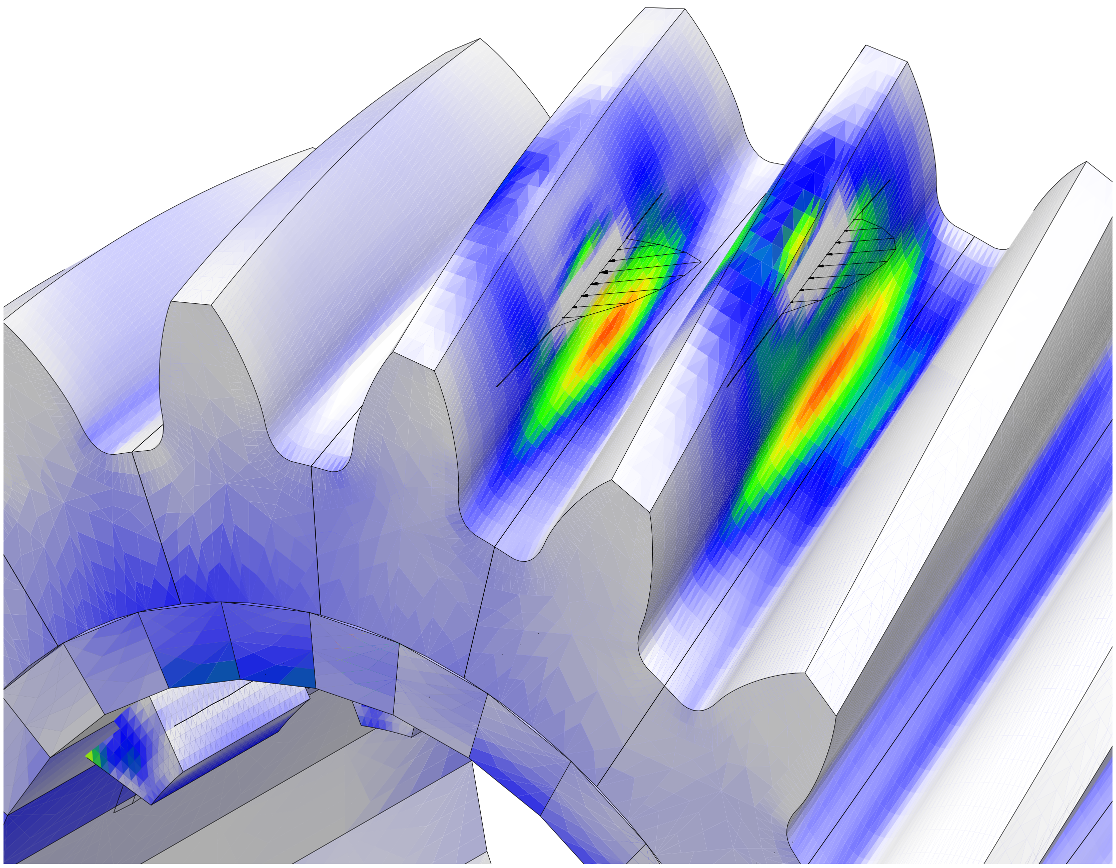
Helical3D User's Manual



Hilliard OH

February 24, 2005

Contents

List of Figures

List of Tables

Preface

We are pleased to release the *Helical3D* software for use by the gearing community. 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. The Small Business Innovation and Research (SBIR) program of the U.S. Army Research Laboratory has funded the development of much of the technology in *Helical3D*.

Sandeep Vijayakar
Samir Abad
Hilliard OH

Chapter 1

Introduction to gear geometry

A review of basic involute geometry is provided in this chapter.

1.1 Gear ratio

The primary purpose of gears is to transmit motion and at the same time, multiply either torque or speed. Torque is a function of horsepower and speed of the power source. It is an indication of the power transmitted through a driving shaft and from it the gear tooth loads are calculated. The load applied to gear trains vary from practically nothing to several tons or more. Gears properly designed and meshed together in mating pairs, can multiply the torque and reduce the higher rotational speed of a power producing source to the slower speeds needed to enable the existing power to move the load. Where application requires speed rather than torque, the process is reversed to increase the speed of the power source.

Rotational speeds of the shafts involved in power transmission are inversely proportional to the number of teeth in the gears mounted on the shafts. With the relative speed of one member of a gear pair known, the speed of the mating gear is given by

$$n_g = \frac{n_p N_p}{N_g} \tag{1.1}$$

where,

$N_p =$ Number of teeth on pinion
$N_g =$ Number of teeth on gear
$n_p =$ Revolutions per minute (RPM) of pinion
$n_g =$ Revolutions per minute (RPM) of gear

1.2 Pitch

Pitch is generally defined as the distance between equally spaced points or surfaces on a given line or curve. On a cylindrical gear it is the arclength between similar points on successive teeth and is known as circular pitch p. In Figure 1.1 the teeth are shown as helical, or at an angle to the axis of the gear cylinder. If the teeth are parallel to the axes then they are called straight or spur teeth. Then the normal circular pitch and the transverse circular pitch are equal. The axial pitch is infinite for spur gear teeth.

One of the most important pitch classifications in an involute gear is the base pitch, shown in Figure 1.2. Primarily, it is the circular pitch on the perimeter of the base circle, but by definition of the involute curve the arc distance becomes the linear normal distance between corresponding sides of adjacent teeth when raised to position as part of taut line. In spur gears there is only one base pitch to consider. In helical gears, base pitch can be defined in the section normal to the helix angle (*normal base pitch*), parallel to the gear axis (*axial base pitch*) and perpendicular to the gear axis (*transverse base pitch*). Since gear teeth are equally spaced it becomes apparent that in order to roll together properly, two gears must have the same base pitch. More specifically, two mating involute gears must have the same normal base pitch.

With the number of teeth N and the circular pitch p given, the circumference of the circle and consequently the pitch diameter D_p can be calculated from

$$D = \frac{N \times p}{\pi} \tag{1.2}$$

Diametral pitch P is the ratio of teeth to the pitch diameter in inches.

$$P = \frac{N}{D_p} \tag{1.3}$$

An industry practice is to not display the units of Diametral pitch, and assume that the units are $inch^{-1}$. This practice leads to much confusion when the unit of length being used is not inches. We encourage users to always track the units of the diametral pitch. The module m is the inverse of the diametral pitch:

$$m = \frac{D}{N} \tag{1.4}$$

Again, industry practice is to not display the units of the module m, but to assume that it is millimeters. We encourage users to explicitly keep track of the units of this parameter, especially when millimeter is not the user's unit of length.

The diametral pitch and module regulate the proportions or the size of the gear teeth. Therefore, for a known load to be transmitted, the pitch is chosen which in turn determines the number of teeth to suit the desired ratio and size of the gear. The number of teeth divided by the diametral pitch produces the diameter of the gear pitch circle. The part of the tooth above the pitch circle is called the addendum and the lower part is the dedendum.

The diametral pitch referred to is usually the pitch of the tool producing the gear teeth and is known as the generating diametral pitch. The strength required of the gear teeth, the number of teeth required to provide the given ratio, and size of the pitch circles to satisfy center distance or space requirements are controlled by the diametral pitch. The load to be transmitted by the gear teeth controls the tooth thickness which is also regulated by the diametral pitch.

1.3 Pressure angles

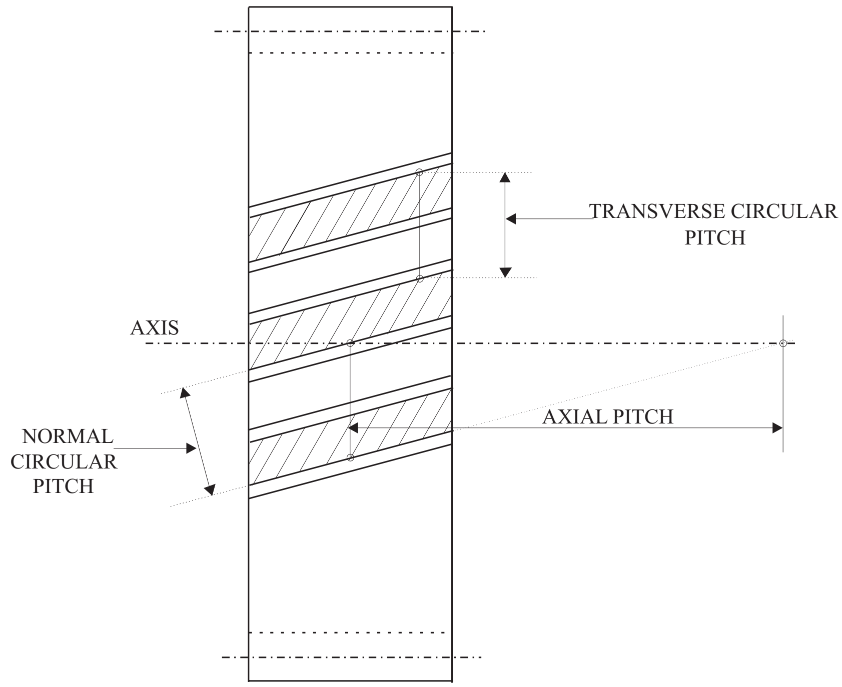


Figure 1.1: Pitch definitions related to helical teeth.

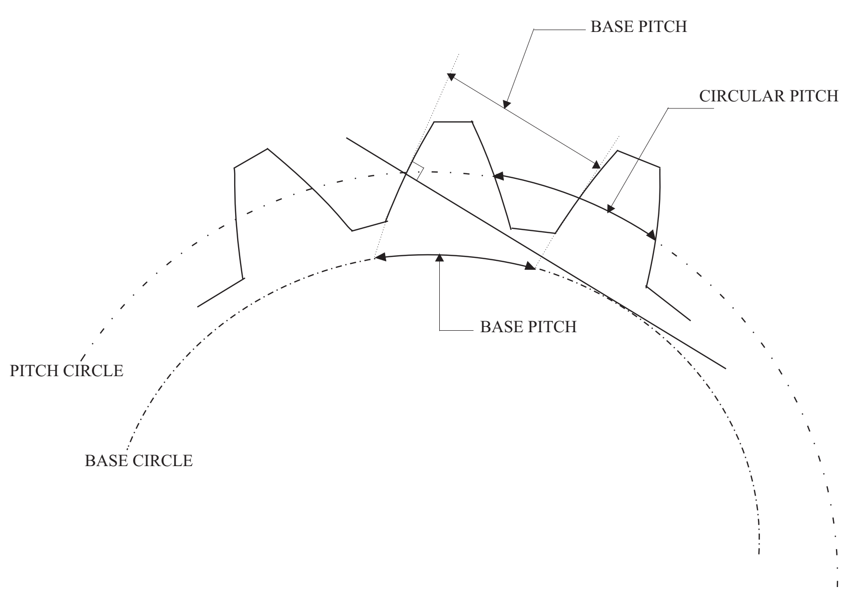


Figure 1.2: The base pitch and the circular pitch.

An involute curve is evolved from origin point A on a base circle shown in Figure 1.3. The point P on a taut line containing point B describes the curve. The taut line is tangent to the base circle at point B, and normal to the involute curve at P. This line segment BP is the radius of curvature ρ_c of the involute curve at point P and is equal in length to the arc AB. The angle θ subtended by the arc AB is the Involute roll angle of the involute to the point P. The angle between OP (radius r) and OB (base radius r_b) is the pressure angle ϕ at point P.

When two involute curves are brought together as profiles of gear teeth and are made tangent at a point P, the pressure angle ϕ is equal on both members (Figure 1.4). The line BB' is the common normal passing through the point of contact P and is tangent to both the base circles. All contact and tooth action will take place along the common normal. If one member is rotated, the involute curves will slide together and drive the other member in the opposite direction. The pressure angle through the point of contact of a pair of involute curves depends on the distance between the centers of their respective base circles. A gear does not really have a pressure angle until its involute curved profile is brought in to contact with a mating curve as shown in Figure 1.4. At that time the pressure angle ϕ becomes the operating or rolling pressure angle between the mating gears. For an operating center distance C, and base circle radii r_{bp} and r_{bg} of the pinion and gear respectively, the operating pressure angle ϕ_{op} is determined by the expression.

$$cos\phi_{op} = \frac{r_{bp} + r_{bg}}{C} \tag{1.5}$$

In helical gears, pressure angles are defined in three planes. The transverse pressure angle is defined in the plane normal to the gear axis or parallel to the gear face. Normal pressure angle is in the plane or section which is normal or perpendicular to the helix.

The pressure angle at any radius r greater than the base radius r_b is given by:

$$cos\phi = \frac{r_b}{r} \tag{1.6}$$

1.4 Spur gear calculations

Some basic equations for spur gear calculations are given below.

For Spur gears with Diametral Pitch P, Number of Teeth N, Pressure Angle ϕ, Addendum constant A:

In English units (using diametral pitch),

$$\begin{aligned}
\text{Pitch Diameter}, D_p &= \frac{N}{P} \\
\text{Addendum}, a &= \frac{A}{P} \\
\text{Outer Diameter}, D_{out} &= D_p + 2a \\
\text{Base Diameter}, D_b &= D_p \cos\phi \\
\text{Circular Pitch}, p &= \frac{\pi}{P}
\end{aligned}$$

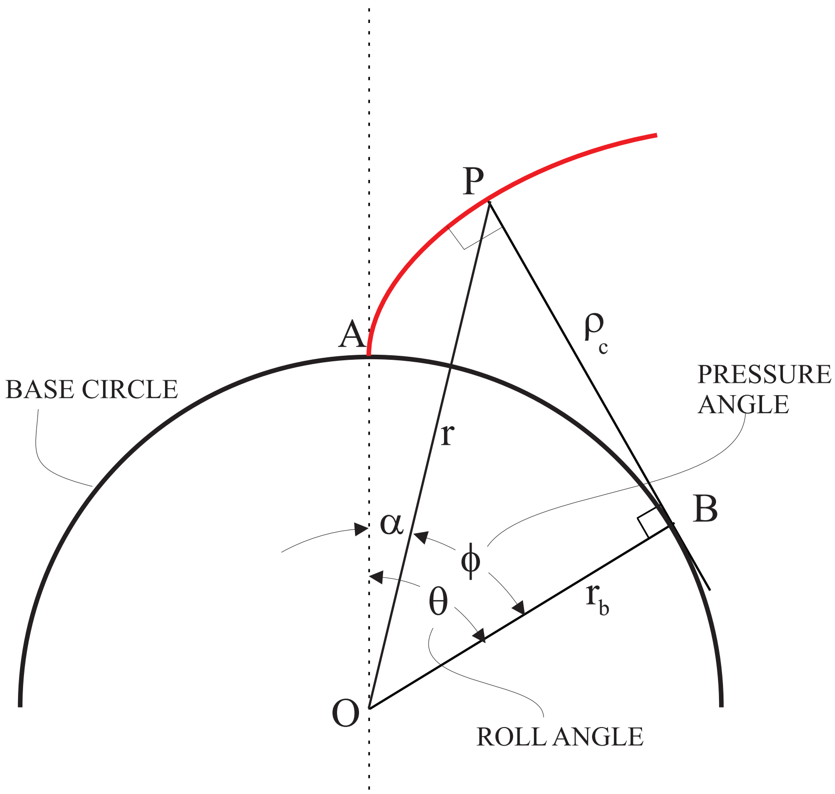


Figure 1.3: The roll angle and the pressure angle.

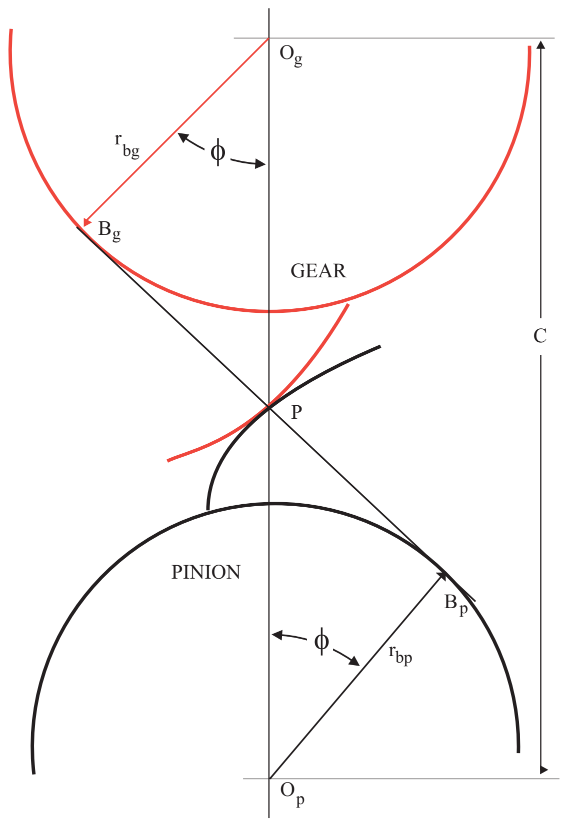


Figure 1.4: Pressure angles of pinion and gear at the contact point.

In metric units (using module),

$$\begin{aligned}\text{Pitch Diameter}, D_p &= mN \\ \text{Addendum}, a &= mA \\ \text{Outer Diameter}, D_{out} &= D_p + 2m \\ \text{Base Diameter}, D_b &= D_p \cos\phi \\ \text{Circular Pitch}, p &= \pi m\end{aligned} \tag{1.7}$$

1.4.1 Operating and generating parameters

Of all the above dimensions, the base radius r_b and the number of teeth N are the only absolute properties of the spur gear. The diametral pitch P (or module m), the pressure angle ϕ and the thickness t depend on the particular pitch circle that they are measured on. Unless otherwise specified, P, m, ϕ and t refer to the *generating diametral pitch*, *generating module*, *generating pressure angle*, and the *generating thickness*, respectively and are measured on the generating pitch circle. If these are are measured on the operating pitch circle, then they are called the *operating diametral pitch* P_{op}, *operating module* m_{op}, *operating pressure angle* ϕ_{op}, *operating tooth thickness* t_{op}, respectively.

To convert the operating pressure angle ϕ_{op} to ϕ first compute the base diameter D_b:

$$D_b = \frac{N}{P_{op}} \cos\phi_{op}$$

then to calculate ϕ use

$$D_b = \frac{N}{P} \cos\phi$$

1.4.2 Involute roll angle

The Involute angle θ at any diameter D is given by

$$\theta = \sqrt{(\frac{D}{D_b})^2 - 1}$$

In terms of the radius r at an arbitrary point and the base radius r_b, Involute angle θ is given by

$$\theta = \sqrt{(\frac{r}{r_b})^2 - 1}$$

1.4.3 Tooth thickness

The thickness t of an involute tooth depends on the diameter D (or radius r) of the circle on which it is measured. If t_1 is the thickness measured on a circle with diameter D_1, radius r_1 and t_2 is the thickness measured on another circle with diameter D_2 and radius r_2, it is easy to compute one from the other. First calculate the involute roll angles for the involute on the two circles:

$$\theta_1 = \sqrt{(\frac{r_1}{r_b})^2 - 1}$$
$$\theta_2 = \sqrt{(\frac{r_2}{r_b})^2 - 1}$$

The relationship between the thicknesses t_1 and t_2 is given by:

For an external spur gear:

$$\frac{t_1}{2r_1} + (\theta_1 - \tan^{-1}(\theta_1)) = \frac{t_2}{2r_2} + (\theta_2 - \tan^{-1}(\theta_2))$$

For an internal spur gear:

$$\frac{t_1}{2r_1} - (\theta_1 - \tan^{-1}(\theta_1)) = \frac{t_2}{2r_2} - (\theta_2 - \tan^{-1}(\theta_2)) \tag{1.8}$$

This formula can be used to convert thicknesses from the generating circle to the operating circle, and vice-versa.

1.5 Helical gear calculations

Some basic equations for helical gear calculations are given below.

For Helical gears, the helix angle ψ is non zero. This helix angle is not an absolute property of the gear, but varies depending on the diameter D (or radius r) at which it is measured. The lead L of the helical gear, on the other hand, is an absolute property of the gear. The helix angle ψ measured at a diameter D is related to the lead L by:

$$\psi = \tan^{-1} \frac{\pi D}{L} \tag{1.9}$$

The relationship between the helix angle at the generating and operating diameter is therefore:

$$\frac{D_{op}}{\tan \psi_{op}} = \frac{D_g}{\tan \psi_g} \tag{1.10}$$

At any diameter, the diametral pitch (or module), pressure angle and tooth thickness may be measured either in the transverse plane or in the normal plane. The normal diametral pitch P_n (normal module m_n), the normal pressure angle ϕ_n and the normal tooth thickness t_n are related to the transverse diametral pitch P_t (transverse module m_t), the transverse pressure angle ϕ_t

Table 1.1: Spur Gear Nomenclature

Item	Description
n_p	Revolutions per minute (RPM) of the pinion
n_g	Revolutions per minute (RPM) of the gear
C	Operating center distance
N	Number of teeth
N_p	Number of teeth on the pinion
N_g	Number of teeth on the gear
θ	Involute roll angle at a point on the involute
r	Radius at a point on the involute
ρ_c	Radius of curvature at a point on the involute
p	Circular pitch
P	Diametral pitch
m	Module
ϕ	Pressure angle at a point on the involute
t	Tooth thickness measured at a point on the involute
P_{op}	Operating diametral pitch
m_{op}	Operating module
ϕ_{op}	Pressure angle at the operating pitch point
t_{op}	Tooth thickness measured at the operating pitch point
r_b	Base radius of the involute
A	Addendum constant
a	Addendum
D_p	Pitch Diameter
D_{out}	Outer diameter
D_b	Base diameter

and the transverse circular (arc) tooth thickness t_t by:

$$\begin{aligned} P_t &= P_n \cos\psi \\ m_t &= m_n / \cos\psi \\ \tan\phi_t &= \tan\phi_n / \cos\psi \\ t_t &= t_n / \cos\psi \end{aligned}$$

ψ is measured at the *same* diameter as the other parameters.

Some other useful relationships involving the number of teeth N, the face width w and addendum constant A are:

$$\begin{aligned} \text{Pitch Diameter}, D_p &= \frac{N}{P_t} \\ \text{Addendum}, a &= \frac{A}{P_n} \\ \text{Outer Diameter}, D_{out} &= D_p + 2a \\ \text{Base Diameter}, D_b &= D_p \cos\phi_t \\ \text{Normal Circular Pitch}, p_n &= \frac{\pi}{P_n} \\ \text{Axial Pitch}, p_x &= \frac{\pi}{P_n \sin\psi} \\ \text{Transverse Circular Pitch}, p_t &= \frac{\pi}{P_t} \\ \text{Transverse Circular Pitch on Base Circle}, p_{bt} &= p_t \cos\phi_t \end{aligned}$$

The normal diametral pitch P_n, the normal pressure angle ϕ_n and the normal tooth thickness t_n are the parameters that are preferred for describing a helical gear. This is because these normal parameters are directly related to the normal parameters of the hob that is used to cut the helical gears.

$$\begin{aligned} \phi_n &= \phi_{nhob} \\ P_n &= P_{nhob} \\ m_n &= m_{nhob} \\ t_n &= w_n \end{aligned}$$

where w_n is the normal space width of the hob.

1.5.1 Contact ratio

The length of contact Z, is:

$$Z = \frac{\sqrt{D_{og}{}^2 - D_{bg}{}^2} + \sqrt{D_{op}{}^2 - D_{bp}{}^2}}{2} - C \sin\phi_t \tag{1.11}$$

The involute (or profile) contact ratio M_p is:

$$M_p = \frac{Z}{p_{bt}} \tag{1.12}$$

The face contact ratio M_f is:

$$M_f = \frac{w \tan\psi}{p_t} \tag{1.13}$$

The total contact ratio M is

$$M = M_f + M_p \tag{1.14}$$

Table 1.2: Helical Gear Nomenclature

Item	Description
w	Face width
L	Lead
ψ	Helix angle
P_n	Normal diametral pitch
P_t	Transverse diametral pitch
m_n	Normal diametral pitch
m_t	Transverse diametral pitch
ϕ_n	Normal pressure angle
ϕ_t	Transverse pressure angle
t_n	Normal tooth thickness
t_t	Transverse circular (arc) tooth thickness
p_n	Normal circular pitch
p_t	Transverse circular pitch
p_{bt}	Transverse circular pitch on base circle
p_x	Axial circular pitch
w_n	Normal space of the hob
L	Length of contact
M_p	Involute or profile contact ratio
M_f	Face or profile contact ratio

Chapter 2

Software Architecture

The *Helical3D* package consists of a group of programs for the analysis of external and internal helical gear pairs. Figure 2.1 shows the program modules in this package, and how they interact.

Calyx is a powerful contact analysis code capable of analyzing a variety of contact problems, including 2D and 3D static and dynamic analysis of systems such as gears, compressors, and brakes. Because *Calyx* is designed 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 or a command line interface. It translates the user's commands into the appropriate programming language statements and sends them on to *Calyx*. A typical user need not even know that *Calyx* is running in the background.

In addition to the user interface, *Multyx* also has built-in model generators. The internal and external helical gear tooth models, the web, and spline 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*. In *Helical3D*, *Multyx* is customized with model generators needed for spur and helical gear applications.

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 a sample of 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=PINION_SURFACE1_GEAR_SURFACE1)
MEMBER          Member (Currently=GEAR)
TOOTHBEGIN      80              Tooth no. or instance no. of surface.
TOOTHEND        2               Tooth no. or instance no. of surface.
```

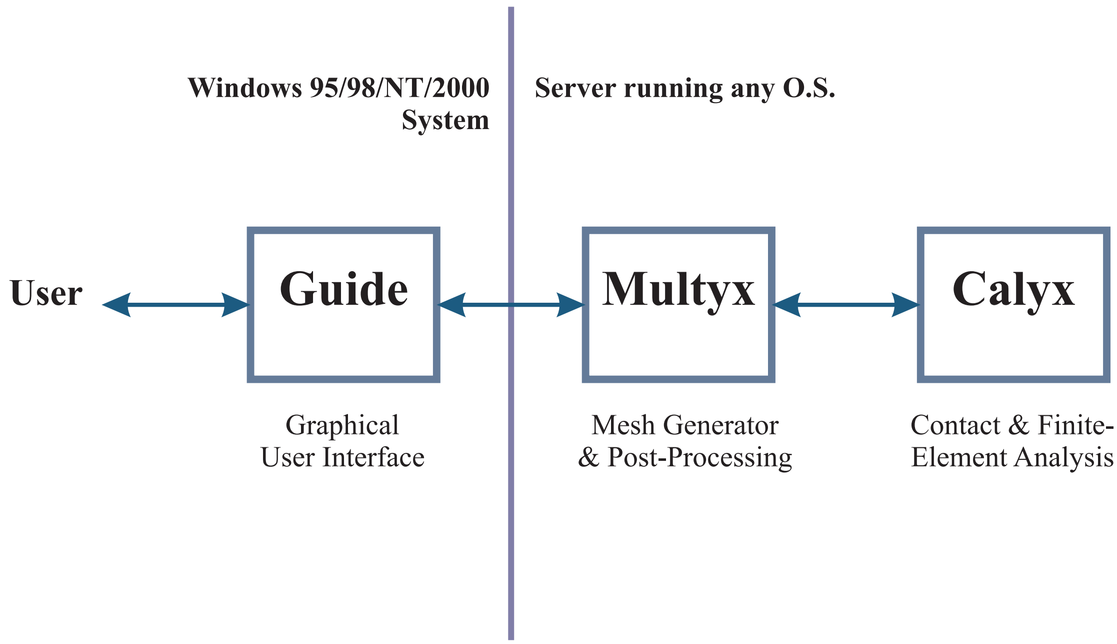


Figure 2.1: The computer programs in the *Helical3D* analysis package.

```
BEGINSTEP       1               Time/Roll angle step at which to begin search.
ENDSTEP         11              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          (FALSE)         Whether to smooth the pressure contours.
OUTPUTTOFILE    Whether to write data to file. (Disabled)
MultyX.PostProc.1/11.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* provides 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. These formats can be used in documentation. In *Helical3D*, *Guide* is customized for spur and helical gear applications.

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/XP systems only.

This manual describes the various features of the *Helical3D* package.

