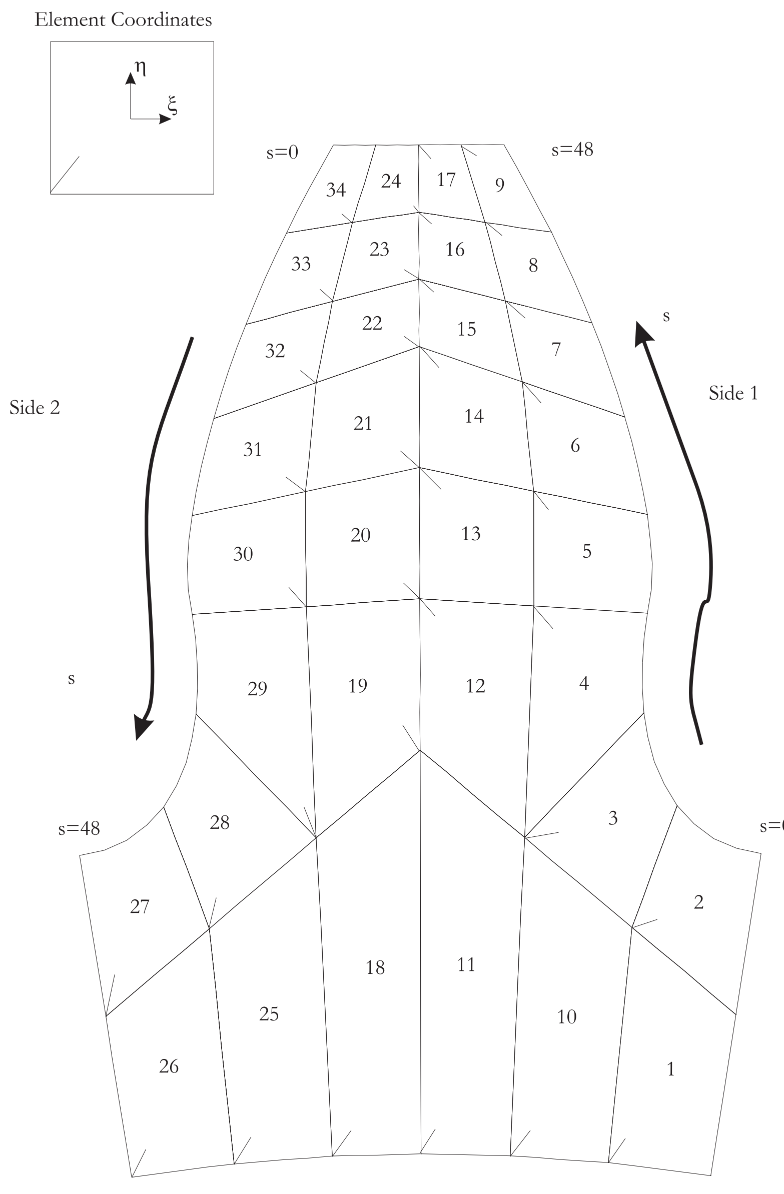


Figure A.4: The THINRIM.TPL template file.