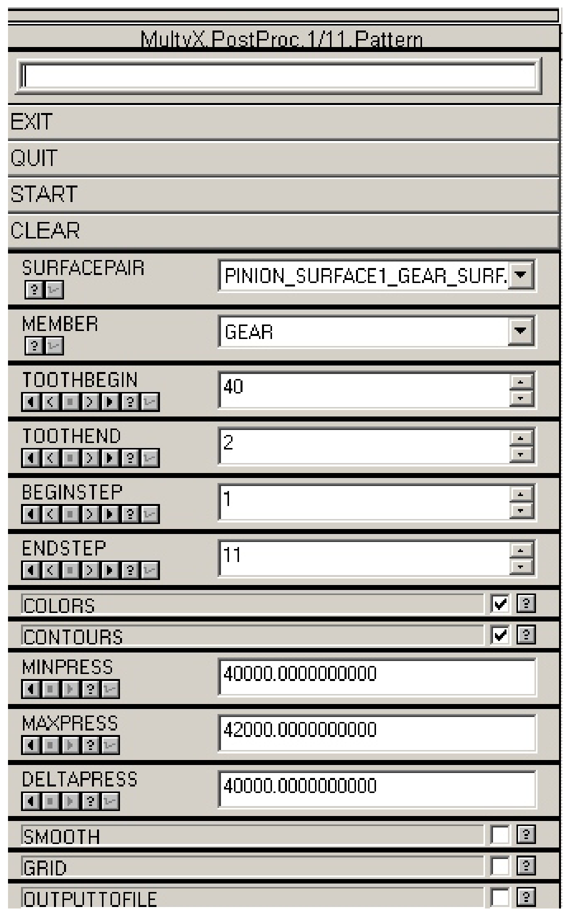


Figure 2.2: A typical Guide menu presented to the user by *Guide*

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 user inputs are consistent with this chosen system. The user is free to choose any units for force, time and length. All other 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 *Helical3D*, the pinion and gear are treated as separate bodies. In addition, if splined supports are used for the pinion, an additional body called the pinion shaft body is created. The pinion body and pinion shaft body interact through contact at the splines. Similarly, if splines are used to support the gear, an additional body called the gear shaft body is created. This gear shaft body interacts with the gear body through contact at the splines.

Thus *Helical3D* creates 2, 3 or 4 bodies depending on the boundary conditions applied to the pinion and gear.

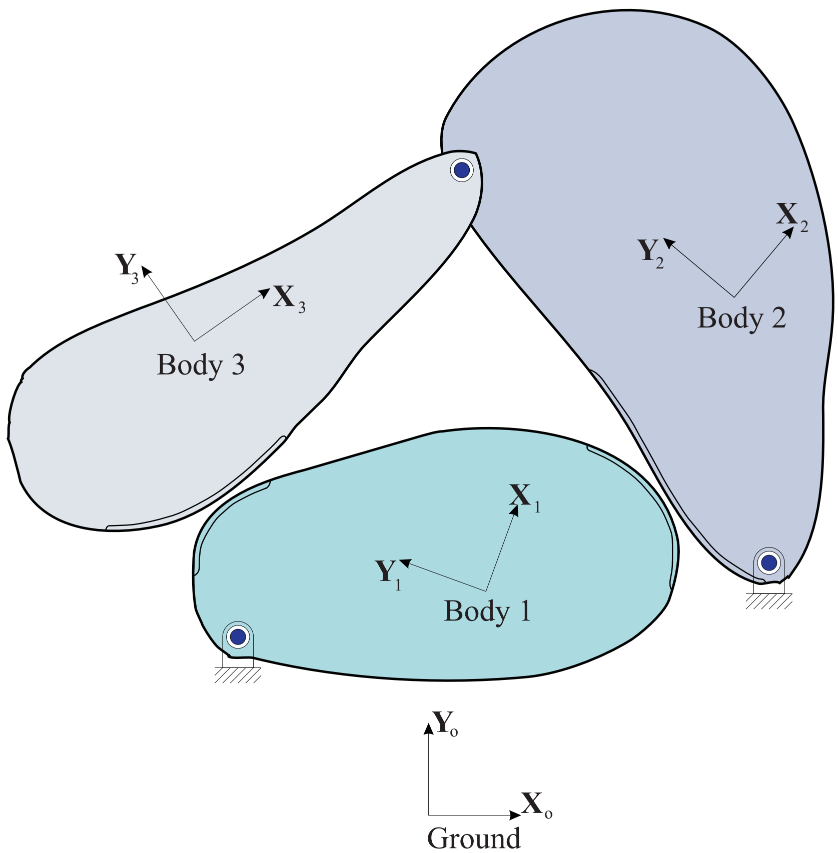


Figure 3.1: A multi-body system

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, θ_y and θ_z (Figure 3.2).

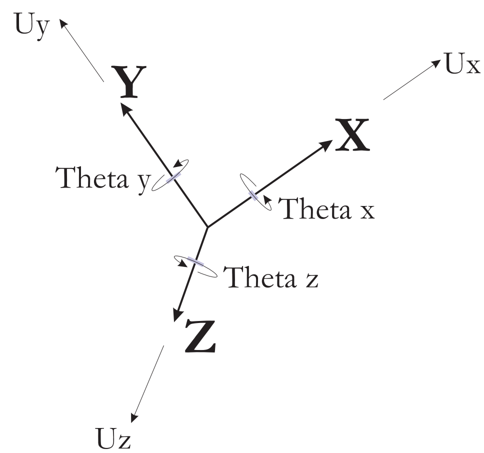


Figure 3.2: Reference frame degrees of freedom

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.

Figures 3.3 and 3.4 show how *Helical3D* sets up the pinion and gear reference frames relative to the fixed reference frame in an external and internal gear set, respectively. The fixed frame is located with its origin at the operating pitch point. Its Z axis is parallel to the axes of rotation of the pinion and gear. Its Y axis is along the center distance direction. The pinion and gear reference frames have their origins at their centers of rotation, with the Z axis being the axis of rotation. At time $t = 0$, the pinion and 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.

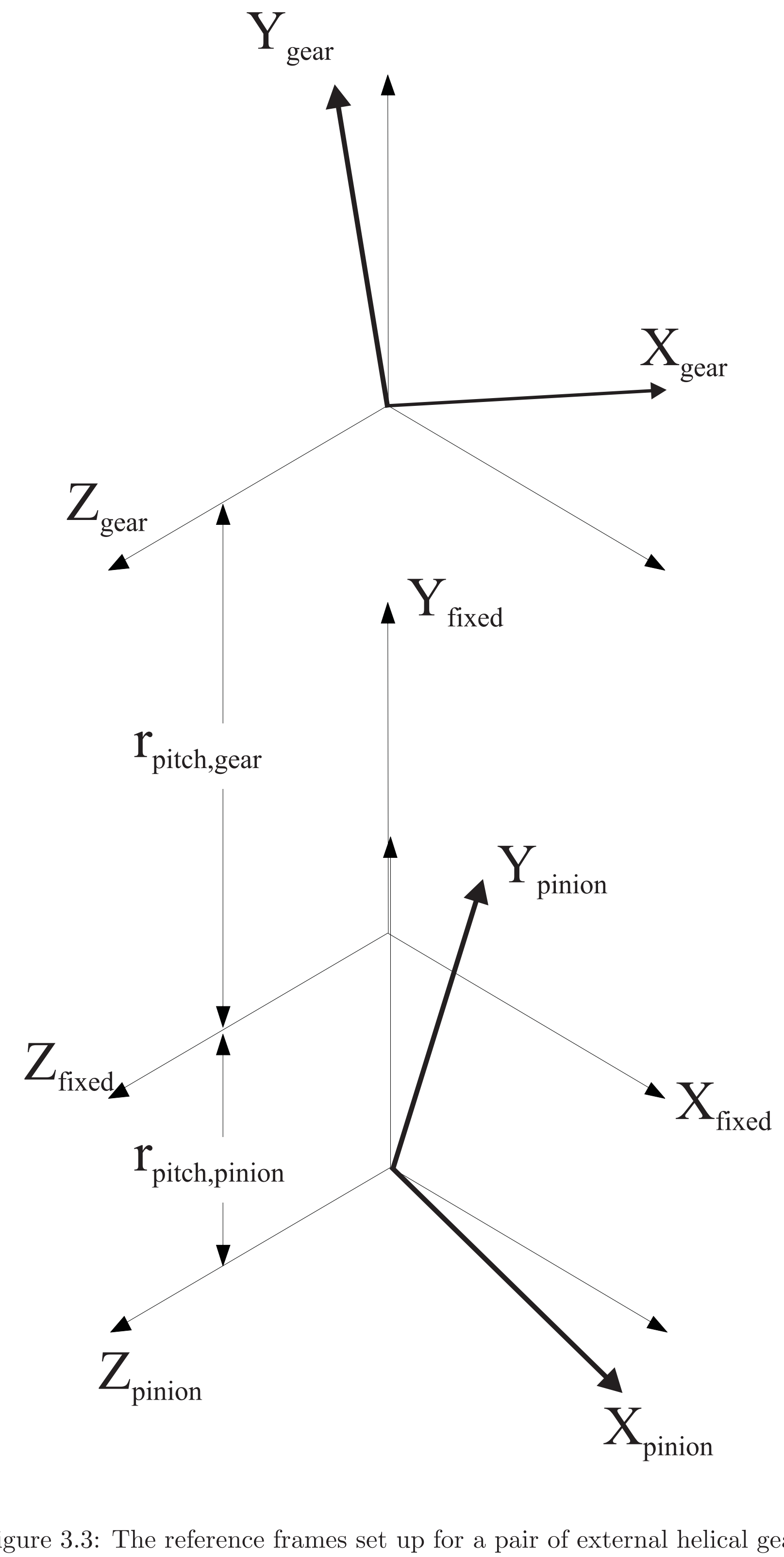


Figure 3.3: The reference frames set up for a pair of external helical gears

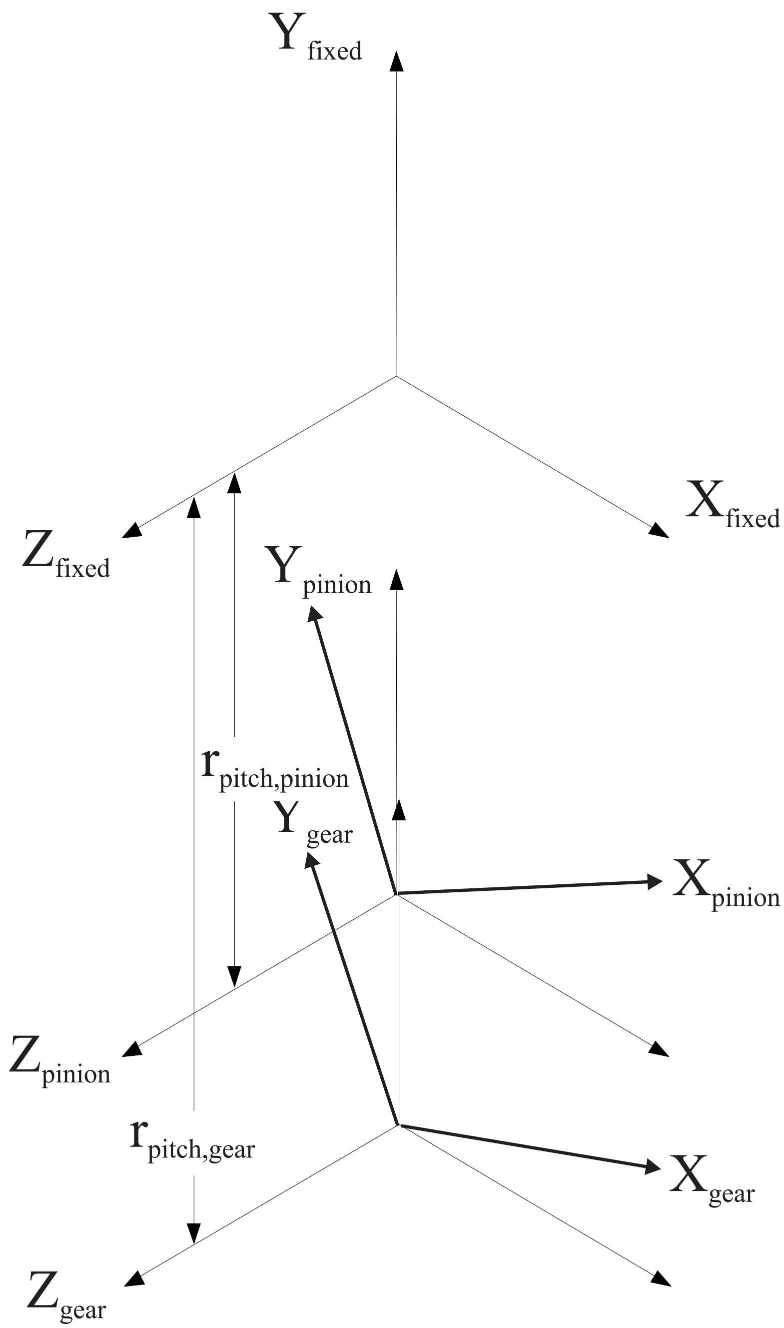


Figure 3.4: The reference frames set up for an internal helical gear and external pinion.

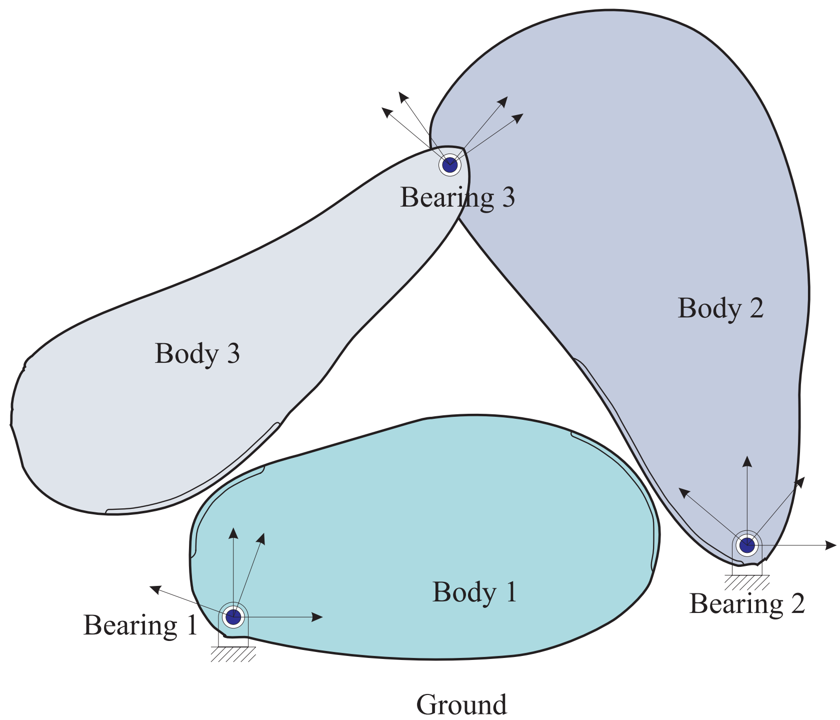


Figure 3.5: Bearing connections in the multi-body model.

3.4 Bearings

In a multi-body system, bodies can also interact through bearings (Figure 3.5).

Bearings are treated as a stiffness connection between two bearing 'races' (Figure 3.6). Each race has an attached reference frame. The race is treated as a rigid body, and the six degrees of freedom of the first bearing race and the six degrees of freedom of the second bearing race are inter-related through a stiffness matrix.

The six degrees of freedom are the three translation degrees of freedom U_x, U_y and U_z, and three rotation degrees of freedom θ_x, $\theta_x$and and θ_z (Figure 3.7). The degrees of freedom represent the motion of race 1 relative to race 2. The components are measured in the reference frame attached to race 2.

The bearing may also generate internal reaction forces and moments. The six components of bearing reaction consist of three forces F_x, F_y and F_z, and the three moments M_x, M_y and M_z. Again, these reactions are those exerted by race 1 on race 2. The components are computed in the reference frame of race 2.

Depending on the user's inputs, *Helical3D* can generate a bearing for the pinion and one for the gear. For the pinion bearing, race 1 is attached to the pinion and race 2 is attached to the fixed body. The gear is treated similarly.

Since *Helical3D* attaches race 2 of both bearings to ground, the bearing rigid body deformation and reaction components can be interpreted as the forces and moments acting parallel to the axes of the fixed reference frame.

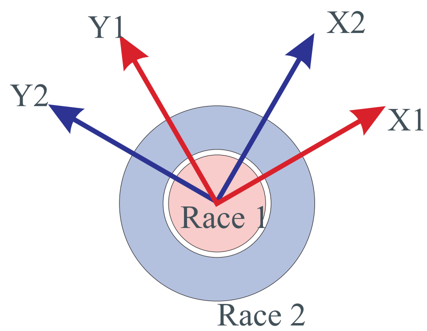


Figure 3.6: Bearing races

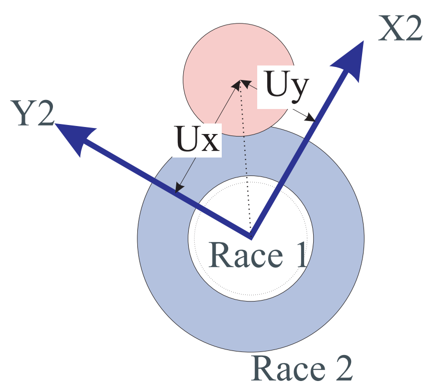


Figure 3.7: Bearing deformation

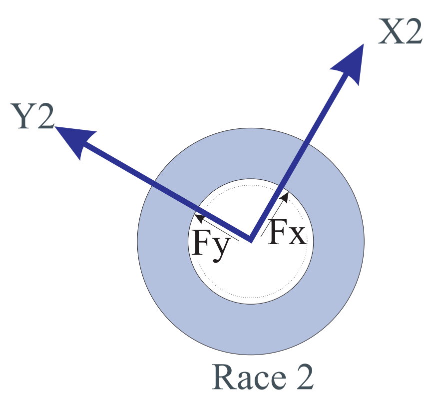


Figure 3.8: Bearing reaction

Chapter 4

Getting Started

4.1 Installing *Helical3D*

The installation process varies with the type of computer system. For most Microsoft Windows based computers, the installation is completed by starting an executable file called `Helical3D.EXE`. After all the software components are installed, the user needs to register a license key with the software.

If this step was skipped during the installation process, it can be completed by using the registration dialog box. To bring up the registration dialog box, hit the Register icon under the Start button (Figure 4.1). The rigistration dialog box is shown in Figure 4.2. The license The license key should be all in one line with no spaces.

4.2 Starting *Helical3D*

To start *Helical3D*, first hit the *Helical3D* icon under the Start button (Figure 4.3). Then hit the Connect icon (Figure 4.4) to bring up the main menu.

4.3 The main menu

After the *Helical3D* package is started, the main menu shown in Figure 4.5 comes up.

All user provided data is saved in a file called the session file. The name of this session files can be changed by typing the name in the SESFILENAME box. Changing the files 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.

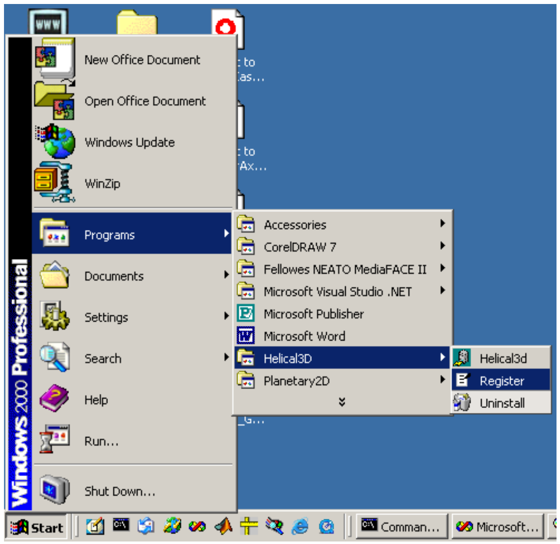


Figure 4.1: The Register icon.

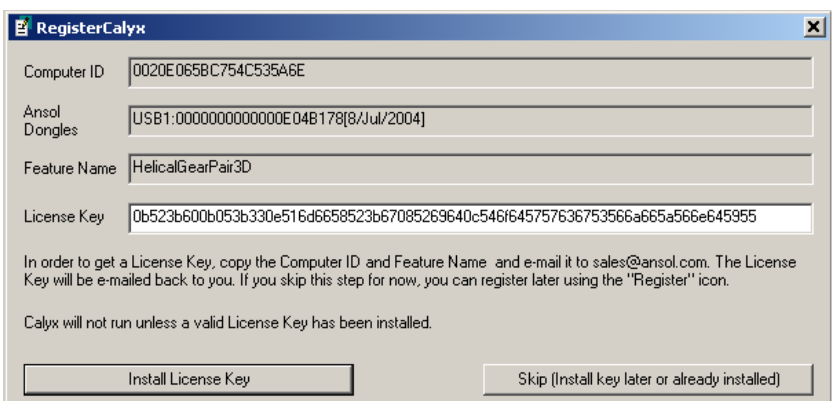


Figure 4.2: The license key registration dialog box.

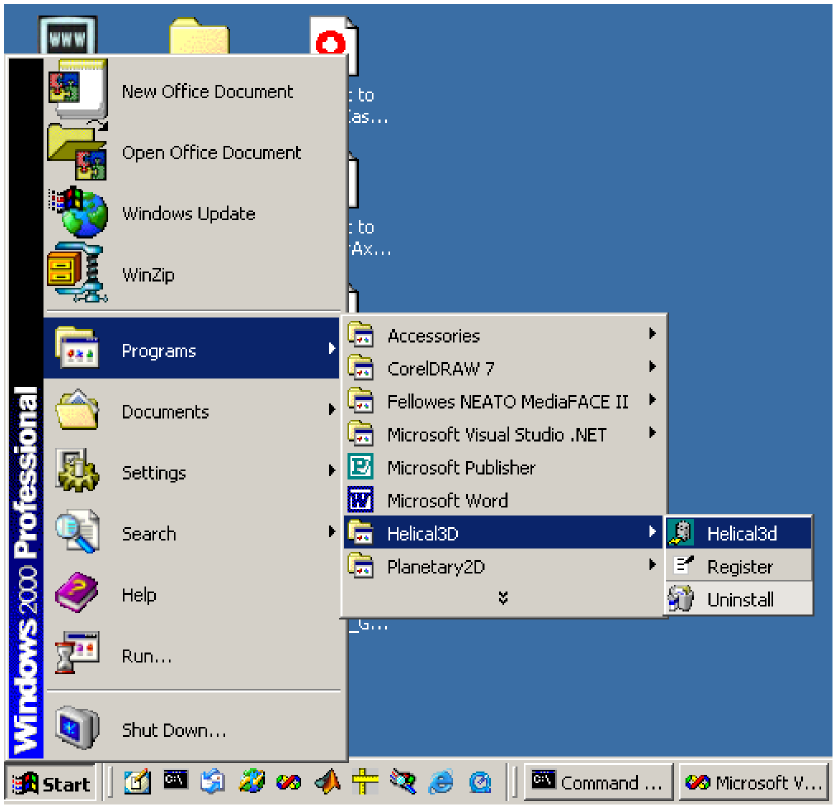


Figure 4.3: The *Helical3D* icon.

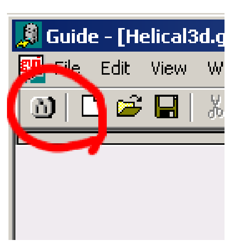


Figure 4.4: The Connect icon.

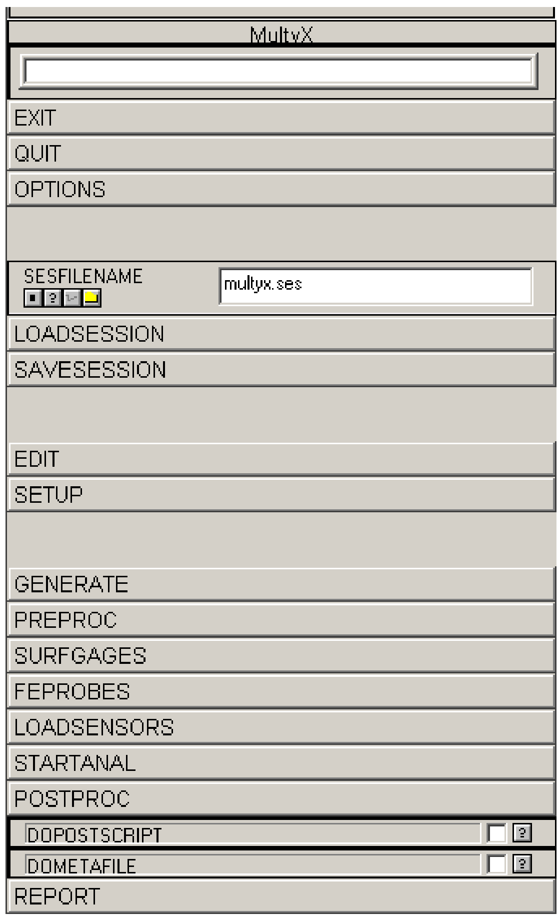


Figure 4.5: The main menu.

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 used to control the data created by the analysis.

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

Chapter 5

The Graphical User Interface

Helical3D's user interface is presented by *Guide* in graphical form, as shown in Figure 5.1. *Helical3D* 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 clicking the appropriate "Error", "Information" or "Warning" tab. Graphical information sent by *Helical3D* is directed to a graphics window.

5.1 Menu command items

In the example shown in Figure 5.1, the large buttons such as those labeled EXIT ,QUIT, OPTIONS, LOADSESSION, EDIT send commands to *Helical3D* when hit by the user. In response to the command, *Helical3D* 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.

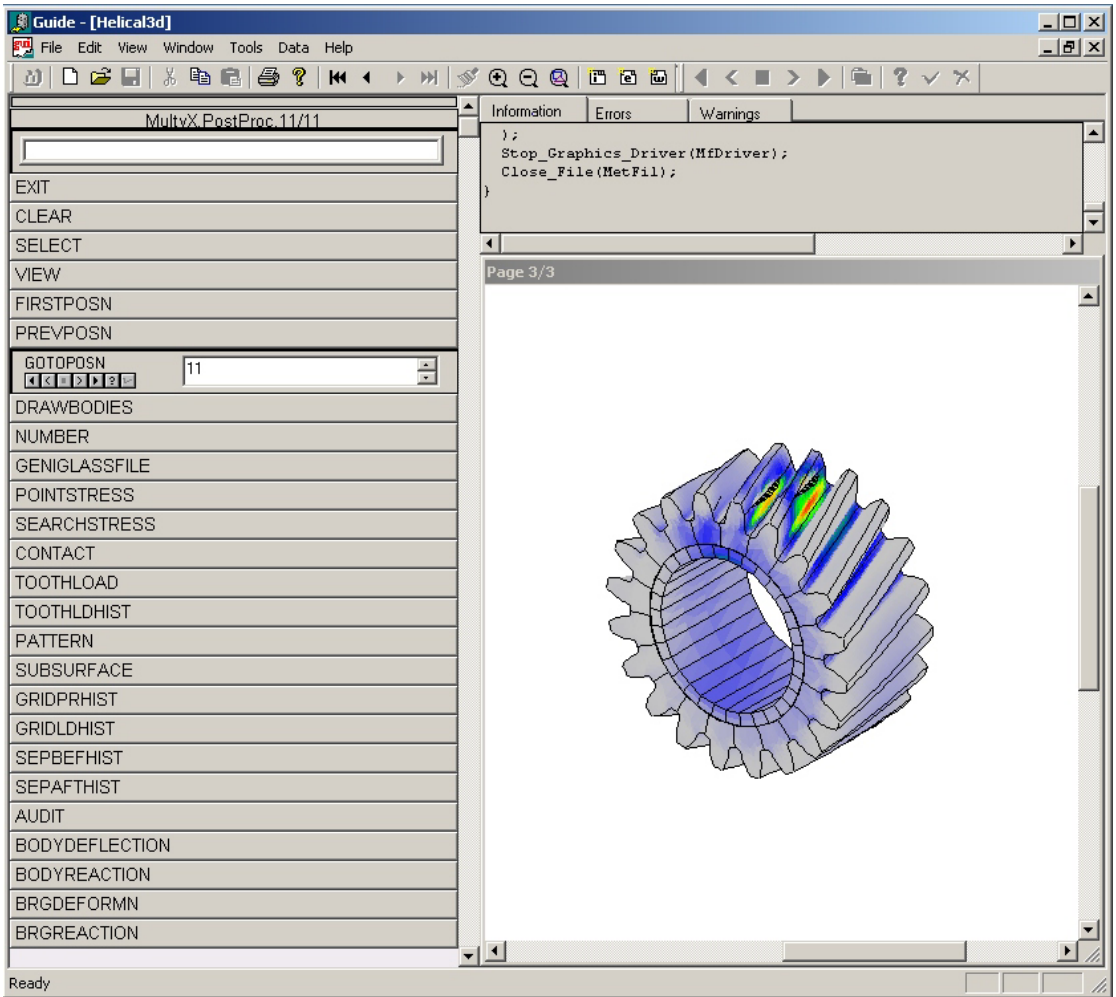


Figure 5.1: *Helical3D*'s user interface.

5.2 Integer menu items

Figure 5.2: An integer data entry box

Integer data items are entered through a dialog box of the kind shown in Figure 5.2. 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.

5.3 Floating point menu items

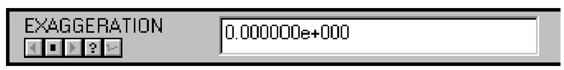


Figure 5.3: An floating point data entry box

Floating point data is entered through the dialog box shown in Figure 5.3.

5.4 Boolean menu items

Figure 5.4: An boolean data entry box

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 5.4.

5.5 String menu items

String data items contain ASCII strings. The dialog box shown in Figure 5.5 allows the user to enter string type data.

Figure 5.5: A string data entry box

5.6 Switch type menu items

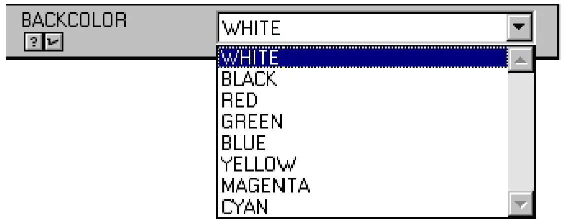


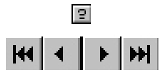
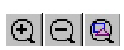
Figure 5.6: An switch type data entry box

The last kind of data item is of the ‘switch’ type. The value of a switch menu item can be selected from a fixed set of valid choices. The choice is made through a drop down list as shown in Figure 5.6.

5.7 Commonly occurring buttons

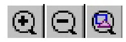
The data entry dialog boxes use a few small buttons as short cuts for common tasks as shown in the Table 5.1. Some of these buttons may be disabled depending upon the particular item and its value.

Table 5.1: 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
	Browse for file name
	Get additional information
	Change the current graphics page
	Change the zoom level
	Refresh the graphics page

5.8 Graphics

Guide directs the graphical output from *Helical3D* to a graphics window. The graphics are stored as separate pages. a new page is started when *Helical3D* 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 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 6

Running *Helical3D* without the Graphical User Interface

Sometimes it is convenient to run *Helical3D* without the interactive graphical user interface. This is useful if a batch file is needed to run several jobs in a batch queue.

6.1 Invoking *Helical3D*

To start *Helical3D*, open a command prompt window, and invoke the executable file `multyx`. Of course the complete path name to the directory (folder) containing the file `multyx.exe` must be provided unless the directory is part of the PATH environment variable. The directory name should not contain any spaces.

If the directory containing `multyx.exe` is `Program Files\Ansol\Helical3D`, then the short version of this `\progra~1\ansol\Helical3D` must be used. One parameter, `helicalpair3dcr.sdf` must be provided on the command line:

```
\progra~1\ansol\Helical3D\multyx helicalpair3dcr.sdf
```

The main menu of *Helical3D* comes up in the command line mode. The names of the commands and data items is the same as when running under *Guide*. You may type `?` or `HELP` for options at any prompt.

```
MultyX v.2.01, Copyright Advanced Numerical Solutions Jun 24 2004
Computer ID: 0020E065BC754C535A6E
Read data from session file:
multyx.ses
MultyX>EDIT PINION
MultyX.Edit.Pinion>TOOTH
MultyX.Edit.Pinion.Tooth>?
MENU            Show menu
?               Show menu
HELP            Show menu
EXIT            Accept changes and return.
QUIT            Return after discarding changes in this menu and all sub-menus.
MODFN           Surface modifications
NTEETH          20              The number of teeth on pinion
NFACEELEMS      4               The number elements across face
COORDORDER      10              limit on order of coord axodes
```

```
DISPLORDER       3               Limit on displ. order of axodes.
PLANE            TRANSVERSE  The plane used to define DP., P.A. & thickness
XVERSEDIAMPITCH 1.0000000E+001  The xverse gen. diametral pitch
XVERSEPRESSANGLE2.0000000E+001  The xverse gen. press. angle (Deg)
XVERSETHICKNESS 1.5708000E-001  Transverse thickness of the pinion tooth
FACEWIDTH       1.0000000E+000  The face width of the pinion
HAND            (LEFT)          The hand of the pinion (Left/Right)
HELIXANGLE      2.0000000E+001  The helix angle of the pinion
PROFILETYPE     SIMPLE  Type of profile on the pinion
RACKTIPRAD      2.0000000E-002  Rack tip radius for the pinion
OUTERDIA        2.1800000E+000  Outer dia. of the pinion
ROOTDIA         1.7600000E+000  Root dia.of the pinion
RIMDIA          1.4000000E+000  Rim diameter of the pinion
YOUNGSMOD       3.0000000E+007  Young's modulus for the pinion
POISSON         3.0000000E-001  Poisson's ratio for the pinion
DENSITY         3.0000000E-001  Density for the pinion
ALPHA           1.0000000E-003  Damping constant alpha for the pinion
BETA            1.0000000E-007  Damping constant beta for the pinion
MESHFILE        (pinion.msh)    Mesh file name for pinion
TEMPLATE        (medium.tpl)    Template file name for pinion
MultyX.Edit.Pinion.Tooth>NTEETH 22
MultyX.Edit.Pinion.Tooth>EXIT
MultyX.Edit.Pinion>EXIT
MultyX.Edit>EXIT
MultyX>EXIT
MultyX.ExitConfirm>OK
```

All standard techniques for batch processing, including redirection of input and output may be used. If the file `input.txt` contains the list of commands for *Helical3D* and if you want to send the output to the file `output.txt`, then the following command will do it for you:

```
\progra~1\ansol\Helical3D\multyx helicalpair3dcr.sdf <input.txt >output.txt
```

The input file `input.txt` might contain commands such as:

```
SESFILENAME case1.ses
LOADSESSION
STARTANAL
EXIT OK
```

In this case, the commands simply load all data and start an analysis.

Chapter 7

Building a Model

All data describing the model is entered in sub-menus of the EDIT menu. Figure 7.1 shows the EDIT menu. In this 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 after saving the 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.

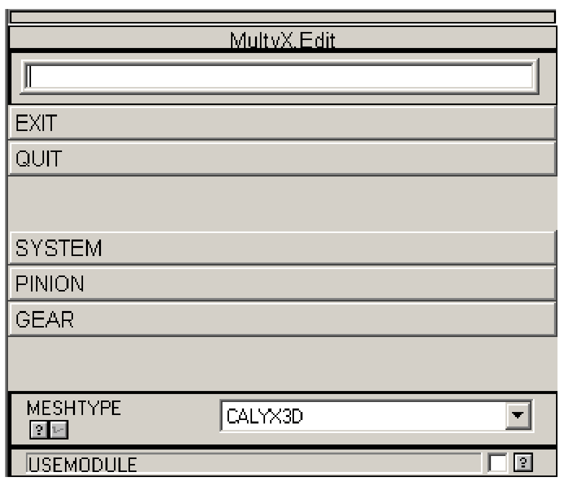


Figure 7.1: The EDIT menu.

Depending on the features licensed at a particular site, an aditional item MESHTYPE may also be visible in this EDIT menu. This allows a user to select the kind of model to be generated. Choices are CAPP and CALYX3D. CAPP is an older analysis and post-processing package. Unless otherwise stated, it will be assumed that the user has selected CALYX3D.

If the flag USEMODULE is checked, then the user can use the gear module instead of diametral pitch. Users in countries where module is preferred should check this item. In the rest of this manual, we will assume that this flag is not checked.

7.1 System level data

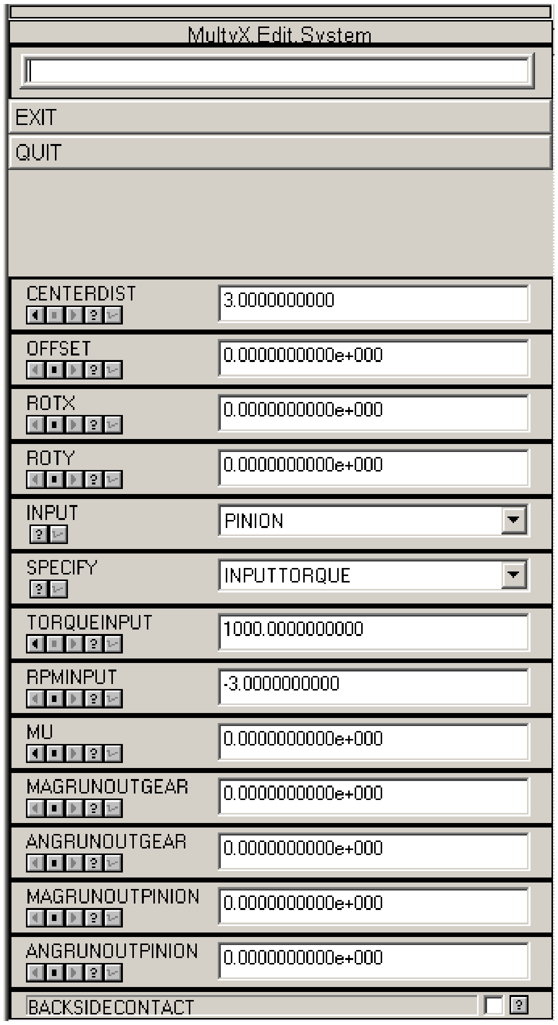


Figure 7.2: The system data menu.

The SYSTEM command in the EDIT menu of Figure 7.1 leads to the SYSTEM menu shown in Figure 7.2. The parameters in this menu are summarized in Table 7.1.

If the MESHTYPE option in the main menu was set to CAPP, then the name of a configuration file can be specified in the CONFIGFILE item.

CENTERDIST is the operating center distance. This should always be a positive value. OFFSET (Figure 7.3) is the axial offset of the gear with respect to the pinion. When this value is zero, the mid face cross-sections of the pinion and gear coincide. When it is positive, the gear is shifted by this amount in the $+Z$ direction.

ROTX and ROTY (Figure 7.4) are the angular misalignment values in Degrees for the pinion about the X and Y axes, respectively. The Z axis is the axis of rotation, the Y axis is along the center distance direction, pointing from the pinion to the gear. The X axis is perpendicular to these two. The three axes form a right handed system. The right hand rule is used to determine the sign of this angle. The misalignment values are in Degrees.

The INPUT switch selects whether the PINION or the GEAR acts as the input for power. The torque and angular speed directions of the input member are the same. They are opposite

Table 7.1: System configuration parameters

Item	Description
CONFIGFILE	String, Configuration file for CAPP (if MESHTYPE=CAPP)
CENTERDIST	Float, The operating center distance
OFFSET	Float, The axial offset of the gear wrt. the pinion
ROTX	Float, The radial misalignment (Deg) of the pinion.
ROTY	Float, The tangential misalignment (Deg) of the pinion.
INPUT	Switch, Which body is the power input choices available are PINION and GEAR
SPECIFY	Switch, Whether INPUTTORQUE or OUTPUTTORQUE is known.
TORQUEINPUT	Float, The torque at the input member IF(SPECIFY=INPUTTORQUE)
TORQUEOUTPUT	Float, The torque at the input member IF(SPECIFY=OUTPUTTORQUE)
RPMINPUT	Float, Input speed in RPM
MU	Float, The coefficient of Coulomb friction
MAGRUNOUTGEAR	Float, Magnitude of high point of runout error
ANGRUNOUTGEAR	Float, Orientation angle (Deg) high point of runout error
MAGRUNOUTPINION	Float, Magnitude of high point of runout error
ANGRUNOUTPINION	Float, Orientation angle (Deg) high point of runout error
BACKSIDECONTACT	Boolean, Check for back side contact
RADIALCONSTRAINT	Boolean, Check to constrain radial motion IF(BACKSIDECONTACT)
RADIALLOAD	Float, The operating center distance IF(BACKSIDECONTACT AND ¬(RADIALCONSTRAINT))

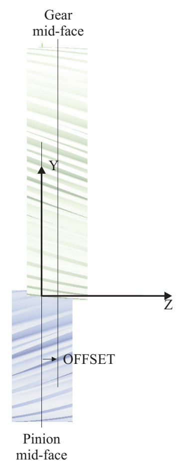


Figure 7.3: The definition of the OFFSET parameter.

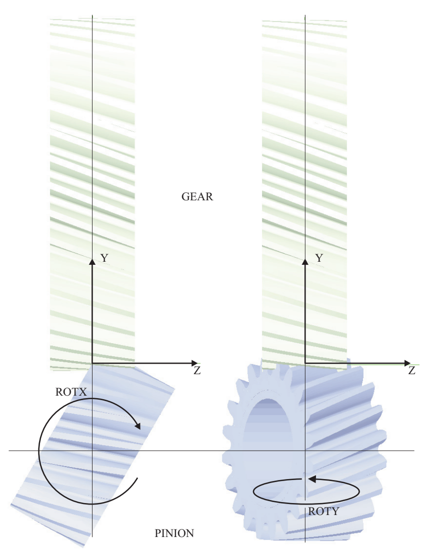


Figure 7.4: The definition of the ROTX and ROTY parameters.

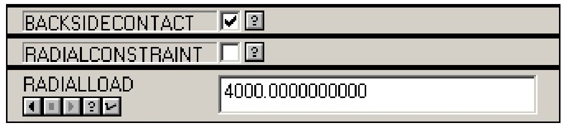


Figure 7.5: Applying a radial load to the pinion.

for the OUTPUT member where power is exiting the system.

The SPECIFY switch allows you to control which torque you want to specify. Depending on the setting for this switch, you will see either a TORQUEINPUT or TORQUEOUTPUT item where you can specify the torque value.

RPMINPUT is the speed of the input member in RPM. By default it corresponds to 1 Degree/s. The sign follows the right hand rule about the Z axis. MU is the coefficient of Coulomb friction μ. It must always be a positive value.

MAGRUNOUTGEAR defines the amplitude of the runout error of the gear. The amplitude of the runout is one half of the peak to peak runout error. ANGRUNOUTGEAR variable defines the orientation of the high point of runout error for the gear at time $t = 0$. A value of zero for this angle implies that the error is along the X axis of the fixed reference frame at time $t = 0$. For $t > 0$, the orientation changes along with the rotation of the gear. A positive value in Degrees is measured in the counter-clockwise direction.

Similarly, MAGRUNOUTPINION and ANGRUNOUTPINION are the amplitude and orientation of the pinion runout error.

If the BACKSIDECONTACT flag is set, then the back side of the teeth will be checked for contact. This flag should be used sparingly, because it will cause CPU time to more than double. If it is checked, then it is possible to release the pinion constraint in the radial direction by unchecking the RADIALCONSTRAINT box, as shown in Figure 7.5. This allows the pinion to float in the center-distance direction. A radial load must then be specified in the RADIALLOAD box to keep the pinion in contact when torque is applied. This RADIALLOAD value should be greater than the radial component of the mesh force created by the INPUTTORQUE value.

7.2 Pinion and gear menus

Figures 7.6 and 7.7 show the main menus for entering data for the pinion and gear, respectively. They are identical, except that for the gear, there is the TYPE switch which can be set to EXTERNAL or INTERNAL. The three lumped parameters are described in Table 7.2.

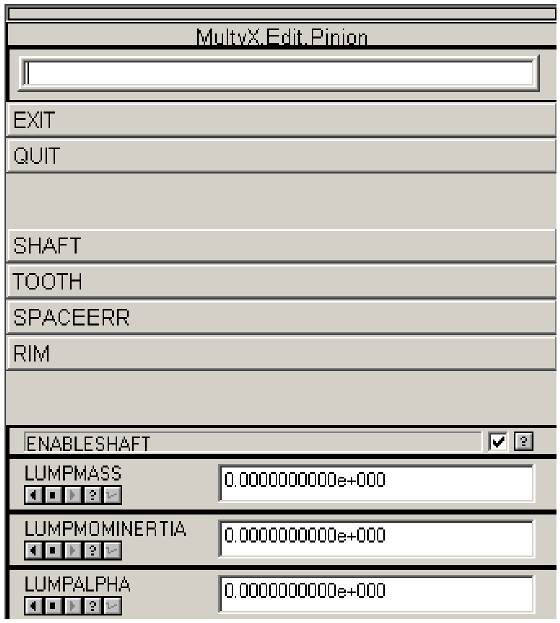


Figure 7.6: The pinion main menu.

Table 7.2: Pinion and gear lumped parameters

Item	Description
LUMPMASS	Float, Lumped mass at pinion/gear center
LUMPMOMINERTIA	Float, Lumped polar moment of inertia J_z about the pinion or gear axis of rotation
LUMPALPHA	Float, The damping constant for the lumped mass and moment of inertia

7.3 Bearings

The BEARING commands in Figures 7.6 and 7.7 lead to the bearing data menu shown in Figure 7.8. Table 7.3 describes the parameters.

If the RIGIDRACE box is checked, then the inner diameter of the pinion or external gear (or the outer diameter in the case of an internal gear) is treated as a rigid body. Otherwise, its deformation is expressed using a Fourier series expansion in the circular direction, and a

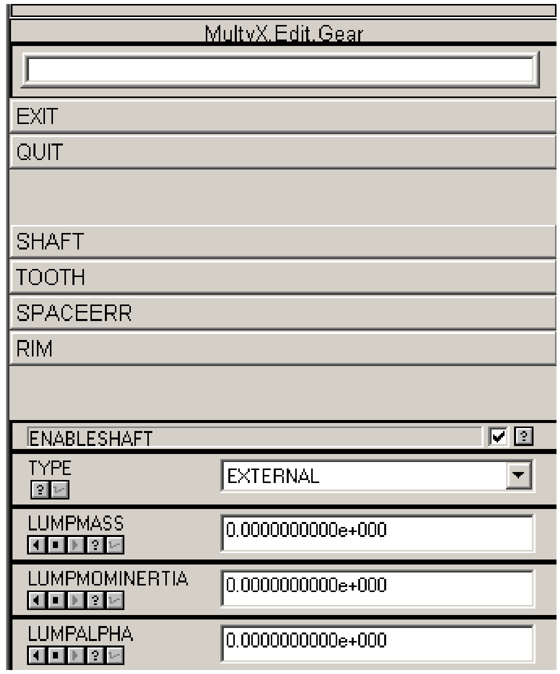


Figure 7.7: The gear main menu.

polynomial series in the axial direction. The order of this series expansion can be specified in the CIRCORDER and AXIALORDER boxes.

The RIGIDRACE box appears only when the pinion or gear model has no rim model, or if the rim model is of the SIMPLE type described later. If the rim is of the WEBBED type, the the bearing race is treated as rigid. If the rim is of the EXTSPLINED or INTSPLINED type, the splines are attached to a flexible surface on the pinion or gear, and the bearing race is a rigid cylinder attached to the spline supports.

In addition to the deformation of the race, the rigid body type motion of the inner diameter of the pinion/gear may be constrained by leaving the BEARING box unchecked. If the BEARING box is checked, then stiffness and damping matrices can be assigned to the rigid body motions. The stiffness and damping matrices of the bearing are read in from a existing input data file. The name of the file should be entered into the BRGFILE box.

The contents of a sample bearing file are shown below.

```
0 0 0 0 0 0
1.0e7 0 0 0 0 0
0 1.0e7 0 0 0 0
0 0 0 0 0 0
0 0 0 0.0e9 0 0
0 0 0 0 0.0e9 0
0 0 0 0 0 0
0 0 0 0 0 0
0 0 0 0 0 0
0 0 0 0 0 0
0 0 0 0 0 0
0 0 0 0 0 0
0 0 0 0 0 0
```

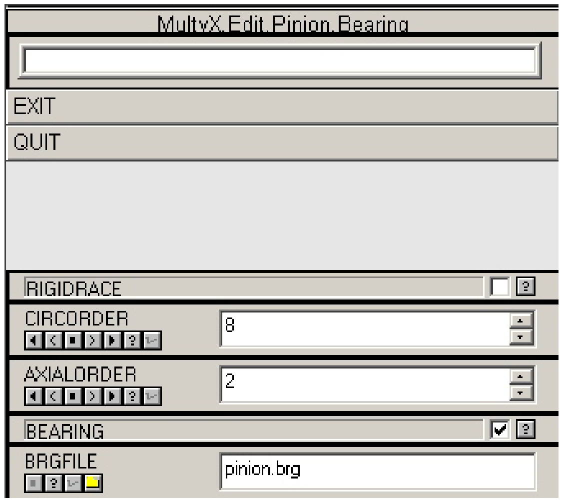


Figure 7.8: The bearing data menu.

0

Inside the bearing file, the first line in this file must always have 6 zeroes. This line is assigned for future use. The next 12 lines contain the 6x6 stiffness and 6x6 damping matrices. The last line contains a zero. This is a flag intended for future use. The 6x6 matrices correspond to the 6 degrees of freedom of bearing race 1 relative to bearing race 2, as measured in the race 2 reference frame.

The six degrees of freedom are the three translations in the X, Y and Z directions and the three rotations about the X, Y and Z directions. The X axis points from the gear center to the pinion center. The Z axis is along the axis of rotation.

Table 7.3: Bearing parameters

Item	Description	Condition
RIGIDRACE	Boolean, Whether the bearing race is a rigid cylinder	if(RIMTYPE=NONE $\vee$ RIMTYPE=SIMPLE)
CIRCORDER	Integer, Fourier series order in the circular direction	if((RIMTYPE=NONE $\vee$ RIMTYPE=SIMPLE) $\wedge$ $\neg$(RIGIDRACE))
AXIALORDER	Integer, Polynomial order in the face direction	if((RIMTYPE=NONE $\vee$ RIMTYPE=SIMPLE) $\wedge$ $\neg$(RIGIDRACE))
BEARING	Boolean, Whether the pinion has a bearing	
BRGFILE	String, Bearing file name for pinion brearing	if(BEARING)

7.4 Tooth numbering

Each individual tooth of the pinion and gear is assigned a tooth number. Figures 7.9 and 7.10 show the numbering schemes used for external and internal gears, respectively. In the figures, the Z axis is pointing out of the plane of the drawing. Additionally, the two sides of each tooth are also labeled, SIDE1 and SIDE2 as shown in Figure 7.9. Depending on the direction of rotation of the INPUT member, Side1 or Side2 of the INPUT(driving) member is in contact with the corresponding sides of the OUTPUT(driven) member. If the INPUT member is the Pinion and if the RPMINPUT is positive then Side2 of the pinion tooth makes contact with Side2 of the gear tooth.

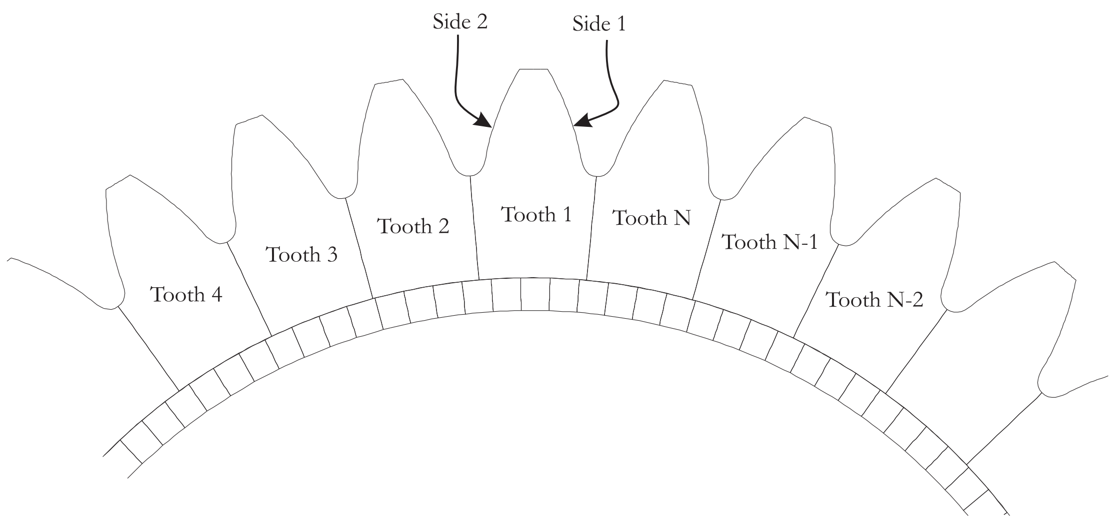


Figure 7.9: The tooth and side numbering scheme for the pinion or an external gear

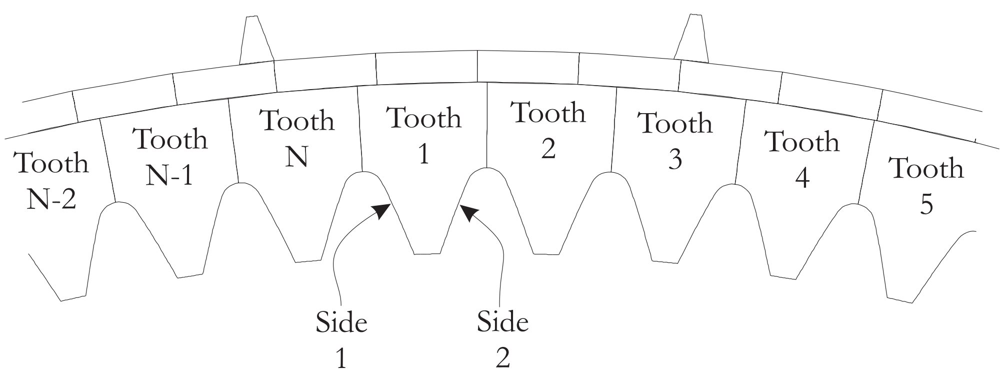


Figure 7.10: The tooth and side numbering scheme for an internal gear

7.5 Spacing errors

The SPACEERR command in Figures 7.6 and 7.7 leads to the spacing error menu shown in 7.11. The spacing error is an angular amount in radians by which the two surfaces of individual teeth on the individual pinions can be rotated from their nominal positions. A positive rotation always follows the right hand rule about the Z axis. The spacing error can be entered for any tooth in this menu.

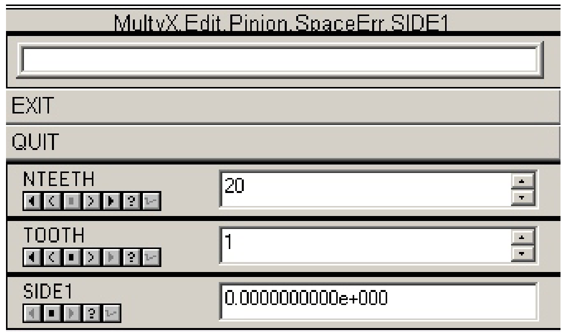


Figure 7.11: The spacing error menu.

7.6 Tooth profile

The TOOTH command in Figures 7.6 and 7.7 leads to the tooth data menu. For the pinion and for an external gear, the menu is shown in Figure 7.12. For an internal gear, the menu is shown in Figure 7.13. Table 7.4 describes the parameters common to external and internal gear menus. Table 7.5 describes parameters specific to external gears, and Table 7.6 describes the parameters specific to internal gears.

The item PROFILETYPE can be set to SIMPLE or NUMERICAL. When PROFILETYPE is NUMERICAL, a new item called PROFILEFILE shows up in the TOOTH menu. Using this file, it is possible to specify a tooth profile that is not a standard involute form. The item PROFILEFILE is the name of an ASCII file. This file contains a numerical description of the transverse cross section of the tooth profile. The first line in the file contains the number of points. Each subsequent line of the file contains X,Y,XN,YN for one point, starting at the root and proceeding to the tip. The first point lies exactly in between two adjacent teeth. The last point is at the intersection of the profile and the outer diameter. (X,Y) are the coordinates of the point on the transverse section. (XN,YN) is a 2D unit vector normal to the point in the transverse plane.

The Y axis lies along the tooth center. For an external gear, the X coordinate of all points is positive. For internal gears, it is negative. It points out from the tooth. XN is positive for an external gear, and XN is negative for an internal gear.