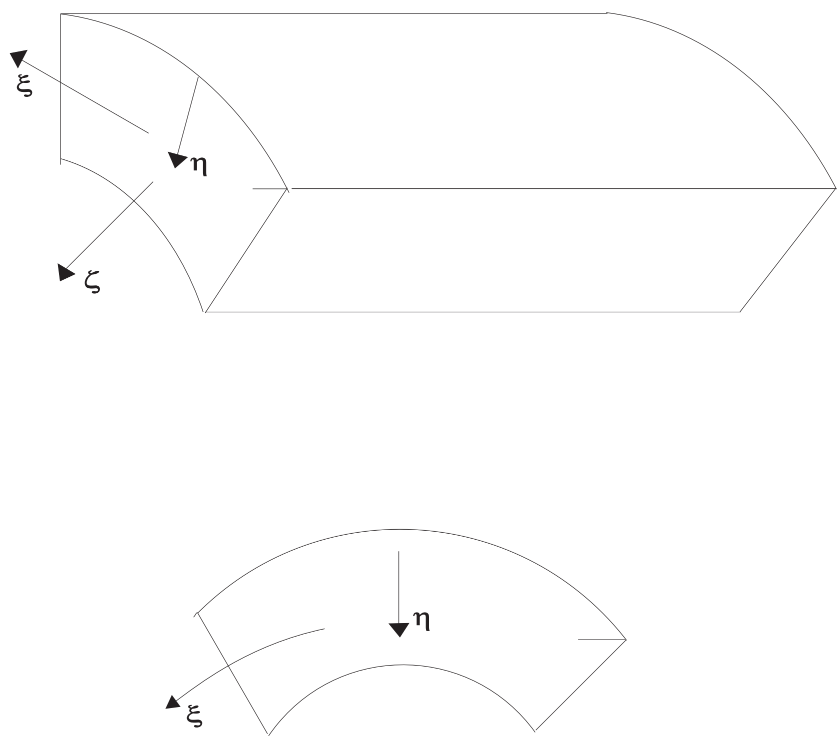


Figure A.5: Element orientation for the rim sector

Appendix B

Special Analysis File

If the hypoid gear is manufactured using the Gleason's manufacturing process then the finite element analyst is given a special analysis file by the gear manufacturer. This file contains the design data used for the manufacturing of the hypoid gear. An example of a special analysis file record is shown in Table B.1.

Table B.1: An example of Special Analysis File Record

ITEM	RECORD 7	GEAR DATA
1	ADDENDUM	0.129287
2	DEDENDUM	0.366338
3	CLEARANCE	0.058615
4	WHOLE DEPTH	0.495625
5	PITCH ANGLE	1.271058
6	FACE ANGLE	1.291448
7	ROOT ANGLE	1.216421
8	OUT. DIA	10.508751
9	BACK ANGLE	1.271058
10	FRONT ANGLE	1.271058
11	DELTA R	0.540584
12	GR	0.063690
13	BO	1.352146
14	BI	0.881508
15	PABCP	0.136397

Bibliography

[1] *Planetary Gear Train Ring Gear and Support Structure Investigation*, Mark Valco, Ph.D. Dissertation, Cleveland State University, 1992.

[2] *Gear Tooth Stress Measurements of Two Helicopter Planetary Stages*, Krantz, T. L., NASA Technical Memorandum 105651, AVSCOM Technical Report 91-C-038, 1992.

[3] A combined surface integral and finite element solution for a three-dimensional contact problem, S. Vijayakar, *International Journal for Numerical Methods in Engineering*, vol.31, pp. 525-545, 1991.

[4] *Nonlinear and dynamic programming*, G. Hadley, Addison Wesley Publishing company, 1964.

[5] *Linear programming*, George Hadley, Addison Wesley, 1962.

[6] *Linear and Combinatorial Programming*, Katta G. Murty, John Wiley, 1976 ISBN: 0-471-57370-1.

[7] Linearization of multibody frictional contact problems, S. Vijayakar, H. Busby and D. Houser, *Computers and Structures*, vol. 29, no. 4, pp. 569-576, 1987.

[8] *Natural Frequency Spectra and Vibration Modes of Planetary Gears*, Jian Lin and Robert Parker, 1998 ASME Design Engineering Technical Conference, September 1998, Atlanta Georgia.

[9] *Gear Dynamics Experiments, Part I: Characterization of Forced Response*, Blankenship and Kahraman, ASME 7^{th} International Power Transmissions and Gearing Conference, San Diego, October 1996.

[10] *Gear Dynamics Experiments, Part II: Effect of Involute Contact Ratio*, Blankenship and Kahraman, ASME 7^{th} International Power Transmissions and Gearing Conference, San Diego, October 1996.

[11] *Gear Dynamics Experiments, Part III: Effect of Involute Tip Relief*, Blankenship and Kahraman, ASME 7^{th} International Power Transmissions and Gearing Conference, San Diego, October 1996.

[12] The use of boundary elements for the determination of the geometry factor, Vijayakar and Houser, 1986 *AGMA Fall Technical Meeting*, Paper no. 86-FTM-10.

[13] Finite element analysis of quasi-prismatic structures, S. Vijayakar, H. Busby and D. Houser, *International Journal for Numerical Methods in Engineering*, vol. 24, pp. 1461-1477, 1987.

[14] Edge effects in gear tooth contact, S. Vijayakar, *ASME* 7^{th} *International Power Transmissions and Gearing Conference*, San Diego, October 1996.

[15] Vibration Measurements on Planetary Gears of Aircraft Turbine Engines, M. Botman, *AIAA Journal*, vol. 17, no. 5, 1980.

[16] Dynamic Tooth Loads in Epicyclic Gears, F. Cunliffe, J. D. Smith, and D.B. Welbourn, *J. Eng. Ind. Trans. ASME*, May 1974.

[17] Effect of Internal Gear Flexibility on the Quassi-Static Behavior of a Planetary Gear Set, A. Kahraman, S. Vijayakar, *Transactions of the ASME*, September 2001.