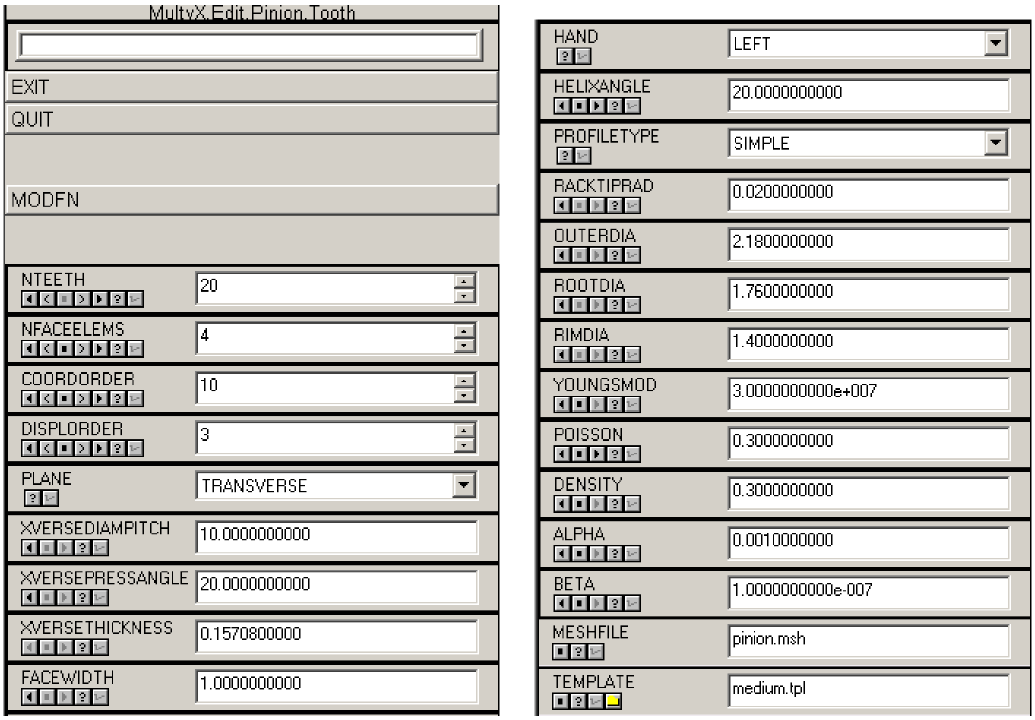


Figure 7.12: The tooth data menu for the pinion or an external gear.

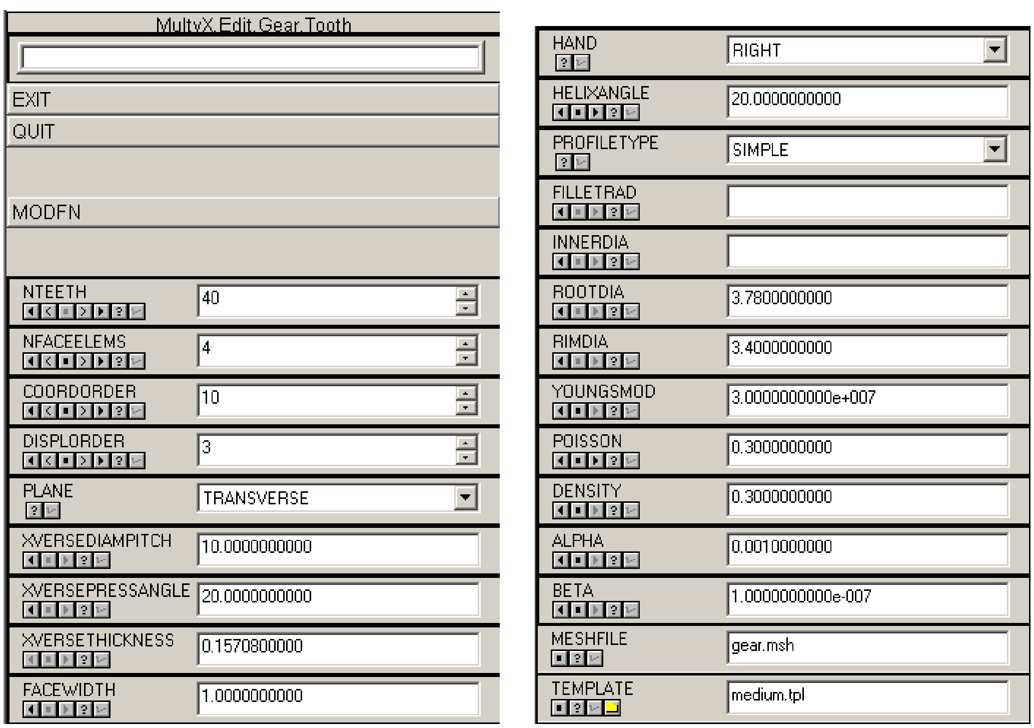


Figure 7.13: The tooth data menu for an internal gear.

Table 7.4: Parameters common to external and internal teeth

Item	Description	Condition
NTEETH	Integer, The number of teeth on gear	
NTEETHMODELED	Integer, The number of teeth modeled on the pinion	
NFACEELEMS	Integer, The number of elements across face	
COORDORDER	Integer, Limit on order of coord axodes	
DISPLORDER	Integer, Limit on displ. order of axodes.	
PLANE	Switch, The plane used to define diametral pitch, pressure angle and thickness. This switch can be set to one of the two options: NORMAL or TRANSVERSE	
NORMALDIAMPITCH	Float, The normal gen. diametral pitch	if(PLANE=NORMAL)
NORMALPRESSANGLE	Float, The normal gen. press. angle (Deg)	if(PLANE=NORMAL)
NORMALTHICK	Float, Normal thickness of the gear tooth	if(PLANE=NORMAL)
XVERSEDIAMPITCH	Float, The transverse gen. diametral pitch	if(PLANE≠NORMAL)
XVERSEPRESSANGLE	Float, The transverse gen. press. angle (Deg)	if(PLANE≠NORMAL)
XVERSETHICK	Float, Transverse thickness of the gear tooth	if(PLANE≠NORMAL)
FACEWIDTH	Float, The face width of the gear	
HAND	Switch, The hand of the gear (Left/Right)	
HELIXANGLE	Float, The helix angle of the gear	
ROOTDIA	Float, Root dia.of the gear	
RIMDIA	Float, Rim diameter of the gear	if(RIMTYPE≠NONE)
YOUNGSMOD	Float, Young's modulus for the gear material	
POISSON	Float, Poisson's ratio for the gear material	
MSHFILE	String, Mesh file name for gear	
TPLFILE	String, Template file name for gear	

Table 7.5: Parameters for defining an external tooth

Item	Description	Condition
RACKTIPRAD	Float, Rack tip radius for the gear	
OUTERDIA	Float, Outer dia. of the gear	
RIMDIA	Float, Inner dia. of the gear	if(RIMTYPE≠NONE)

Table 7.6: Parameters for defining an internal tooth

Item	Description	Condition
FILLETRAD	Float, Fillet radius for the gear	
INNERDIA	Float, Inner dia. of the gear	
RIMDIA	Float, Outer dia. of the gear	if(RIMTYPE≠NONE)

7.7 Surface modifications

The MODFN command in the tooth data menus (Figures 7.12 and 7.13) lead to the menu shown in Figure 7.14. This menu is used to specify surface modifications. Six simple surface modifications can be specified directly in this menu. Four of them are profile modifications: the linear and quadratic tip relief, and linear and quadratic root relief. There are two types of lead modification: the standard crown lead modification, and a 'flat' crown lead modification.

Linear tip relief is applied using the parameters shown in Table 7.7 and Figure 7.15. The modification magnitude is a linear function of the involute roll angle. Linear root relief is applied using the parameters shown in Table 7.8 and Figure 7.16. The modification magnitude is a linear function of the involute roll angle.

Quadratic tip relief is applied using the parameters shown in Table 7.9 and Figure 7.17. The modification magnitude is a quadratic function of the involute roll angle. Quadratic root relief is applied using the parameters shown in Table 7.10 and Figure 7.18. The modification magnitude is a quadratic function of the involute roll angle.

Crown modification is applied using the parameters shown in Table 7.11 and Figure 7.19. The modification magnitude is a quadratic function of the surface coordinate ζ. ζ varies from -1 to $+1$ along the face width of the tooth.

'Flat' crown modification is applied using the parameters shown in Table 7.12 and Figure 7.20. This modification has an unmodified flat region in the moddle. The crowning magnitude at the two ends of the tooth face can be independently controlled.

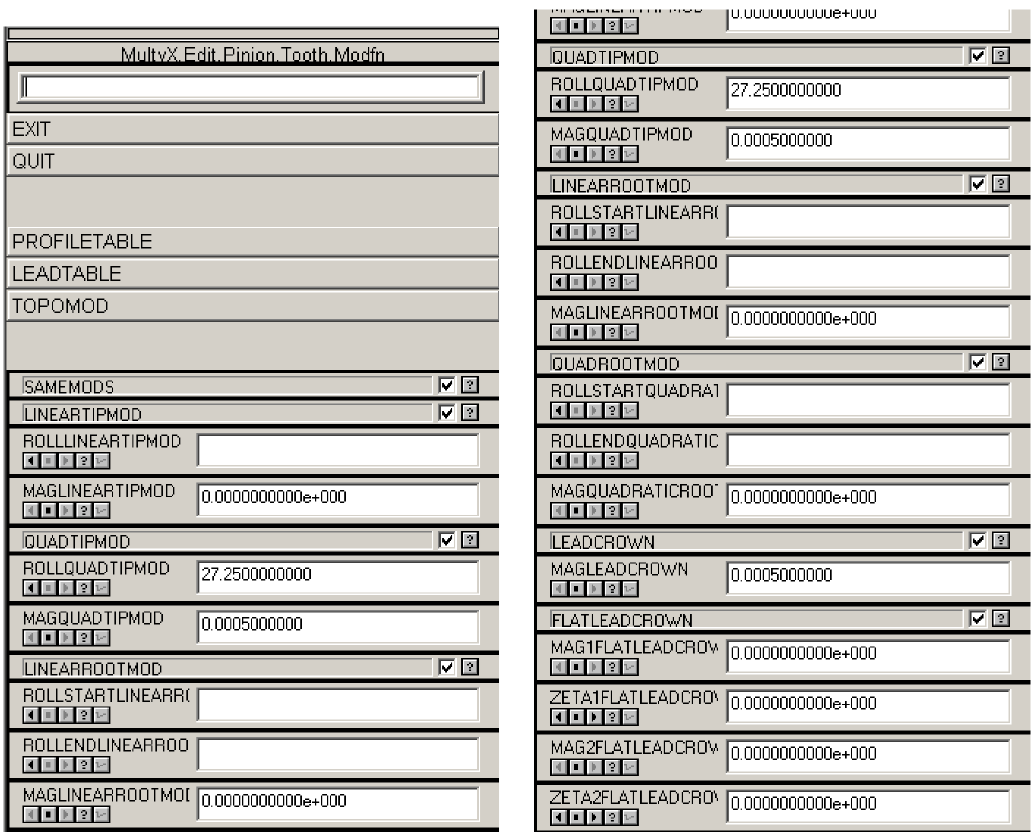


Figure 7.14: The tooth modification menu.

Table 7.7: Linear tip modification parameters

Item	Description
LINEARTIPMOD	Boolean, Whether to apply the linear tip modfn.
ROLLLINEARTIPMOD	Float, Roll angle at start of the linear tip modfn.
MAGLINEARTIPMOD	Float, Magnitude of the linear tip modfn.

Table 7.8: Linear root modification parameters

Item	Description
LINEARROOTMOD	Boolean, Whether to apply the linear flank modfn.
ROLLSTARTLINEARROOTMOD	Float, Roll angle at start of the linear flank modfn.
ROLLENDLINEARROOTMOD	Float, Roll angle at the end of the linear flank modfn.
MAGLINEARROOTMOD	Float, Magnitude of the linear flank modfn.

The PROFILETABLE command in the tooth modification menu of Figure 7.14 leads to the menu shown in Figure 7.21. This menu is used to specify an arbitrary profile modification in tabular form. Table 7.13 and Figure 7.22 show the meaning of the parameters in this menu. The modification is specified at an arbitrary number of roll angle values. In between these roll angle values, the profile modification is linearly interpolated.

Similarly, an arbitrary lead modification can be specified in tabular form. The LEADTABLE command in the tooth modification menu of Figure 7.14 leads to the menu shown in Figure 7.23. Table 7.14 and Figure 7.24 show the meaning of the parameters in this menu. The modification is specified at an arbitrary number of ζ values. The surface coordinate ζ varies from -1 to $+1$ along the face width of the tooth. In between these ζ values, the lead modification is linearly interpolated.

Finally, a surface modification that is an arbitrary function of both the roll angle and ζ can be specified using the topographic modification option. The TOPOMOD command in the tooth modification menu of Figure 7.14 leads to the menu shown in Figure 7.25. Table 7.15 shows the meaning of the parameters in this menu. The modification is specified at an arbitrary number of ζ and roll angle values. The magnitude is specified for each pair of these values. Bilinear interpolation is used between them.

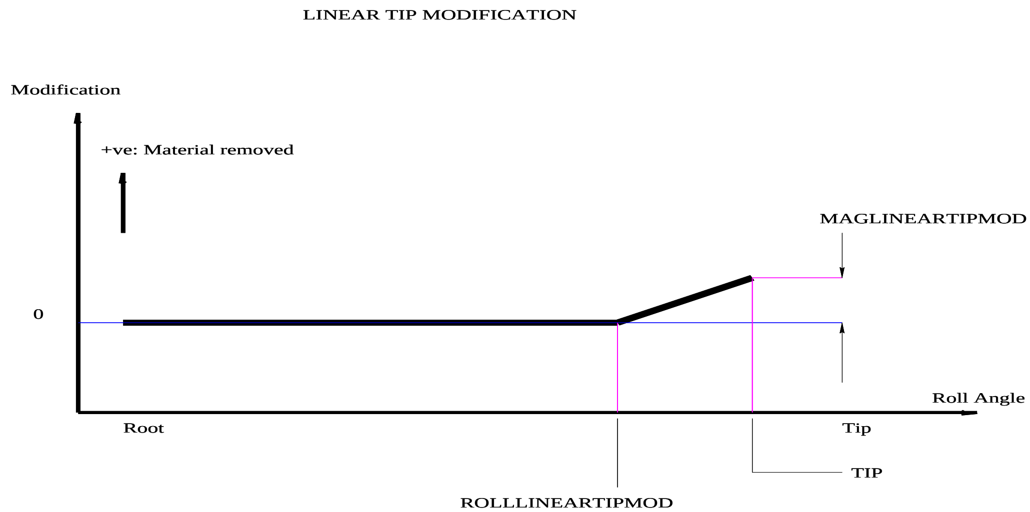


Figure 7.15: Linear tip modification

Table 7.9: Quadratic tip modification parameters

Item	Description
QUADTIPMOD	Boolean, Whether to apply the quadratic tip modfn.
ROLLQUADTIPMOD	Float, Roll angle at start of the quadratic tip modfn.
MAGQUADTIPMOD	Float, Magnitude of the quadratic tip modfn.

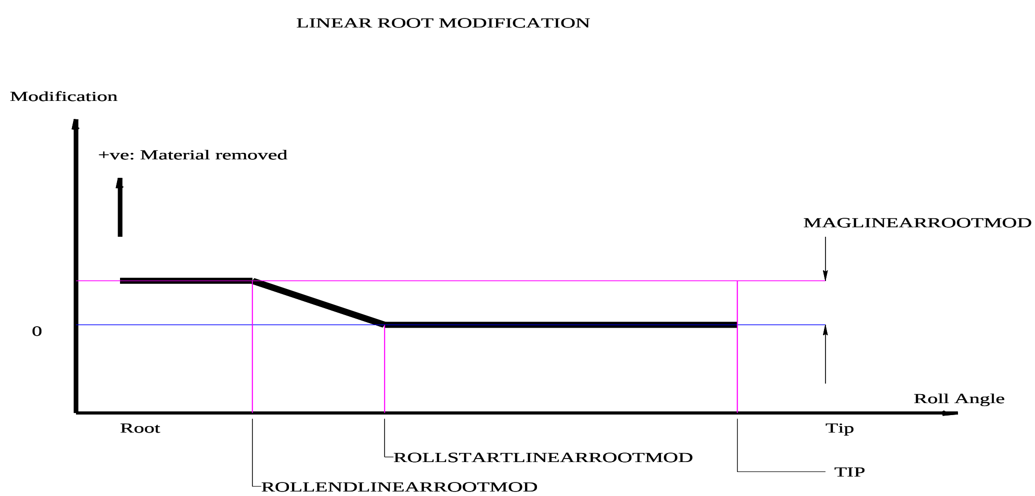


Figure 7.16: Linear root modification

Table 7.10: Quadratic root modification parameters

Item	Description
QUADROOTMOD	Boolean, Whether to apply the quadratic flank modfn.
ROLLSTARTQUADROOTMOD	Float, Roll angle at start of the quadratic flank modfn.
ROLLENDQUADROOTMOD	Float, Roll angle at the end of the quadratic flank modfn.
MAGQUADROOTMOD	Float, Magnitude of the quadratic root modfn.

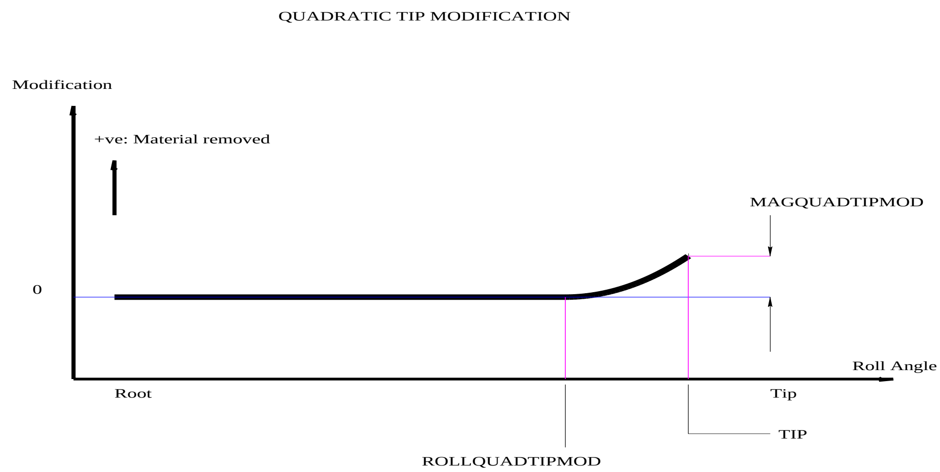


Figure 7.17: Quadratic tip modification

Table 7.11: Crown modification parameters

Item	Description
LEADCROWN	Boolean, Whether to apply the lead crowning.
MAGLEADCROWN	Float, Magnitude of the lead crown.

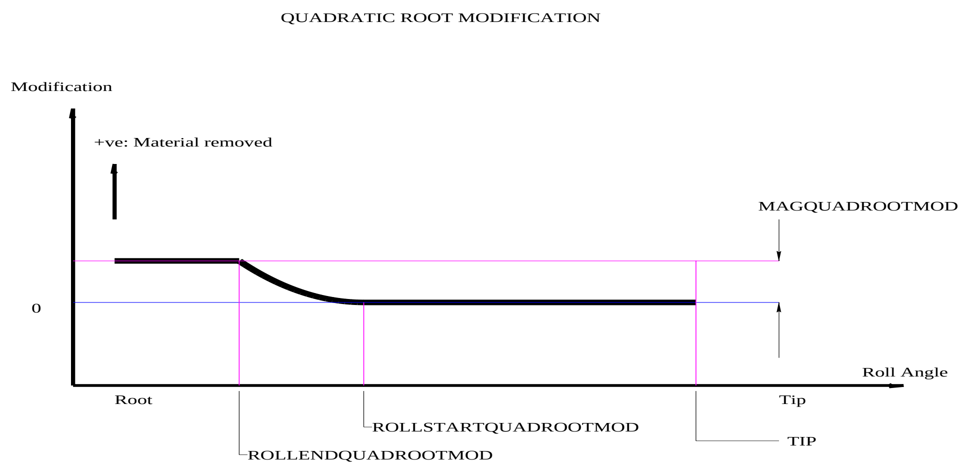


Figure 7.18: Quadratic root modification

Table 7.12: Flat Lead Crown modification parameters

Item	Description
FLATLEADCROWN	Boolean, Whether to apply the lead crowning.
MAG1FLATLEADCROWN	Float, Magnitude of the lead crown at the $\zeta = -1.0$ end.
ZETA1FLATLEADCROWN	Float, Start ζ at the $\zeta = -1.0$ end.
MAG2FLATLEADCROWN	Float, Magnitude of the lead crown at the $\zeta = +1.0$ end.
ZETA2FLATLEADCROWN	Float, Start ζ at the $\zeta = +1.0$ end.

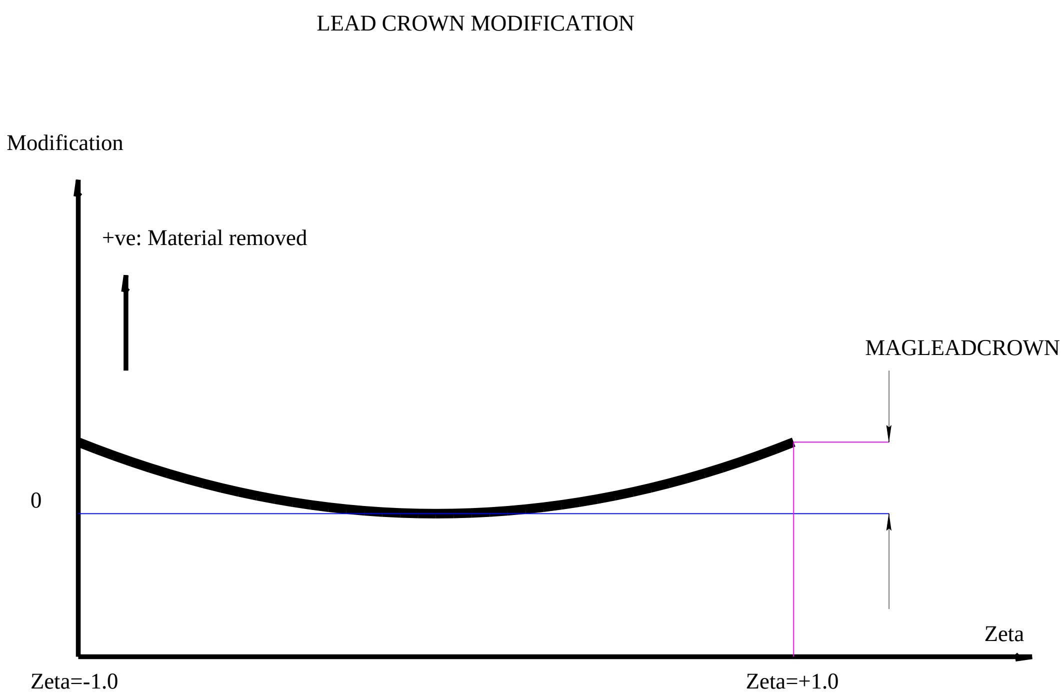


Figure 7.19: Crown modification

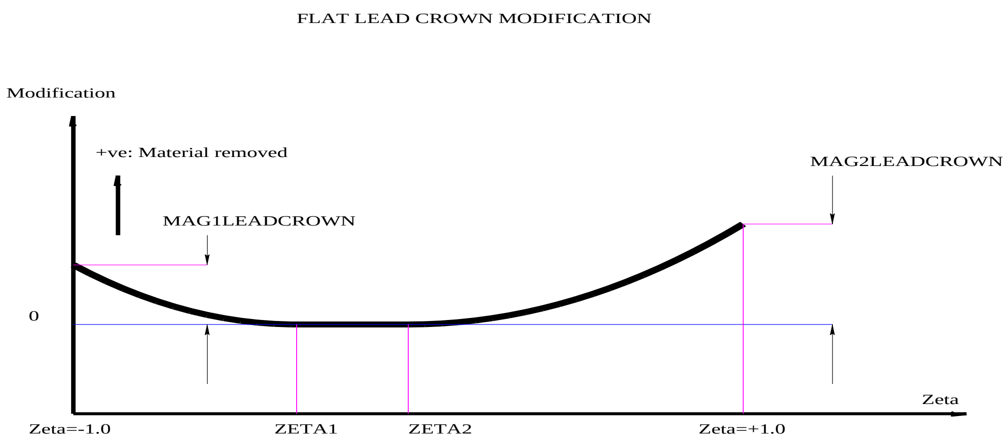


Figure 7.20: Flat Crown modification

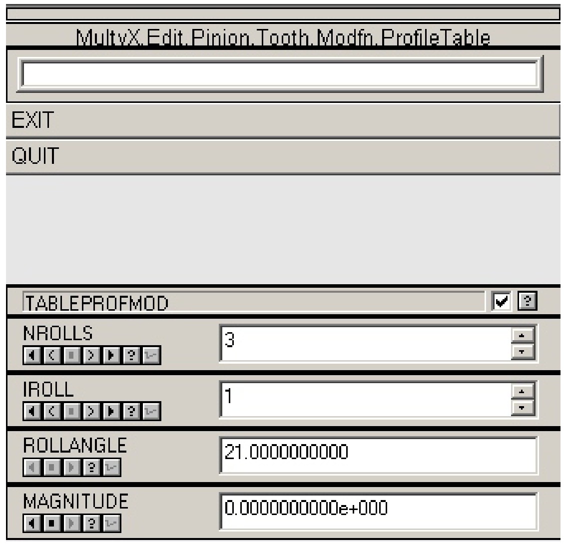


Figure 7.21: The tabular profile modifications menu.

Table 7.13: Tabular profile modification parameters

Item	Description
TABLEPROFMOD	Boolean, Whether to use a profile modfn table.
NROLLS	Integer, The number roll angles used in the profile modfn. table
IROLL	Integer, The roll angle number in the profile modfn. table
ROLLANGLE	Float, Vector, indexed by IROLL, Roll angle in the profile modfn. table
MAGNITUDE	Float, Vector, indexed by IROLL, Magnitude of modification in the profile modfn. table

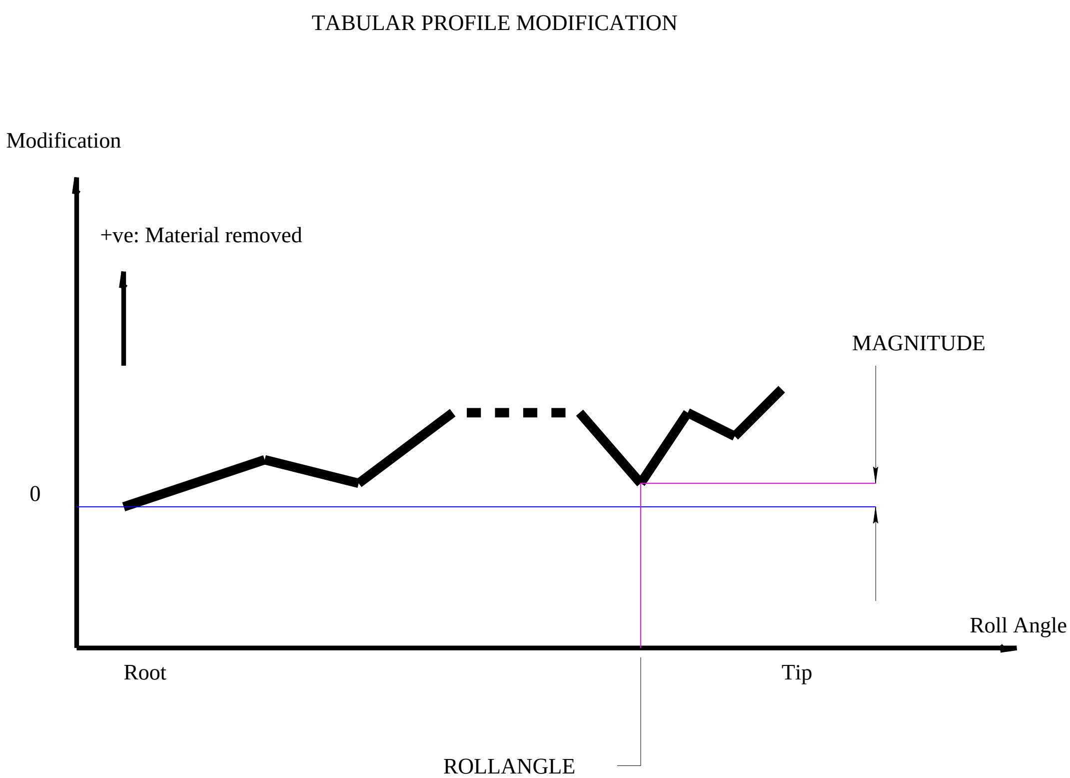


Figure 7.22: Tabular tip modification

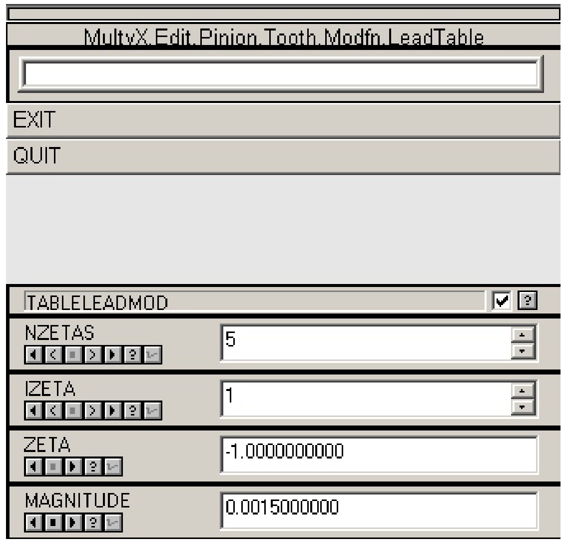


Figure 7.23: The tabular lead modifications menu.

Table 7.14: Tabular lead modification parameters

Item	Description
TABLELEADMOD	Boolean, Whether to use a lead modfn table.
NZETAS	Integer, The number zetas used in the lead modfn. table
IZETA	Integer, The zeta number in the lead modfn. table
ZETA	Float, Vector indexed by IZETA, Zeta value in the lead modfn. table
MAGNITUDE	Float, Vector indexed by IZETA, Magnitude of modification in the lead modfn. table

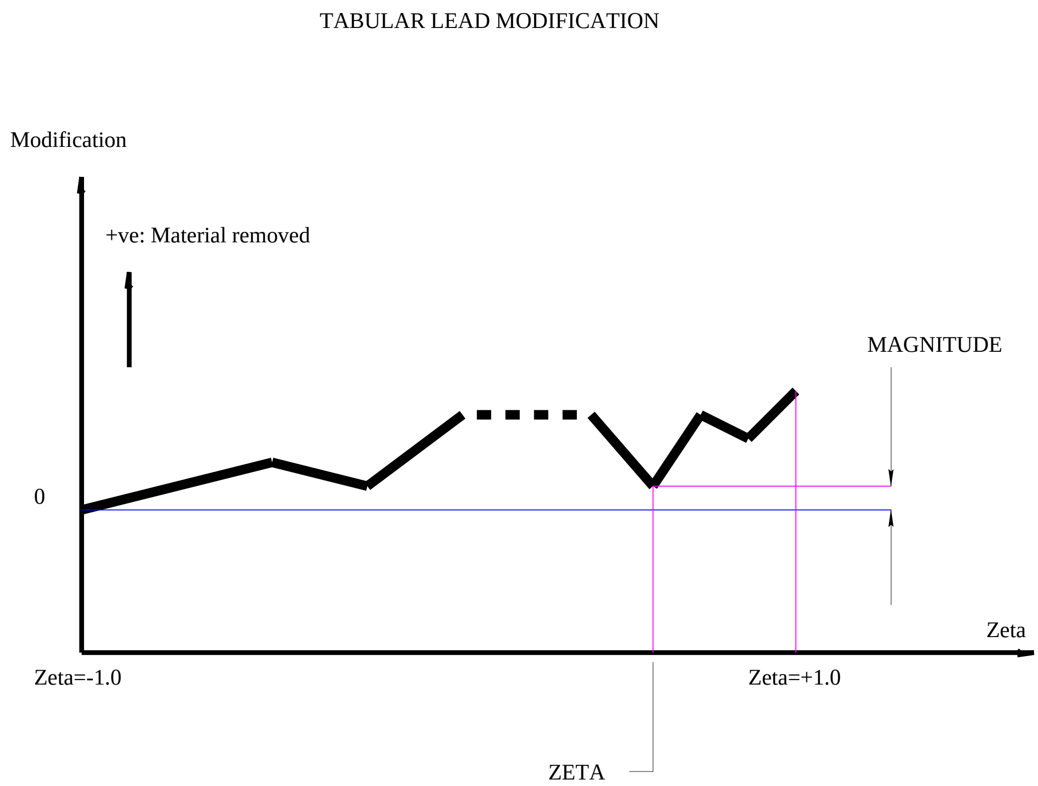


Figure 7.24: Tabular lead modification

Table 7.15: Topographical surface modification parameters

Item	Description
TOPOMOD	Boolean, Whether to use topographic modfns.
NZETAS	Integer, The number of zetas used in the topographic modfn. table
IZETA	Integer, The zeta number in the topo modfn. table
ZETA	Float, Vector indexed by Zeta value in the topo. modfn. table
NROLLS	Integer, The number of roll angles used in the topo. modfn. table
IROLL	Integer, The roll angle number in the topo. modfn. table
ROLL	Float, Vector indexed by IROLL, Roll angle in the topo. modfn. table
MAGNITUDE	Float, Array indexed by IZETA and IROLL, Magnitude of modification in the lead modfn.

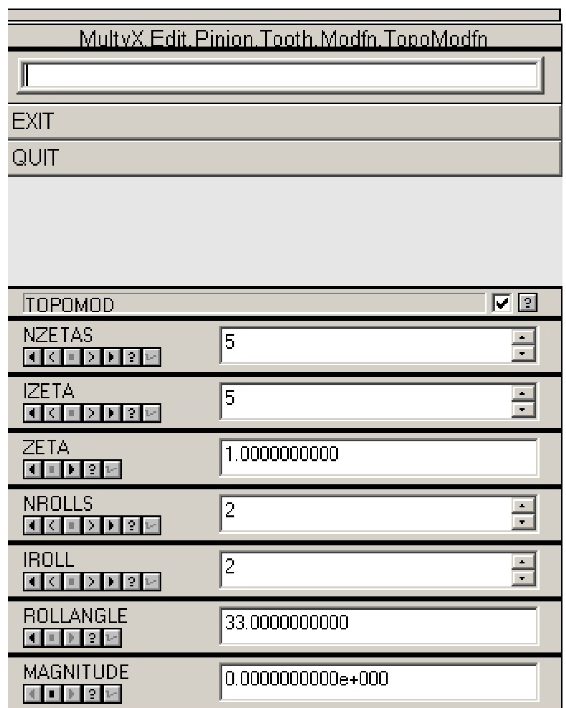


Figure 7.25: The topographical modifications menu.

7.8 Modeling the rim

The purpose of including a rim model is to apply the correct boundary condition to the tooth model. This is important because the boundary conditions can have a significant effect on the load and stress distribution.

The RIM command in Figures 7.6 and 7.7 leads to the rim data menu shown in Figure 7.26. The default rim type is NONE. For this default case, no data is necessary.

The other rim options are SIMPLE, WEBBED, EXTERNALSPLINED and INTERNALSPLINED

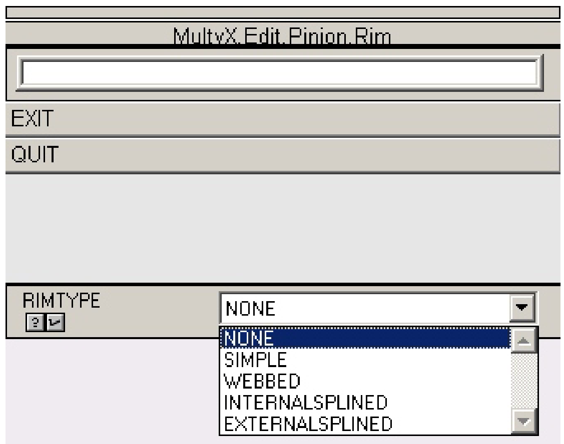


Figure 7.26: The menu for specifying rim data.

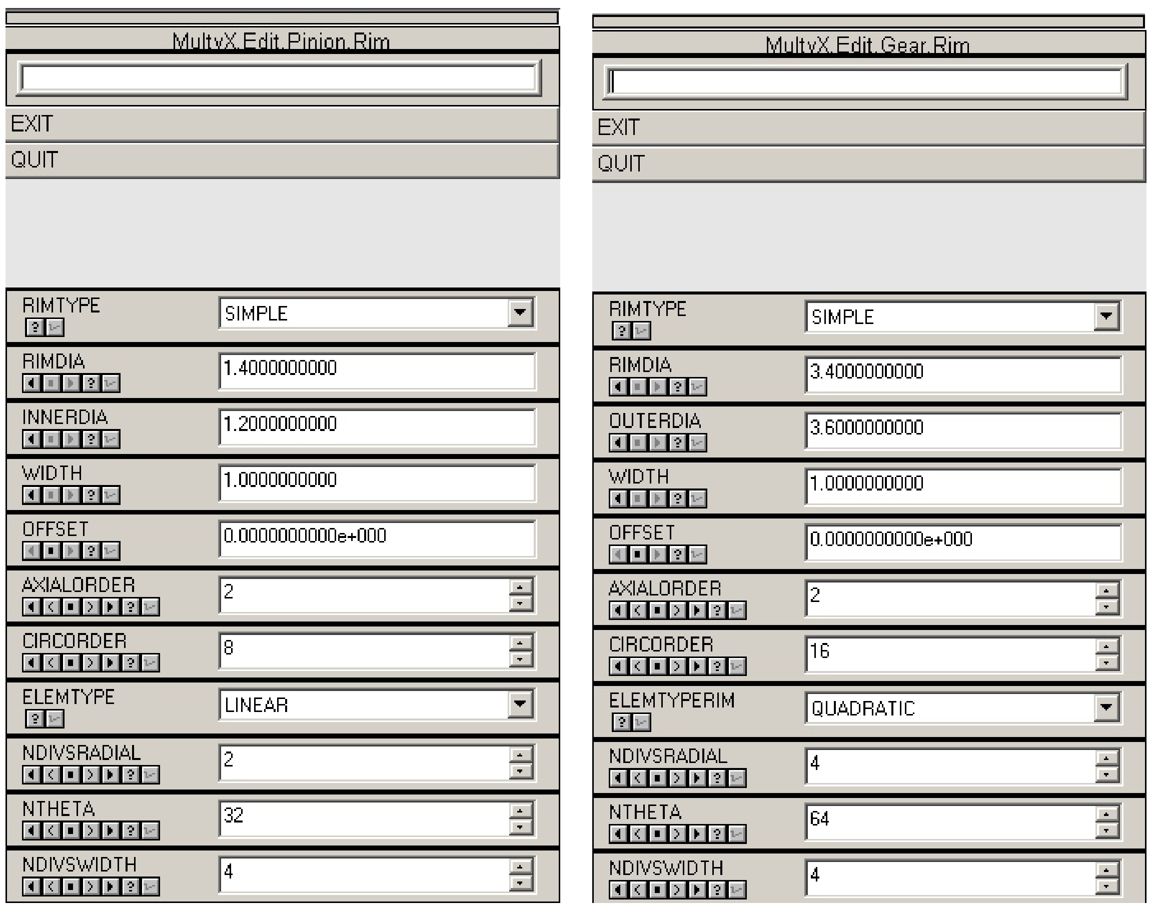
Figure 7.27 shows the menus as they appear for the SIMPLE rim, for external and internal gears, respectively. Table 7.16 and Figures 7.28 and 7.29 explain the meaning of the input parameters.

Figure 7.30 shows the menus as they appear for the WEBBED rim, for external and internal gears, respectively. Table 7.17 explains the meaning of the input parameters. The WEBBED rim is composed of an arbitrary number of 'segments'. The position of the first segment cannot be changed. Each of the remaining segments can be placed in four different positions relative to the previous segment, as shown in Figures 7.31 through 7.38

Figure 7.39 shows an example of an external gear with a webbed rim. Refer to the session file- `externalwebbed.ses` in the `WORKING` directory for rim details. Figure 7.40 shows an internal gear with a webbed rim. Refer to the session file- `internalwebbed.ses` in the `WORKING` directory for rim details.

Figures 7.41 and 7.42 show the menus as they appear for the EXTSPLINED and INTSPLINED rims, for external and internal gears, respectively. Table 7.18 explains the meaning of the input parameters. The EXTSPLINED option refers to external splines, and the INTSPLINED option refers to internal splines. Either of these can be used with external and internal gears. These four combinations, and the parameters required are depicted in Table 7.18 and Figures 7.43 through 7.46.

Examples of external and internal gears with external and internal splines are shown in Figures 7.47 through 7.50.



(a) Pinion or external gear (b) Internal gear

Figure 7.27: The menu for a simple rim

Table 7.16: Simple rim parameters

Item	Description	Condition
RIMDIA	Float, Rim diameter of the gear	
INNERDIA	Float, Inner dia. of the gear	if(GEARTYPE=EXTERNAL)
OUTERDIA	Float, Outer dia. of the gear	if(GEARTYPE=INTERNAL)
WIDTH	Float, Width in the axial direction of rim.	
OFFSET	Float, Offset in the axial direction of rim.	
AXIALORDER	Integer, Polynomial order in the face direction	
CIRCORDER	Integer, Fourier series order in the circular direction	
ELEMTYPE	Switch, Type of finite element. Available options are LINEAR, QUADRATIC and CUBIC	
NDIVSRADIAL	Integer, Number of elements in the radial direction	
NTHETA	Integer, Number of elements in the circular direction	
NDIVSWIDTH	Integer, Number of elements in the axial direction	

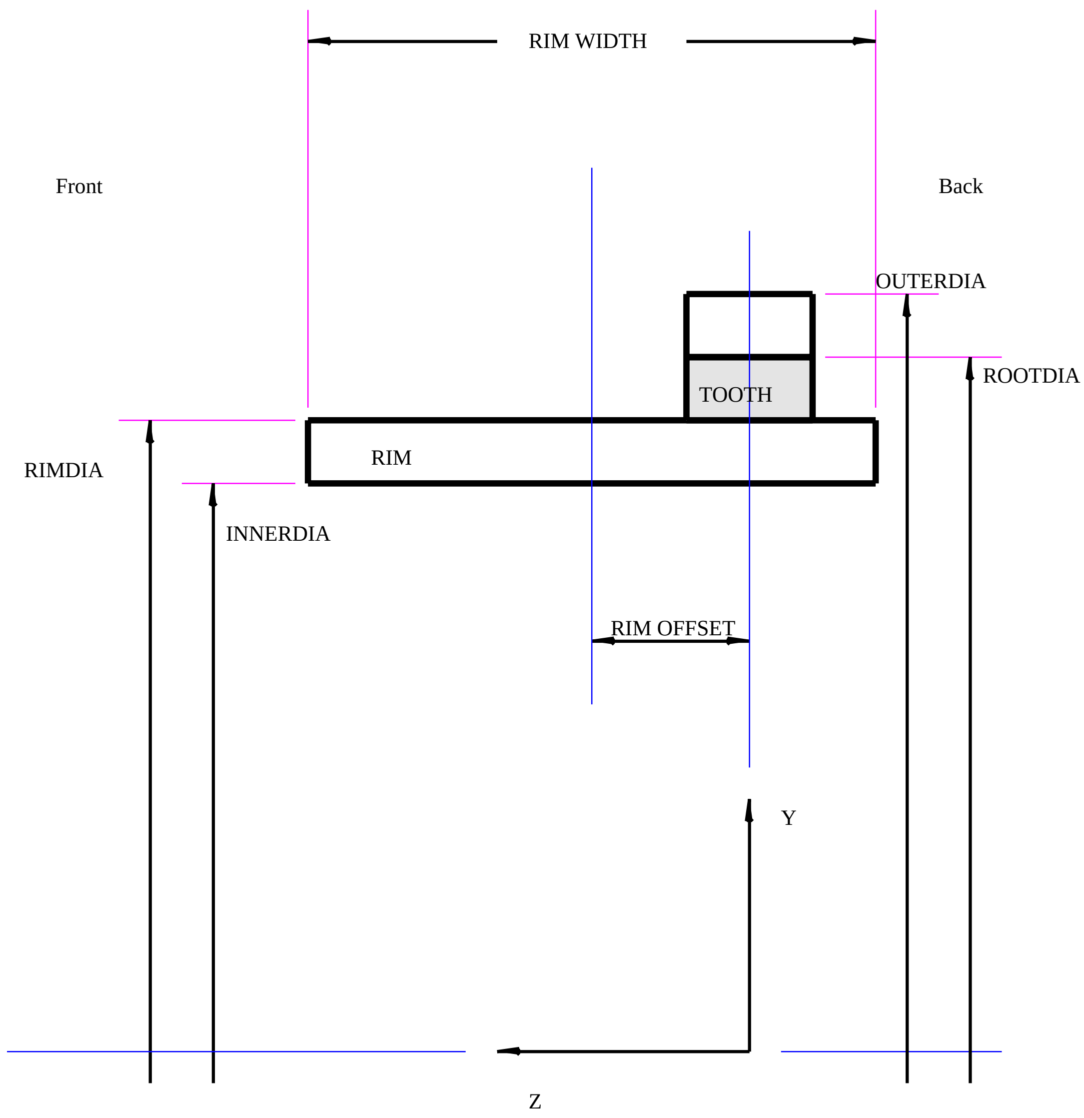


Figure 7.28: An external pinion or gear with a simple rim

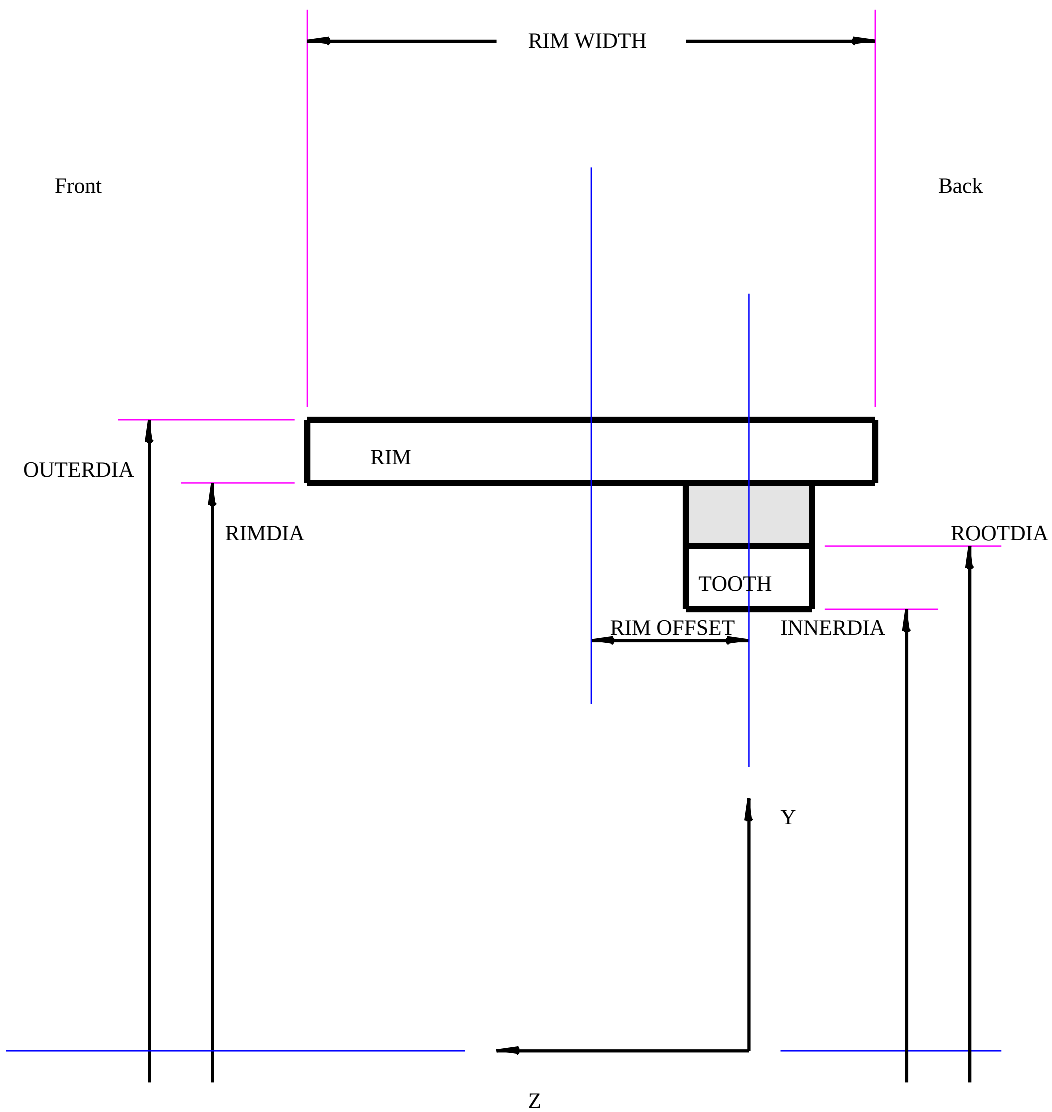
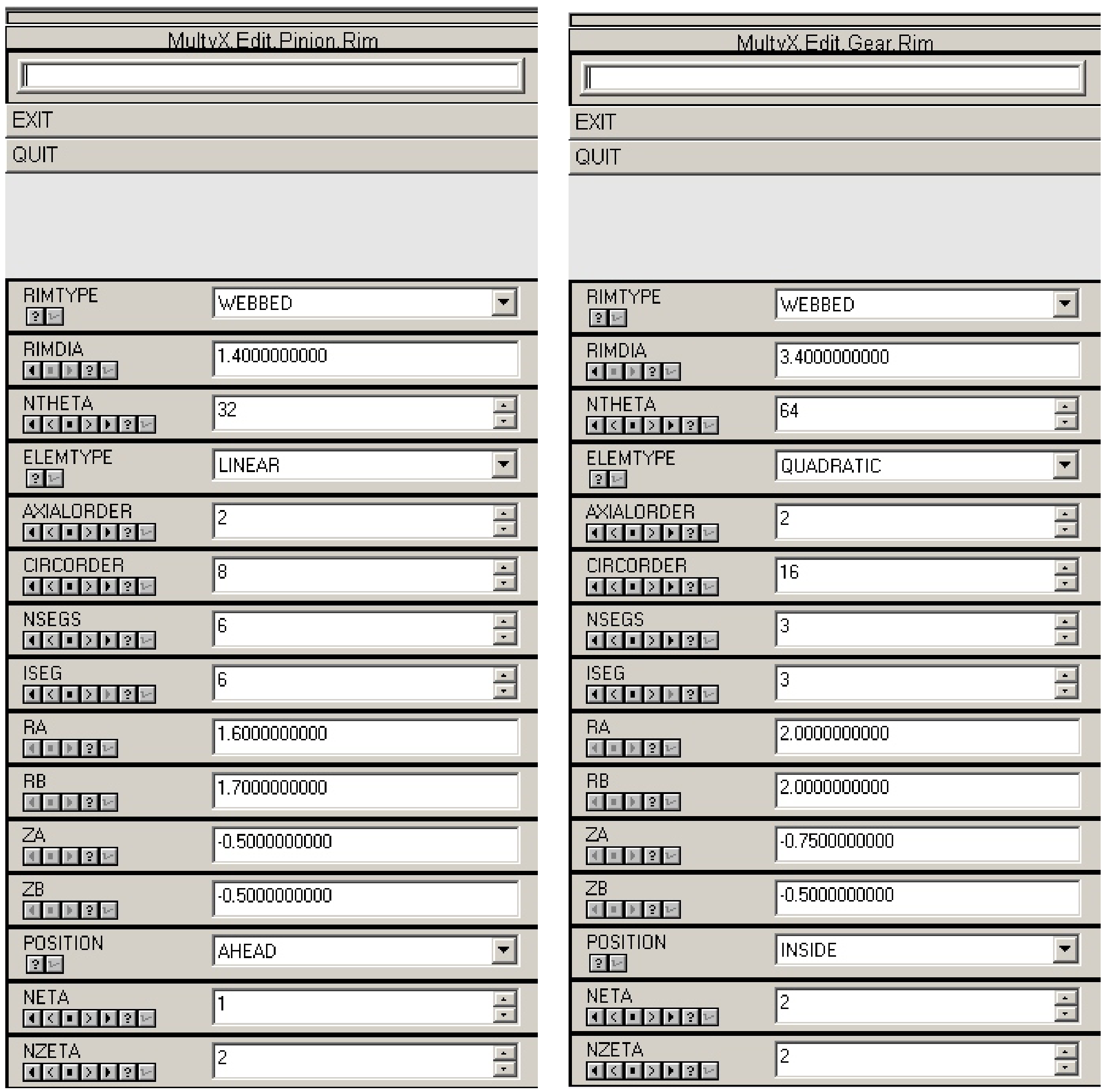


Figure 7.29: An internal gear with a simple rim



(a) Pinion or external gear

(b) Internal gear

Figure 7.30: The menu for a webbed rim

Table 7.17: Webbed rim parameters

Item	Description
RIMDIA	Float, Rim diameter of the gear
NTHETA	Integer, Number of elements in the circular direction
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.
POSITION	Switch, Vector indexed by ISEG, Relative position of the rim segment.
NETA	Integer, Vector indexed by ISEG, Number of elements in the η direction direction
NZETA	Integer, Vector indexed by ISEG, Number of elements in the ζ direction direction

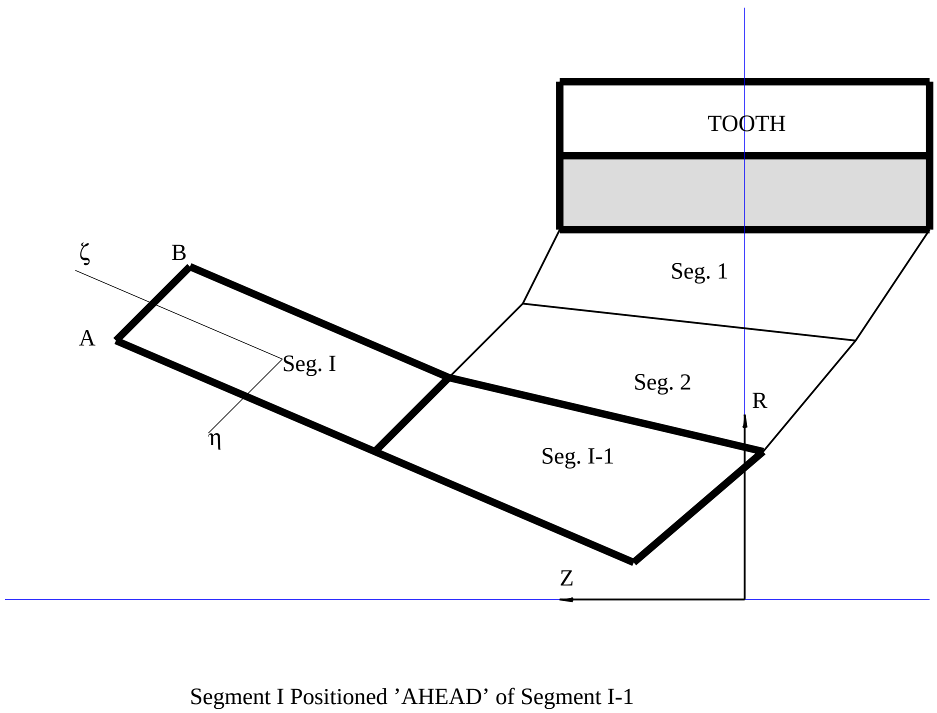


Figure 7.31: An external pinion or gear with a webbed rim, showing segment i located ahead of segment $i-1$

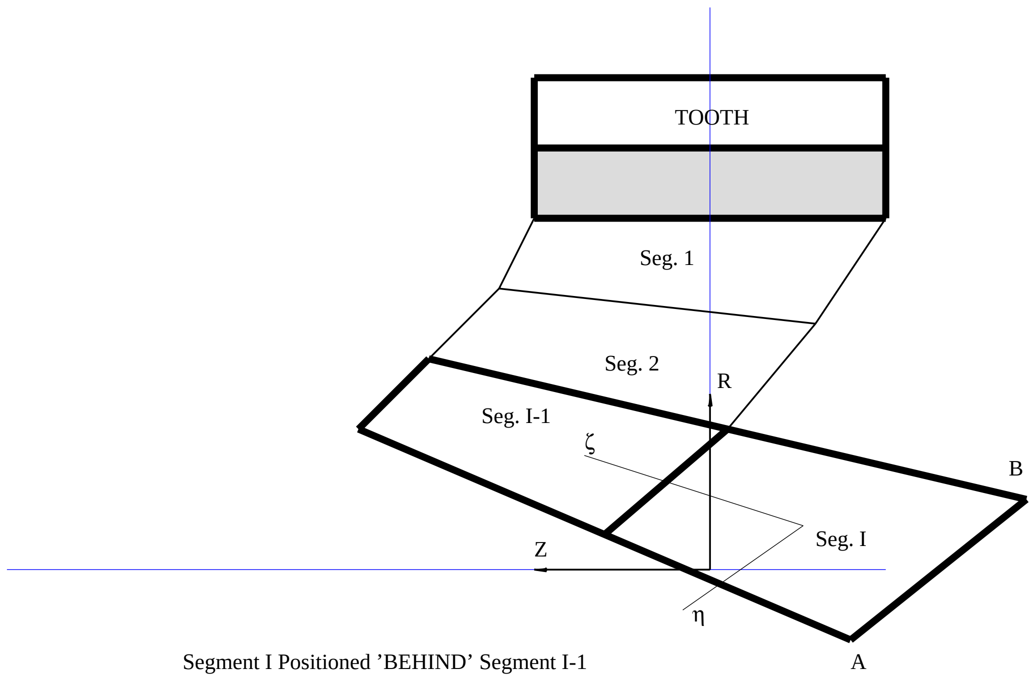


Figure 7.32: An external pinion or gear with a webbed rim, showing segment i located behind segment $i-1$

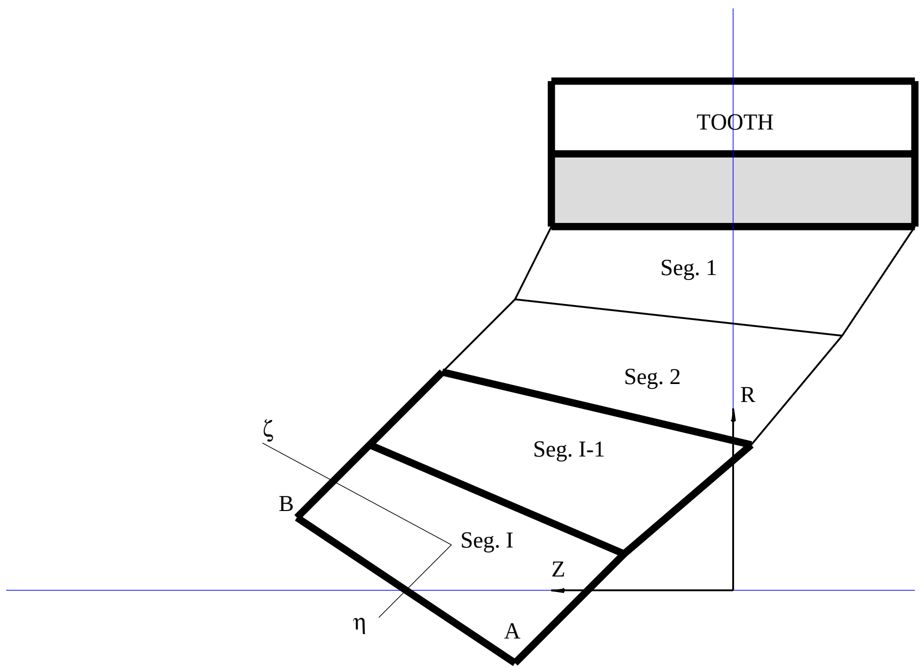


Figure 7.33: An external pinion or gear with a webbed rim, showing segment i located inside segment $i-1$

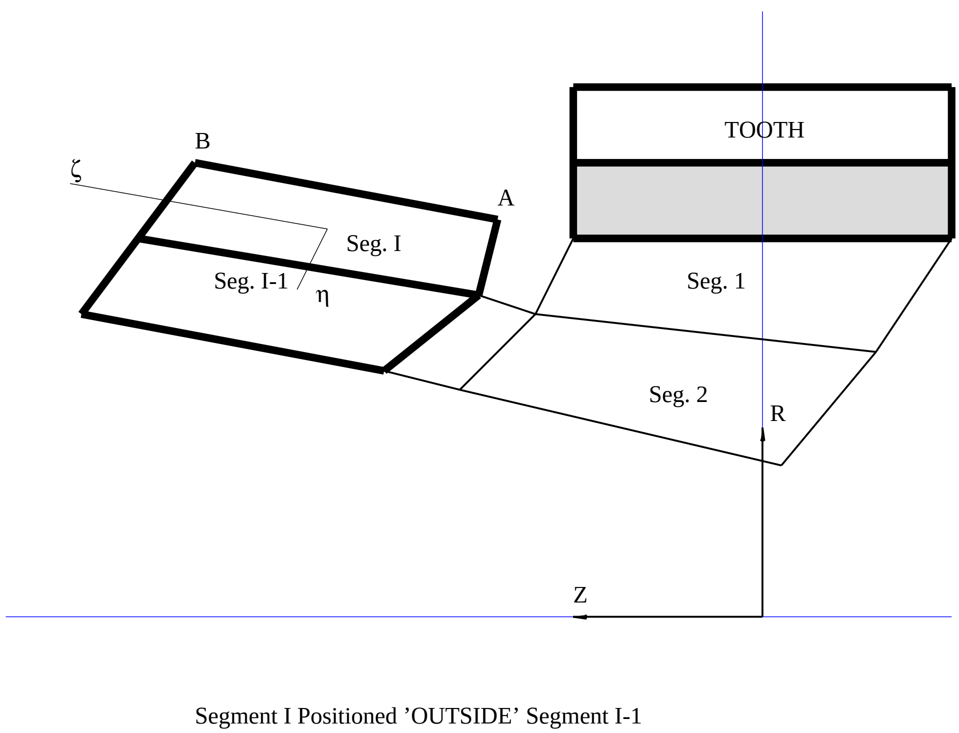


Figure 7.34: An external pinion or gear with a webbed rim, showing segment i located outside segment $i-1$

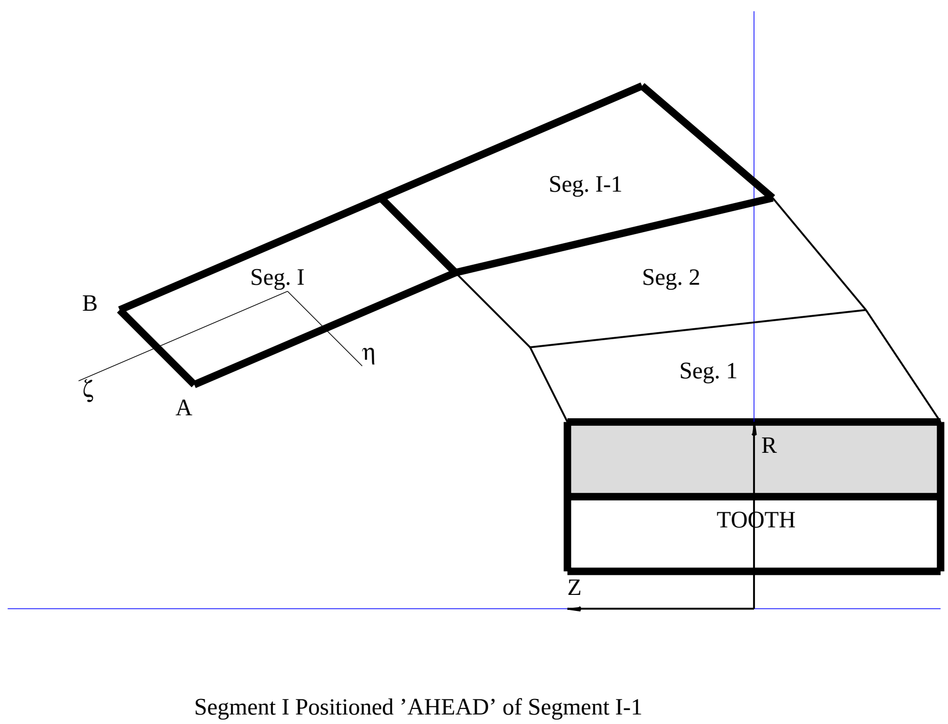


Figure 7.35: An internal gear with a webbed rim, showing segment i located ahead of segment $i-1$

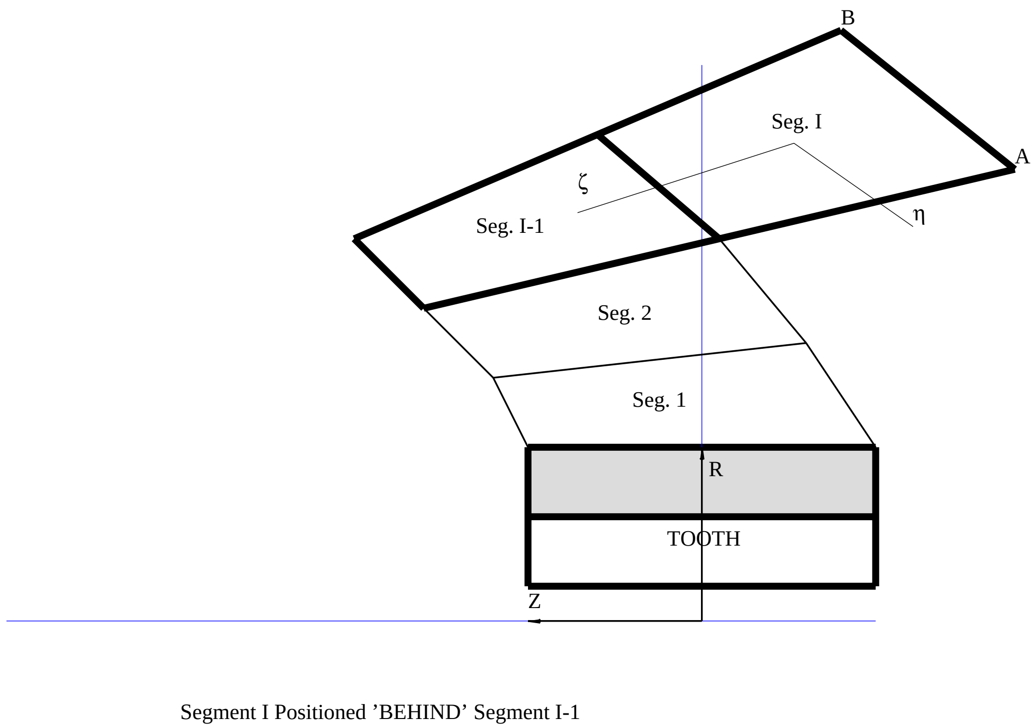


Figure 7.36: An internal gear with a webbed rim, showing segment i located behind segment $i-1$

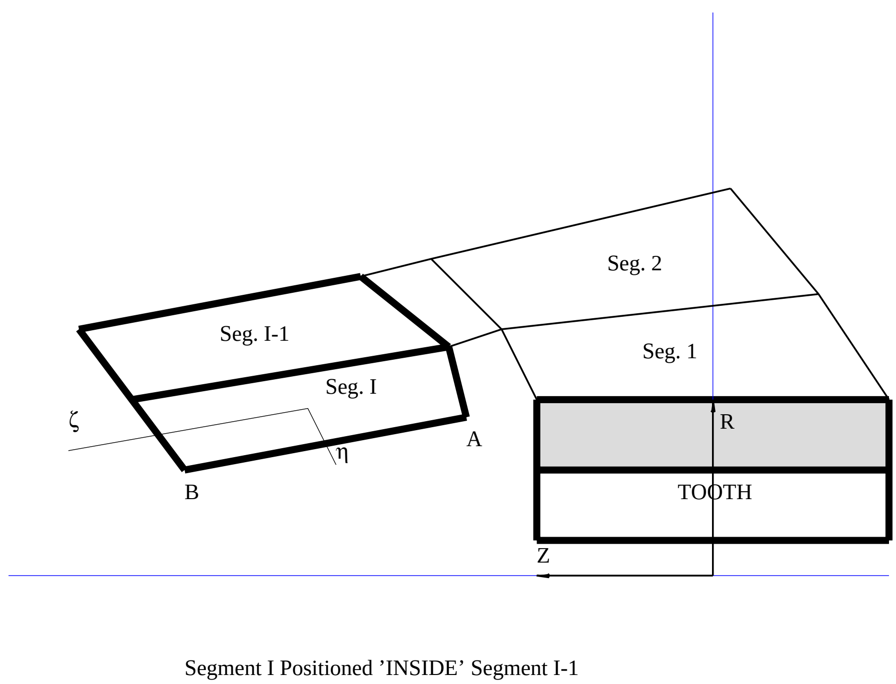


Figure 7.37: An internal gear with a webbed rim, showing segment i located inside segment $i-1$

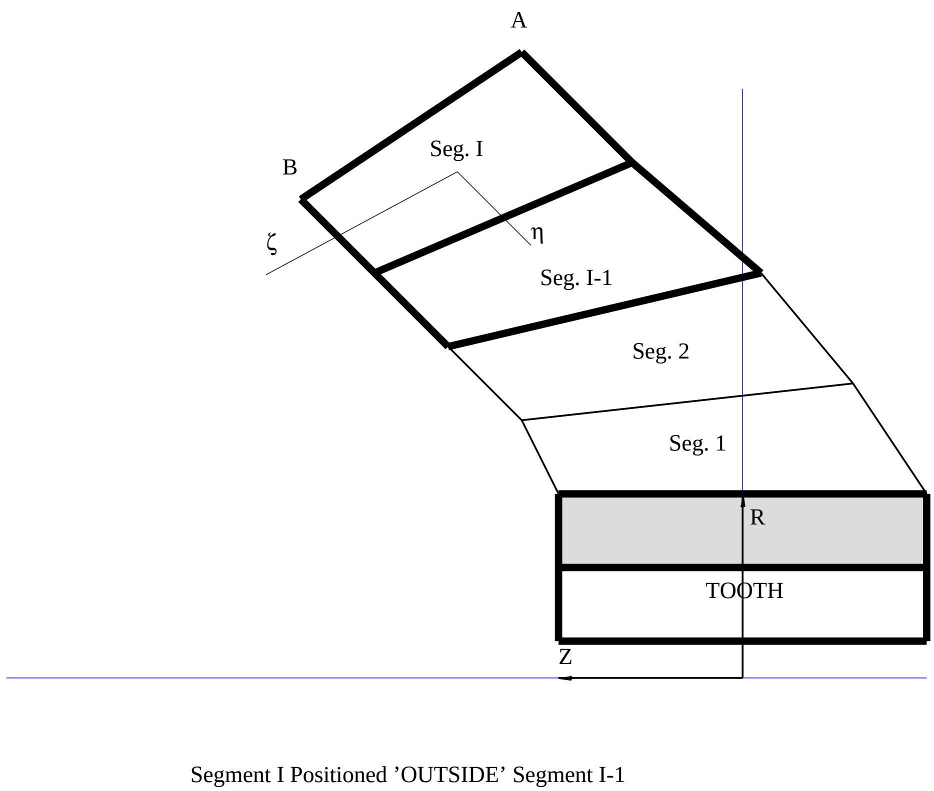


Figure 7.38: An internal gear with a webbed rim, showing segment i located outside segment $i-1$

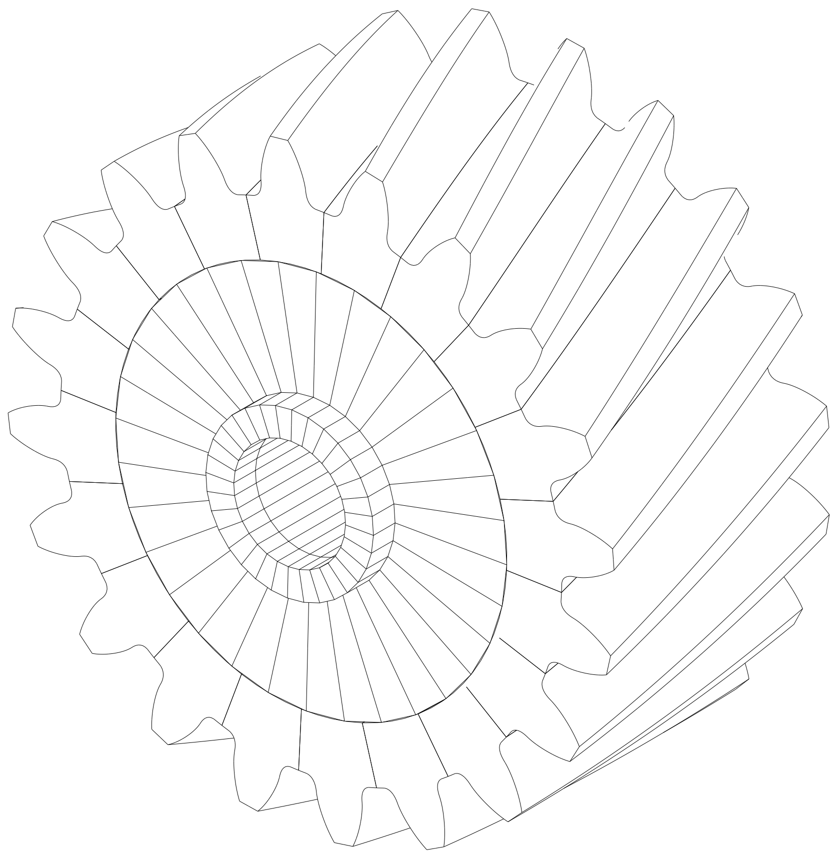


Figure 7.39: An example of a pinion or an external gear with a webbed rim.

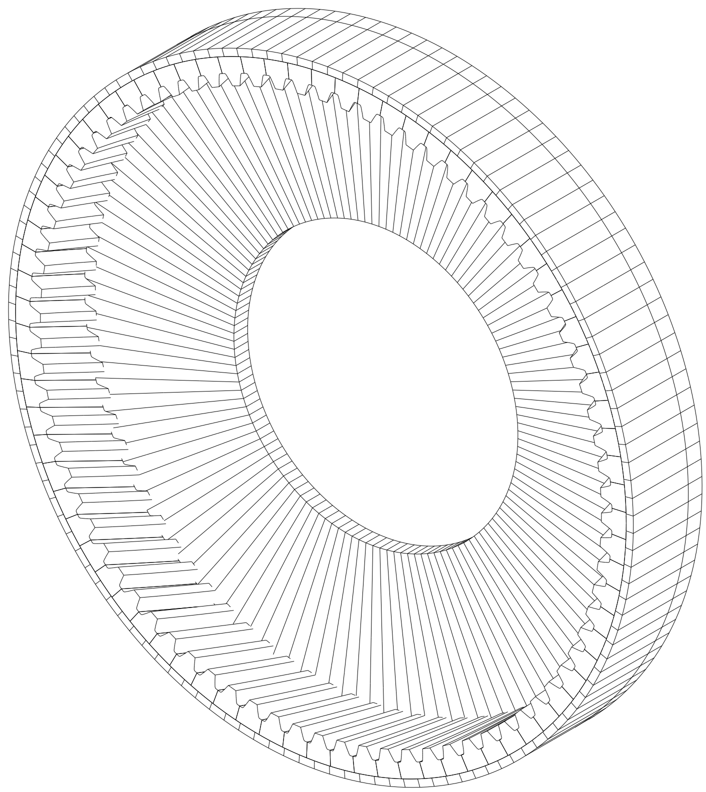


Figure 7.40: An example of an internal gear with a webbed rim.

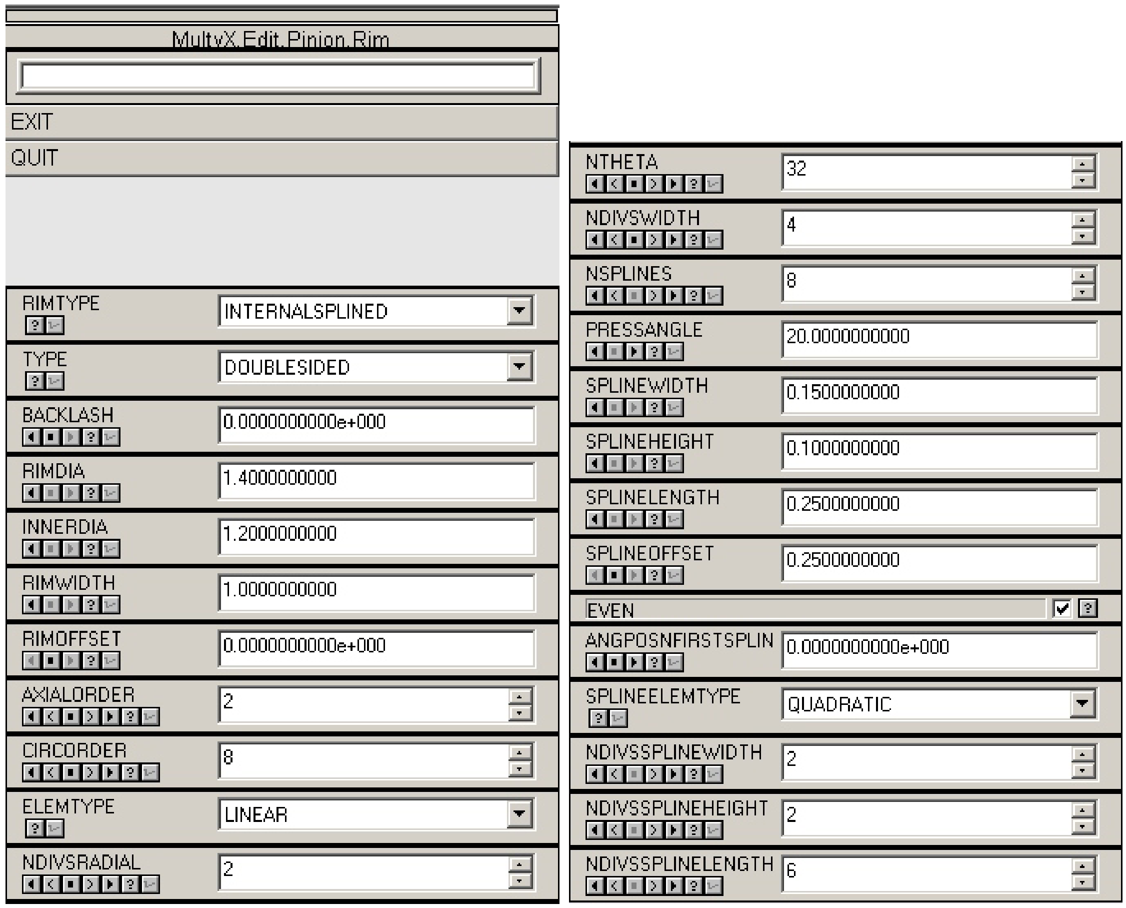


Figure 7.41: The menu for a splined rim on an external gear.

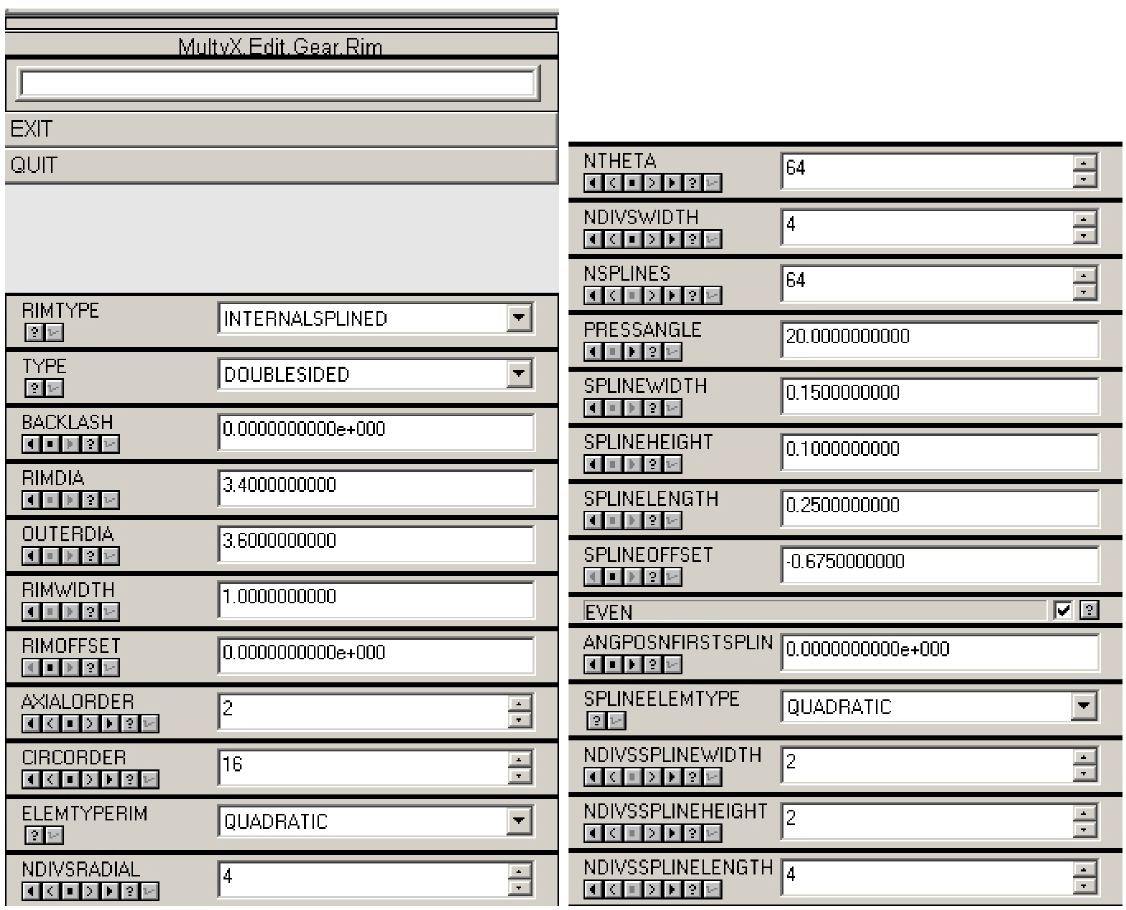


Figure 7.42: The menu for a splined rim on an internal gear.

Table 7.18: Splined rim parameters

Item	Description	Condition
TYPE	Switch, Type of contact at the spline. Available options are SINGLESIDED and DOUBLESIDED.	
BACKLASH	Float, Amount of backlash (Deg.) at the gear.	if(TYPE = DOUBLESIDED)
RIMDIA	Float, Rim diameter of the gear	
INNERDIA	Float, Inner dia. of the gear	if(GEARTYPE = EXTERNAL)
OUTERDIA	Float, Outer dia. of the gear	if(GEARTYPE = INTERNAL)
RIMWIDTH	Float, Width in the axial direction of rim.	
RIMOFFSET	Float, Offset in the axial direction of rim.	
AXIALORDER	Integer, Polynomial order in the face direction	
CIRCORDER	Integer, Fourier series order in the circular direction	
ELEMTYPE	Switch, Type of finite element	
NDIVSRADIAL	Integer, Number of elements in the radial direction	
NTHETA	Integer, Number of elements in the circular direction	
NDIVSWIDTH	Integer, Number of elements in the axial direction	
NSPLINES	Integer, Number of splines on the rim.	
PRESSANGLE	Float, Pressure angle (Deg) of the spline.	
SPLINEWIDTH	Float, Width of the spline.	
SPLINEHEIGHT	Float, Height of the spline.	
SPLINELENGTH	Float, Length of the spline.	
SPLINEOFFSET	Float, Axial offset of the spline.	
EVEN	Boolean, Whether splines are evenly distributed.	
ANGPOSNFIRSTSPLINE	Float, Angular position (Deg) of the first spline.	if(EVEN)
SPLINES	Integer, Spline number.	if(¬(EVEN))
ANGPOSNSPLINE	Float, Vector indexed by SPLINE, Angular position (Deg) of the spline.	if(¬(EVEN))
SPLINEELEMTYPE	Switch, Type of finite element. Available options are LINEAR, QUADRATIC and CUBIC	
NDIVSSPLINEWIDTH	Integer, No. of elems along the width.	
NDIVSSPLINEHEIGHT	Integer, No. of elems along the height.	
NDIVSSPLINELENGTH	Integer, No. of elems along the length.	

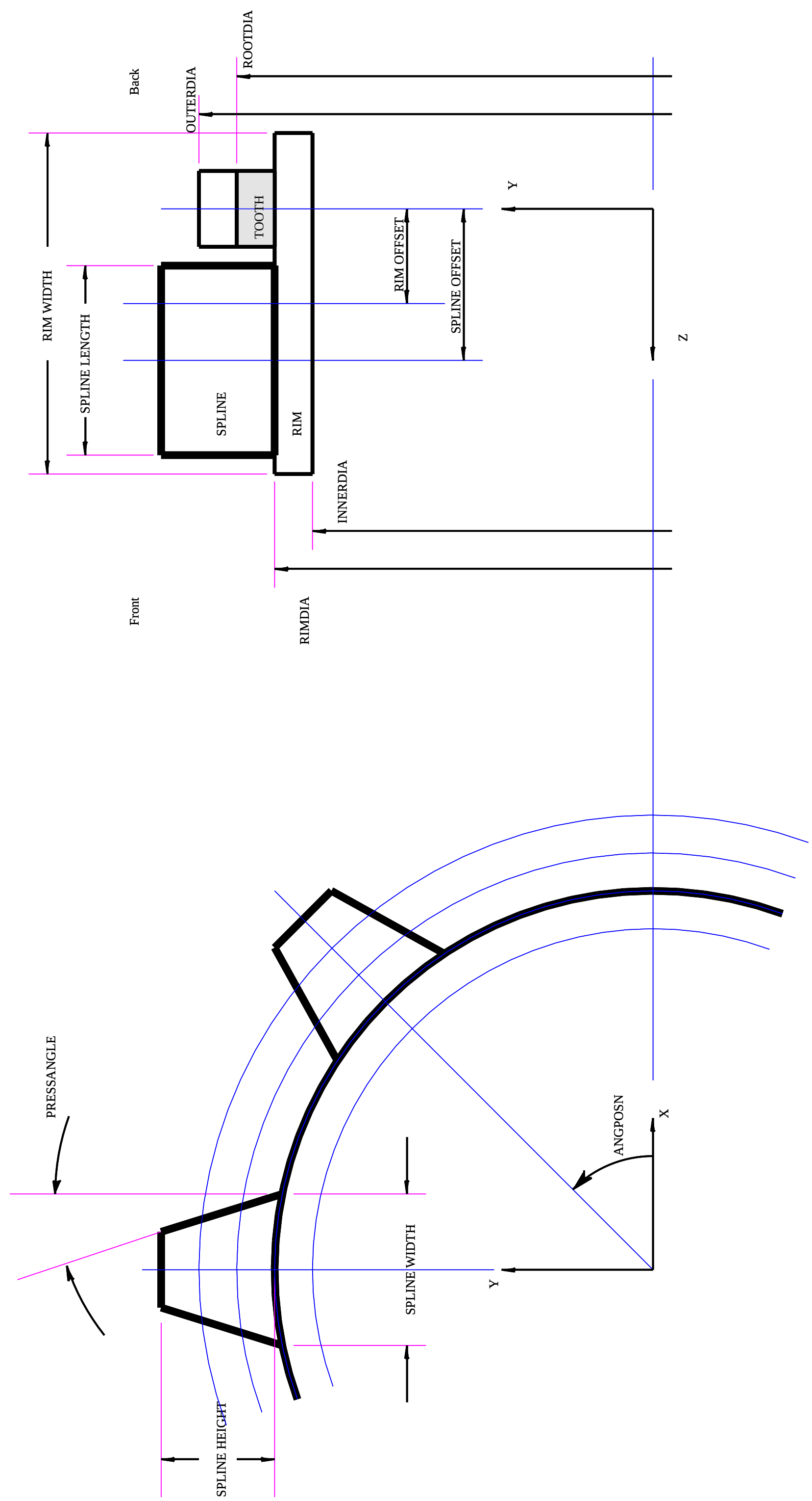


Figure 7.43: An external pinion or gear with an externally splined rim.

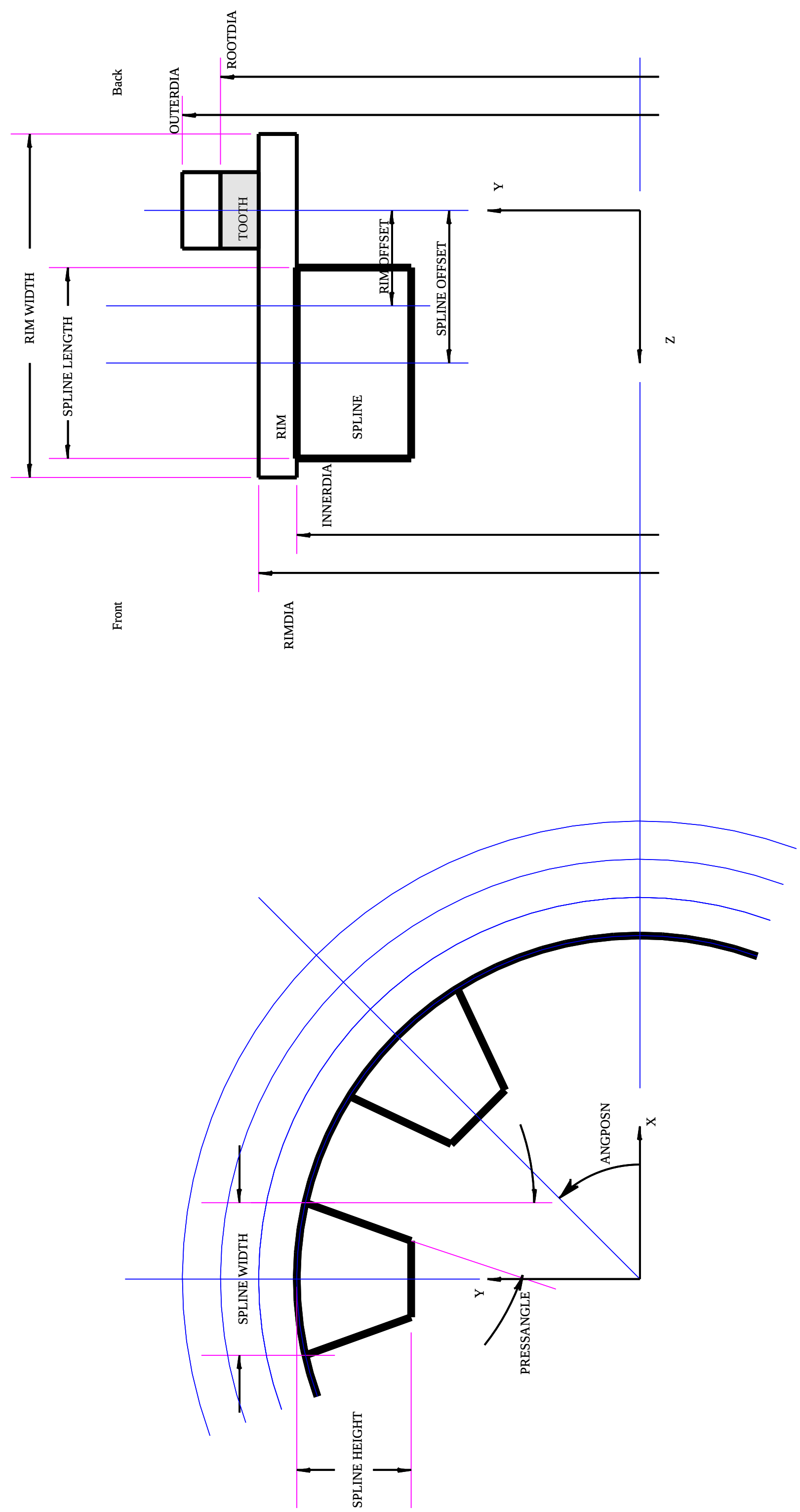


Figure 7.44: An external pinion or gear with an internally splined rim.

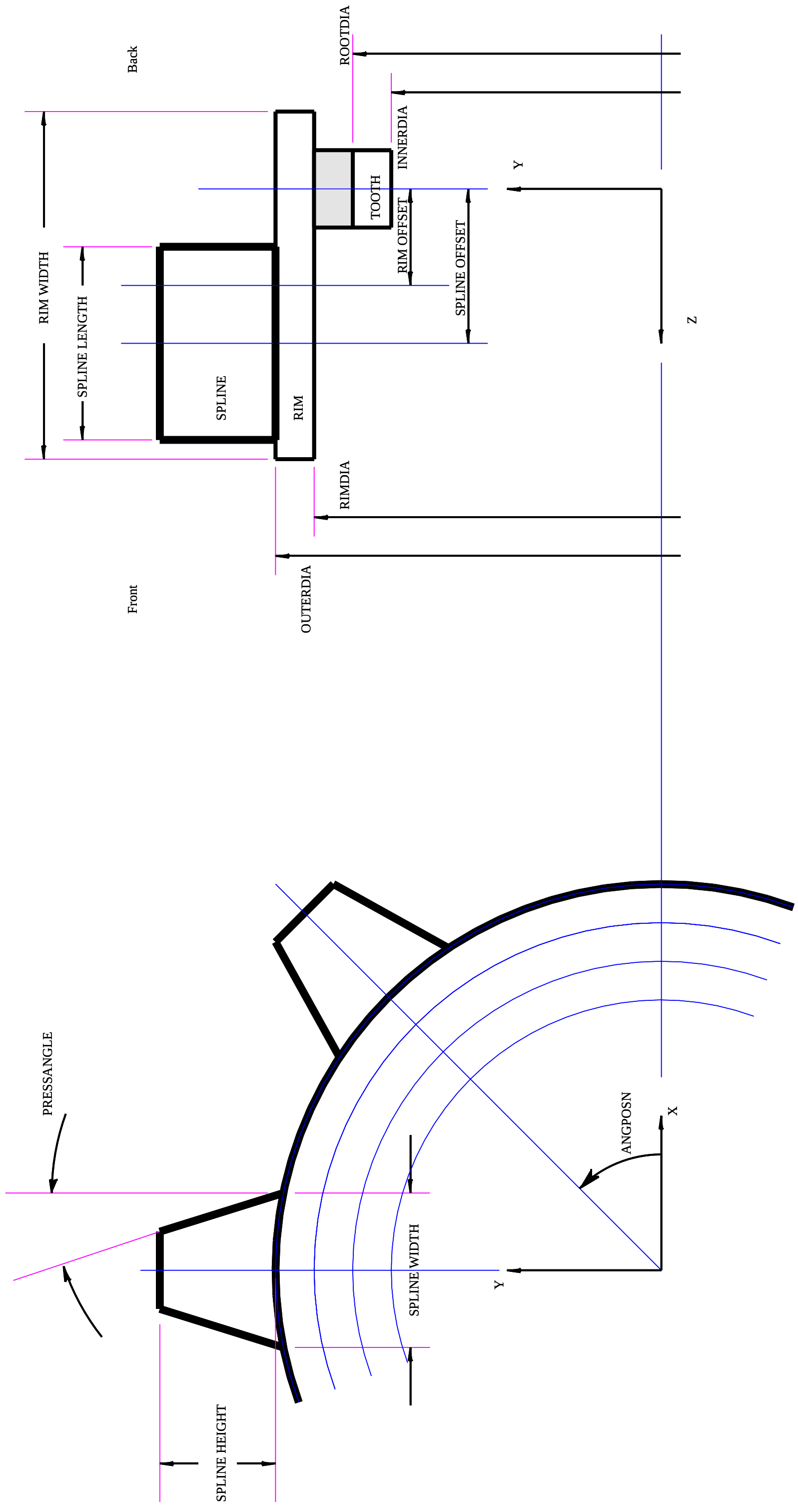


Figure 7.45: An internal gear with an externally splined rim.

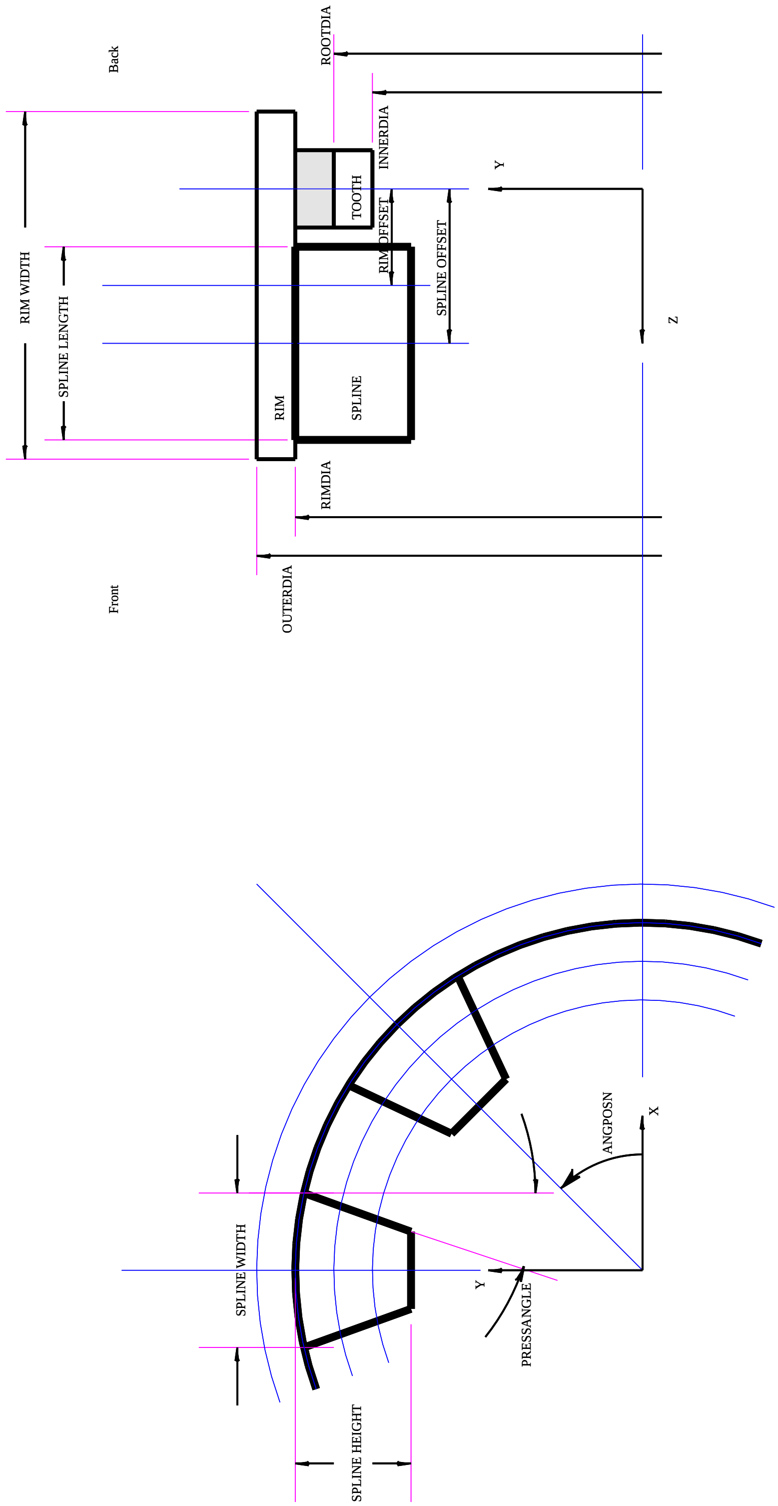


Figure 7.46: An internal gear with an internally splined rim.

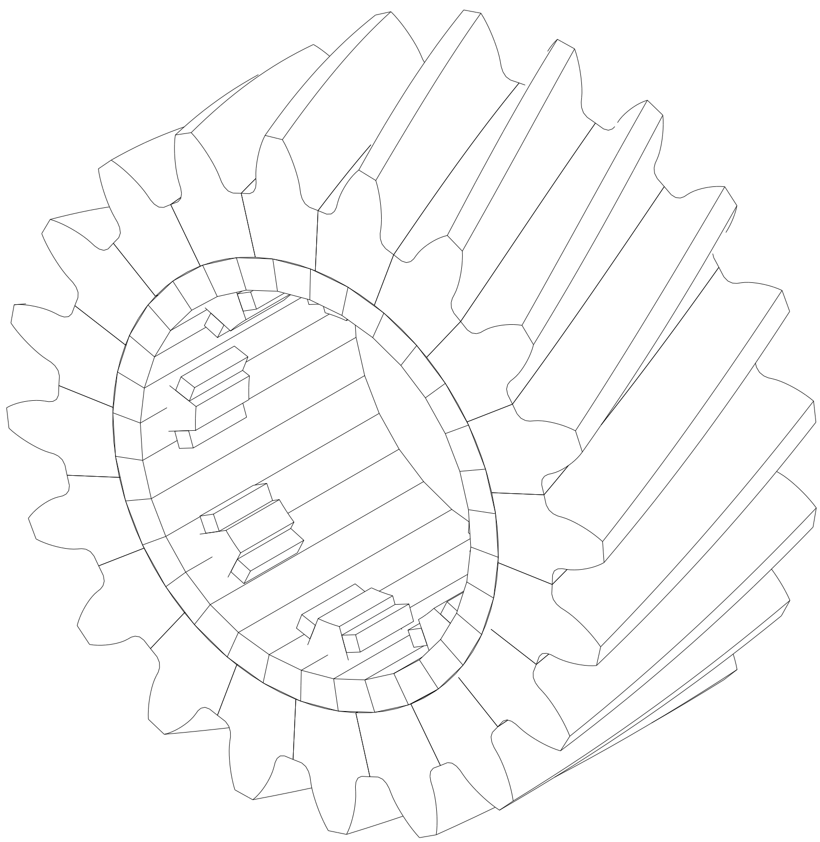


Figure 7.47: An example of an external gear with internal splines.

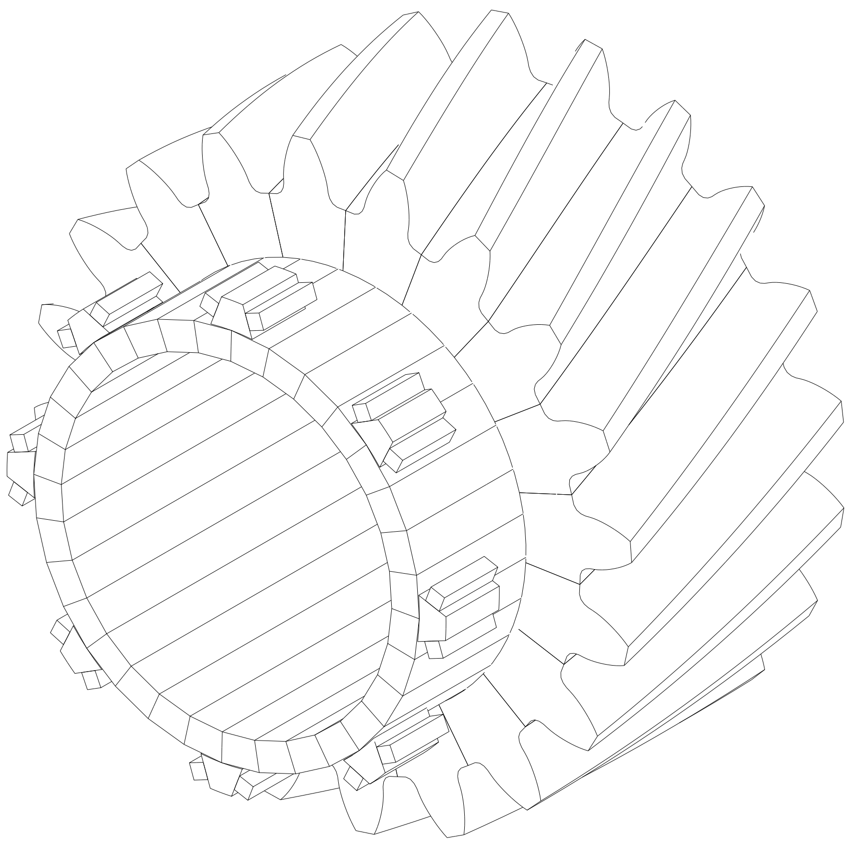


Figure 7.48: An example of an external gear with external splines.

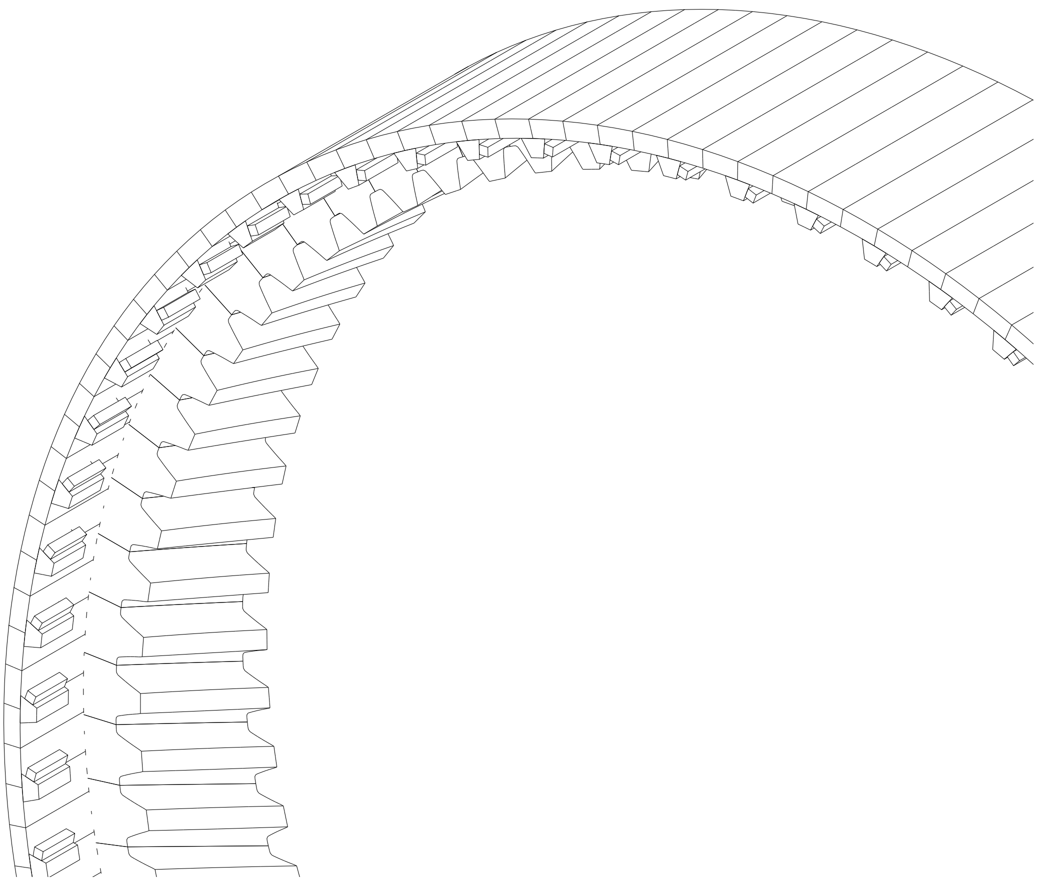


Figure 7.49: An example of an internal gear with internal splines.

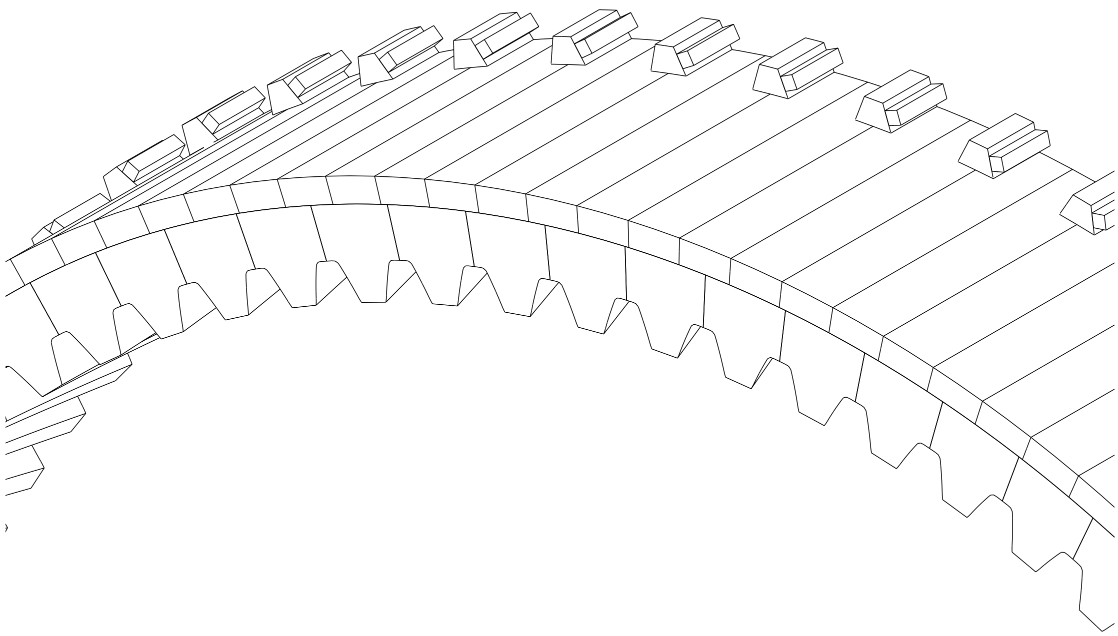


Figure 7.50: An example of an internal gear with external splines.

7.9 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 7.6 and 7.7), the submenu SHAFT appears. This submenu is shown in Figure 7.51.

The shaft is built of a number NSEGS of segments, as shown in Figure 7.52. 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 7.53). 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 7.54).

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.

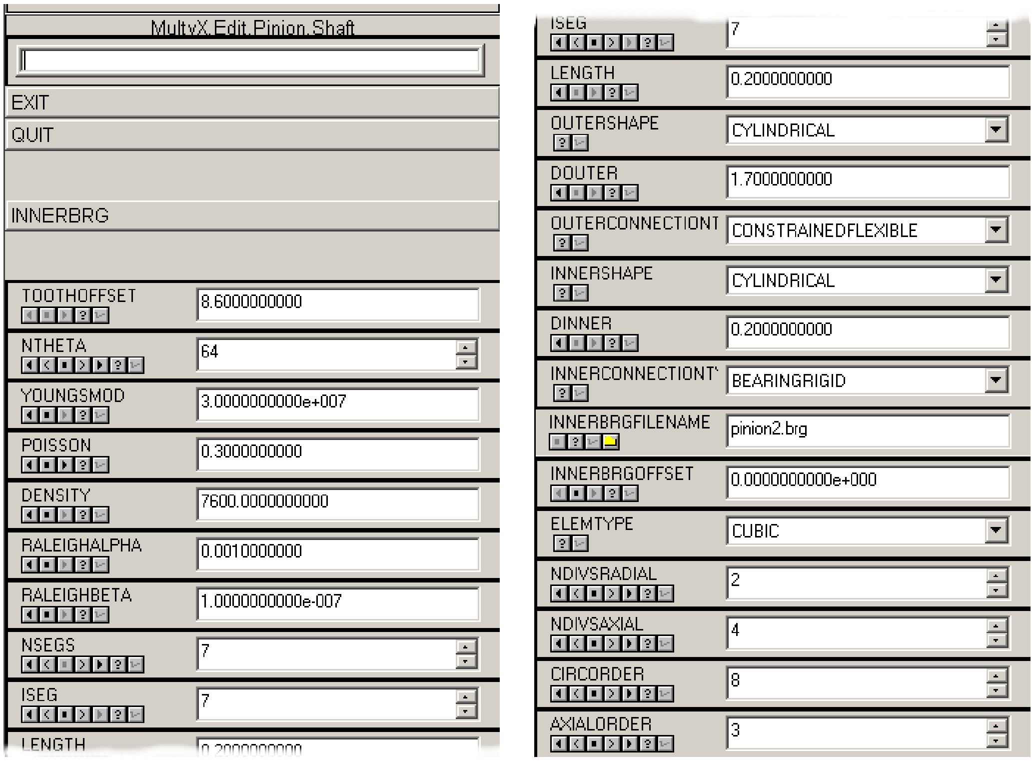


Figure 7.51: The menu for specifying shaft data.

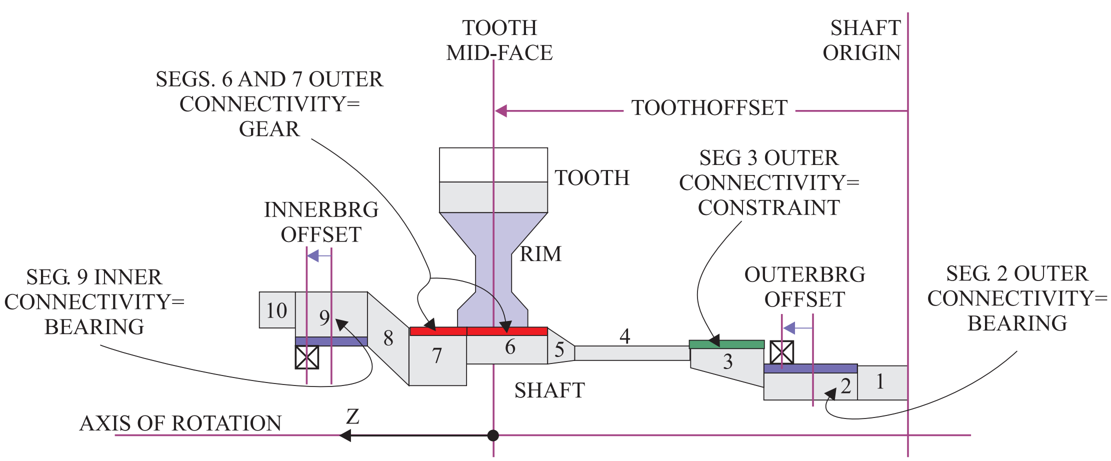
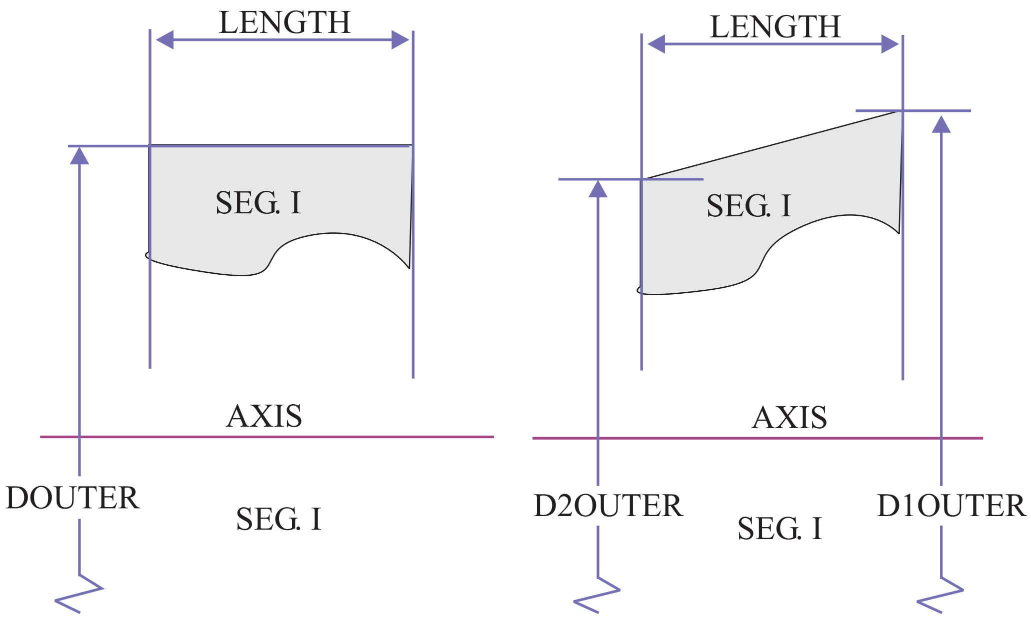
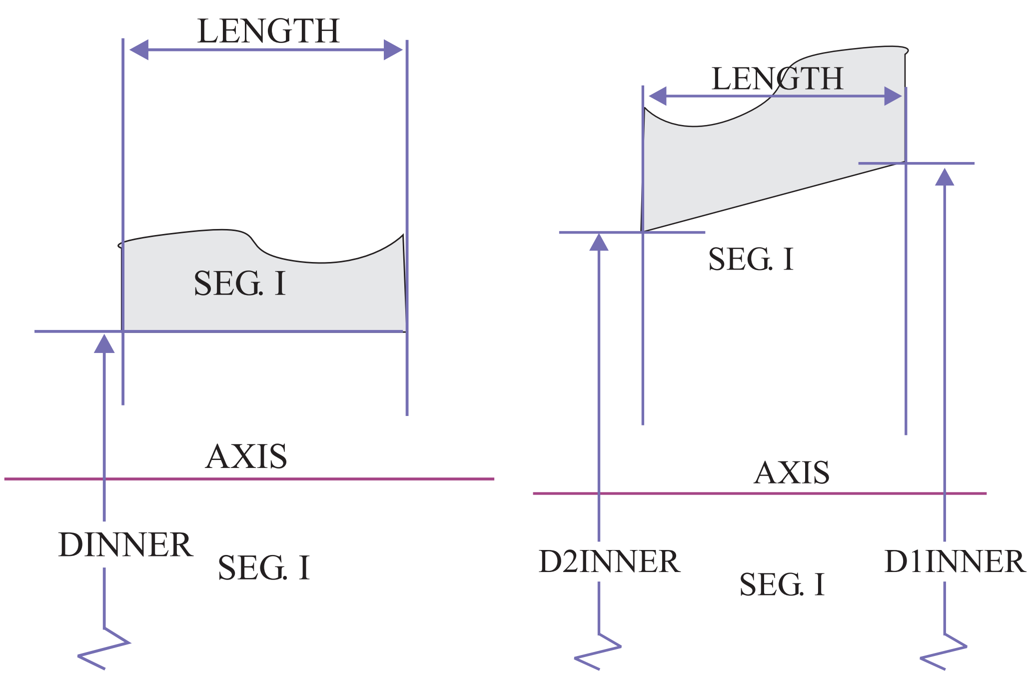


Figure 7.52: The shaft model.



(a) CYLINDRICAL (b) CONICAL

Figure 7.53: The dimensions of the outer surface of a shaft segment.



(a) CYLINDRICAL (b) CONICAL

Figure 7.54: The dimensions of the outer surface of a shaft segment.

Chapter 8

Running an Analysis

The analysis is started by using STARTANAL command of Figure 4.5.

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 4.5). Additional analysis parameters and settings are controlled through the SETUP command.

8.1 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 8.1 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 helical pair analysis, there are two bodies, the pinion and the gear. If splines are used on the pinion, an additional body is created for the pinion shaft. Similarly if splines are used on the gear, then a gear shaft body is also created.

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. If splines are used on the pinion or gear, then there will be two more surfaces for the pinion or gear body, SPLINESURF1 and SPLINESURF2, which are the two sides of the spline teeth.

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.

8.2 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 8.2 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. Note that the template files(medium.tpl, fineroot.tpl and finest.tpl) are only used to model the tooth. The mesh for the rim and the lumped mass is different from that of the tooth model. 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.

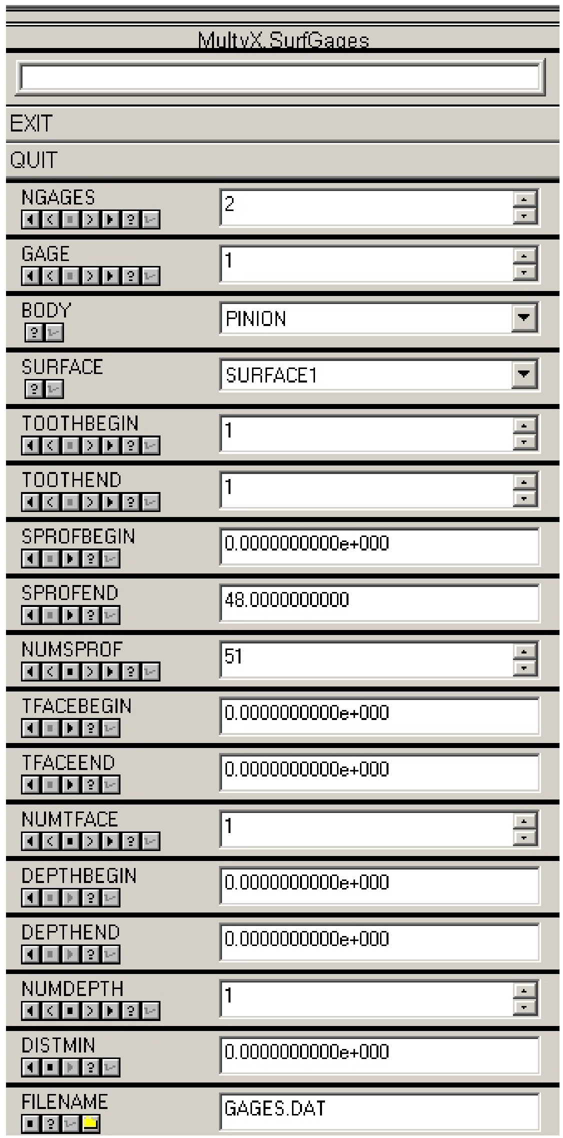


Figure 8.1: The surface gage menu

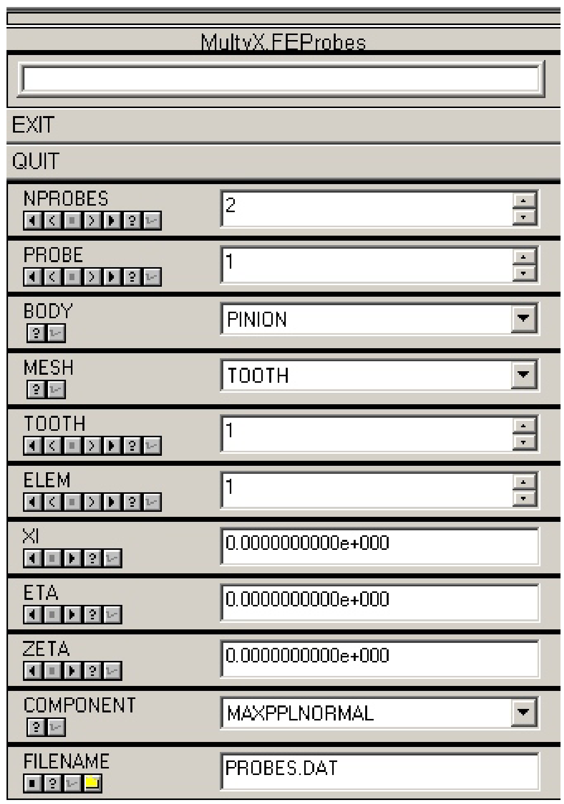


Figure 8.2: The finite element probe menu

8.3 Load sensors

Load sensors are used to measure the contact loads generated at the contact surfaces. Figure 8.3 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.

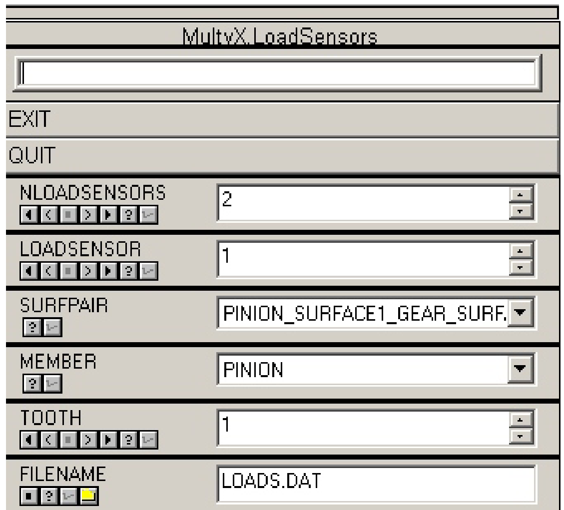


Figure 8.3: The load sensor menu

8.4 Specifying a contact grid

Figure 8.4 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 8.9). 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 8.9), 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 8.9) is user selectable. The dimension of the grid cells in the profile direction Δs (DSPROF in Figure 8.9) 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 8.5). 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 8.6). If the grid size is correct, a variation of contact pressure similar to that in Figure 8.7 should be obtained.

Figure 8.8 shows an example of a contact grid set up on a pair of contacting teeth.

8.5 The setup menu

Figure 8.9 shows the analysis setup menu accessed by using the SETUP command in the main menu.

The CONSERVEMEMORY flag should be turned on only when the problem size is too large, and there is a chance that the system might get low on memory. Using this option might increase the CPU time.

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 of time ranges(NRANGES). The time step DELTATIME, solution method SOLMETHOD and the number of time steps NTIMESTEPS can be specified separately for each time range.

For a helical gear pair, we are usually only interested in running the model in static mode. So SOLMETHOD should be kept in this default state. At present, due to CPU limitations, we cannot run dynamic analysis for *Helical3D*.

It is possible to control the operating speed in each time range by specifying a speed factor at the beginning of the range(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. The torque factor at the end of a time range need not be same as the torque factor at the beginning of the next time 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 SAVEFILENAME 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. The maximum file size on a 32 bit machine 2 gigabytes. If the size of the post-processing file exceeds this limit because there are a very large number of time steps, then the results of the entire analysis will be lost. By turning on the SPLITPOSTPROCFILE flag, a separate data file will be written for each time step under a folder with the name specified by POSTFILENAME. The user does not need to know the specifics of the structure of this folder.

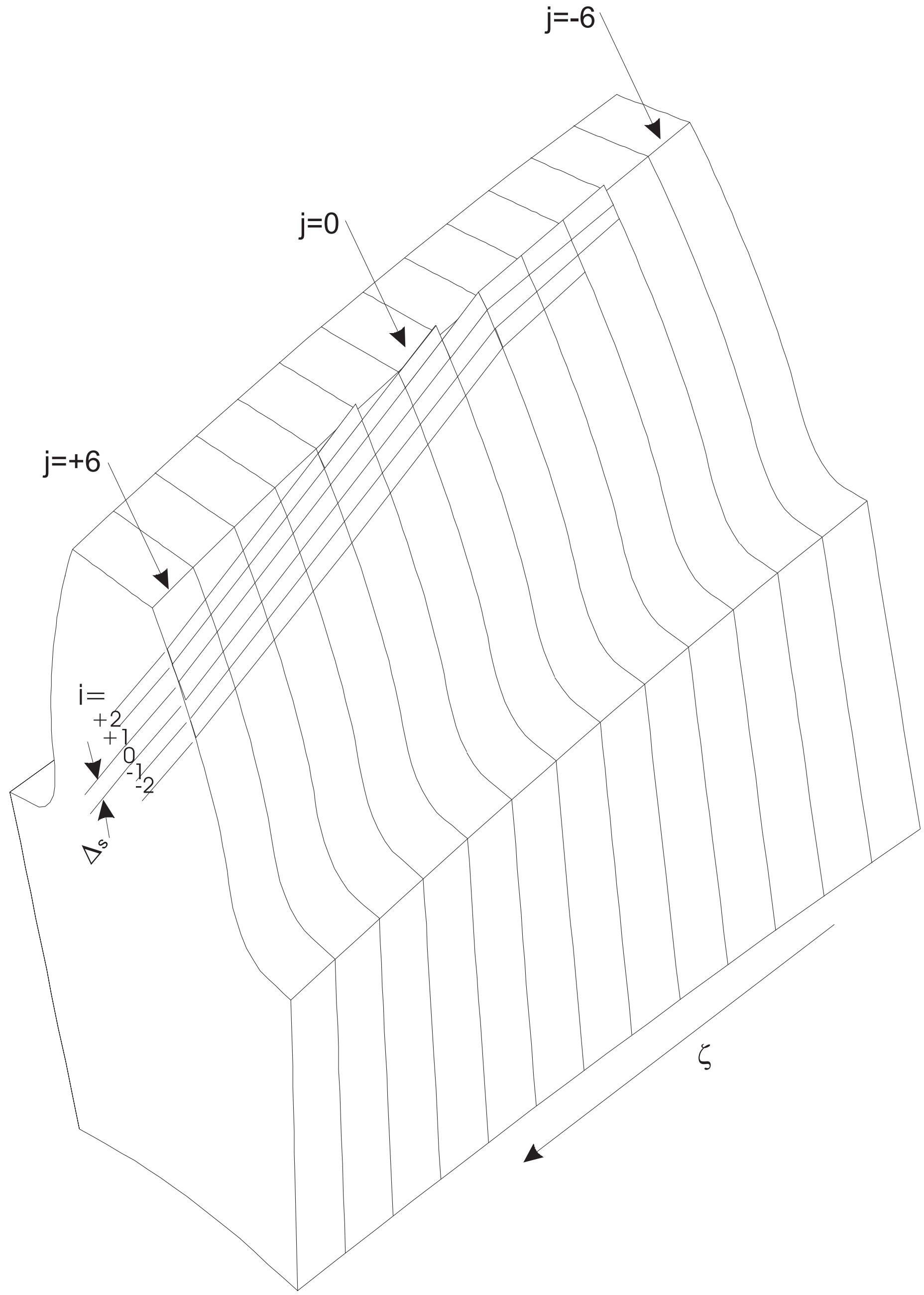


Figure 8.4: Computational grid in the contact zone of the gears

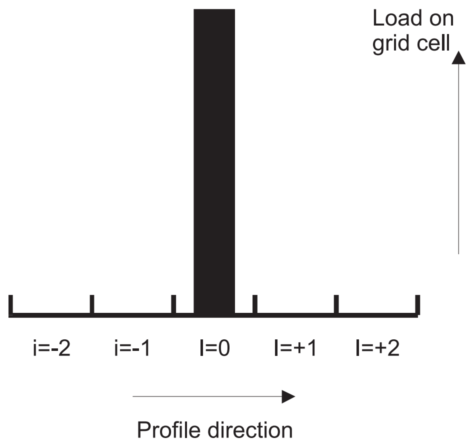


Figure 8.5: Contact pressure distribution across the width of contact obtained when the contact grid is too wide.

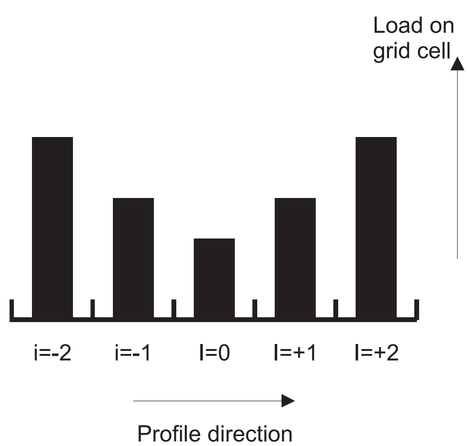


Figure 8.6: Contact pressure distribution across the width of contact obtained when the contact grid is too narrow.

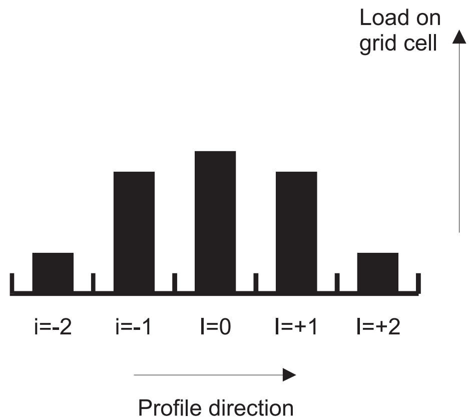


Figure 8.7: Contact pressure distribution across the width of contact obtained when the contact grid is correct.

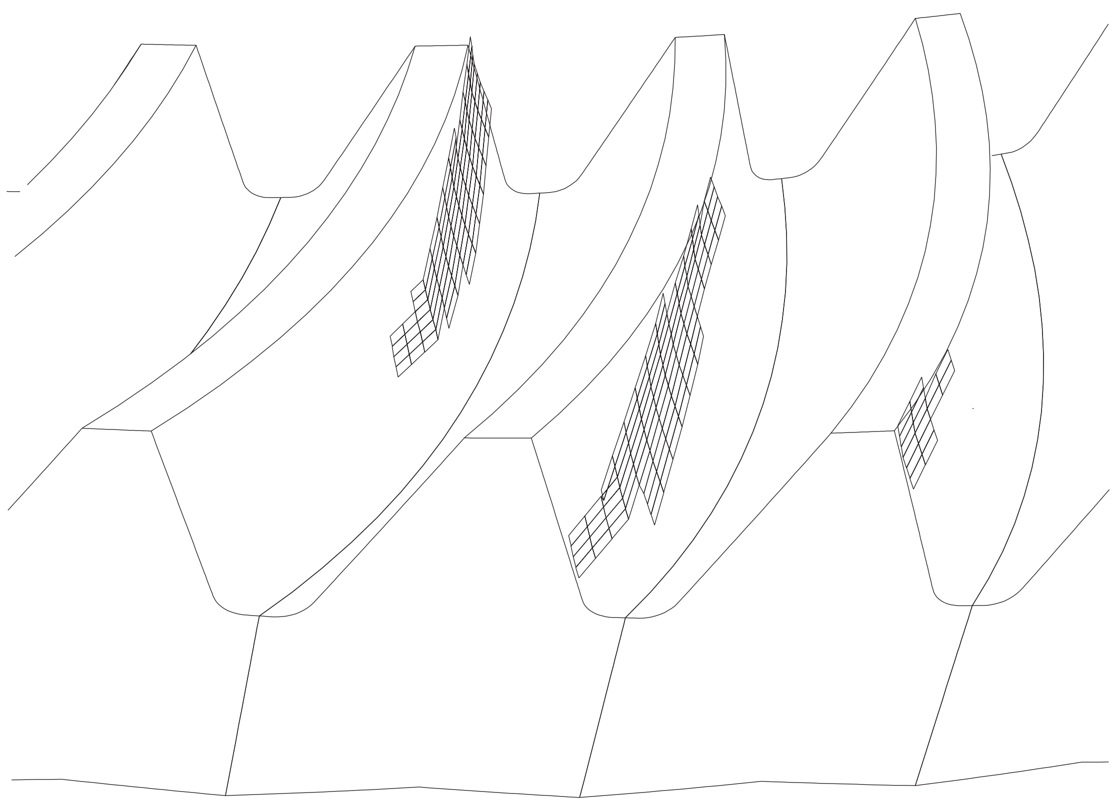


Figure 8.8: An example of a contact grid set up on a pair of contacting teeth.

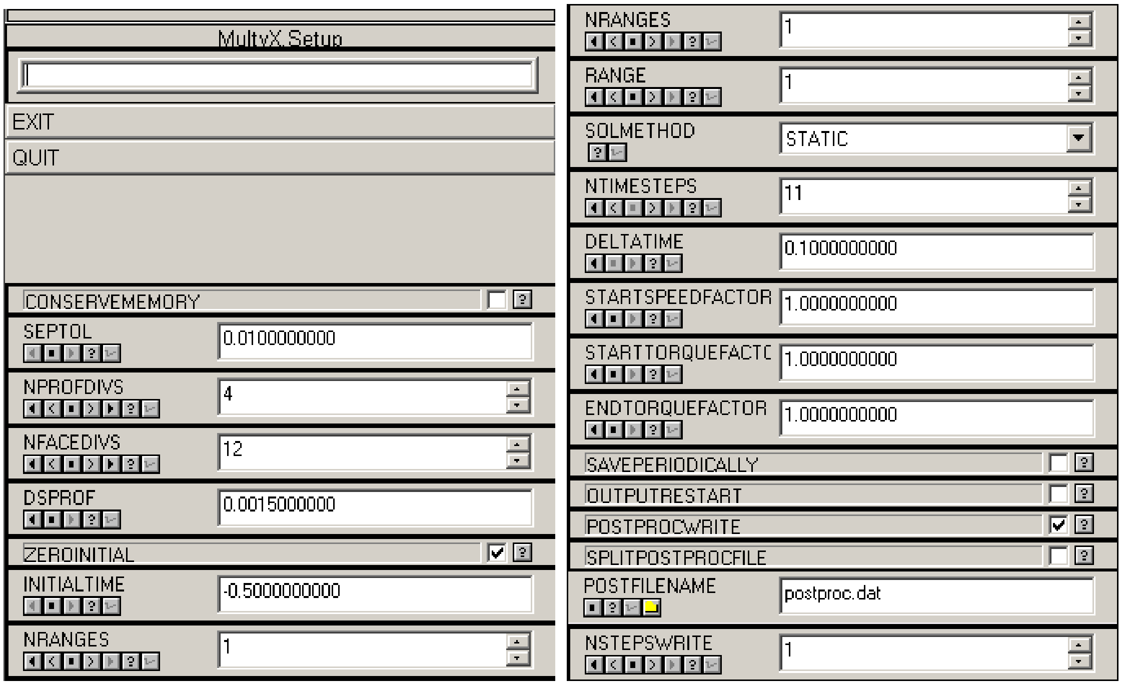


Figure 8.9: The setup menu

8.6 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.

The deformation and reaction forces generated in each bearing are also saved in data files during analysis. These data files are named after the bearings. For example, the file `PINIONBRGRES.DAT` contains the results for the pinion bearing, `GEARBRGRES.DAT` contains results for the gear bearing. 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 bearing deformation, u_x, u_y, u_z, θ_x, θ_y, and θ_z. The last 6 columns contain the 6 components of bearing reaction, F_x, F_y, F_z, M_x, M_y, and M_z.

The bearing deformation components are those of bearing race 1 with respect to race 2, measured in race 2. For the pinion as well as the gear, race 2 is attached to ground, and its axes are parallel to those of the fixed (global) reference frame. Thus the deformation components are along the global axes.

The reaction forces are conjugates to the deformation values. For instance, if you specify the torque you need not specify the displacement, θ or vice-versa. Also note that a positive deformation component causes a positive reaction force.

Chapter 9

Pre- and Post-processing

The PREPROC command in the main menu leads to the pre-processing menu shown in Figure 9.1. The POSTPROC command leads to the dialog box shown in Figure 9.2, 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 9.3 comes up.

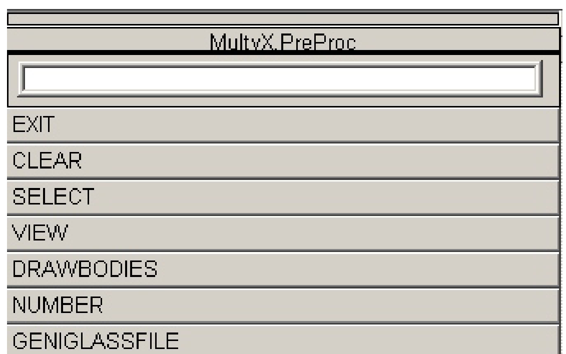


Figure 9.1: The pre-processing menu.

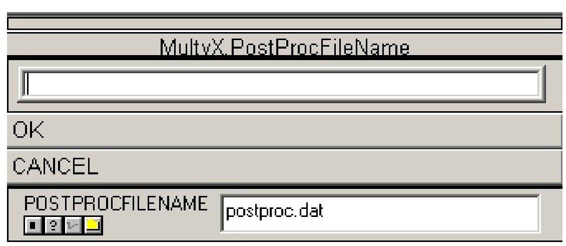


Figure 9.2: The post-processing file name dialog box.

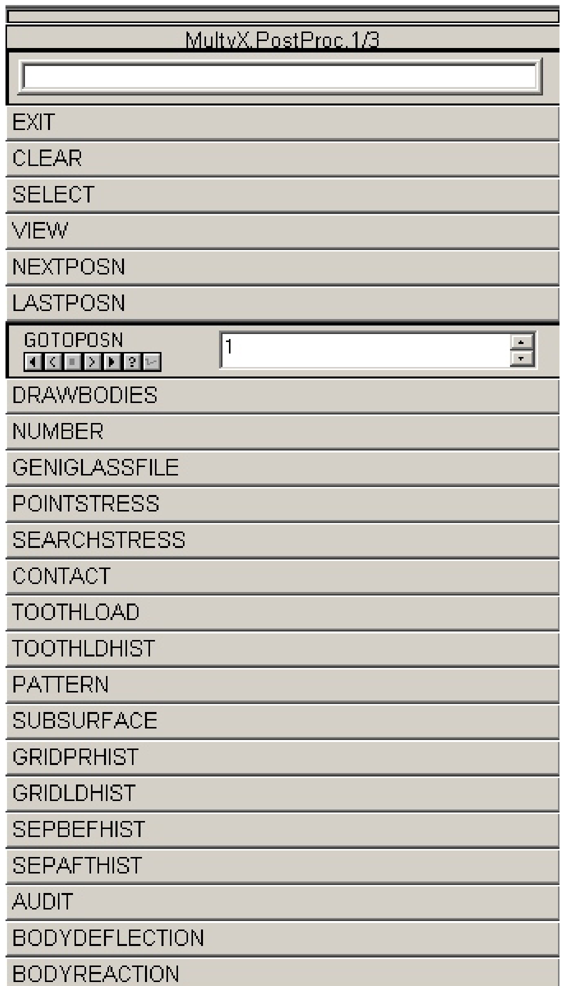


Figure 9.3: The post-processing menu.

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 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.

9.1 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 9.4. The objects that should be drawn are selected from this menu.

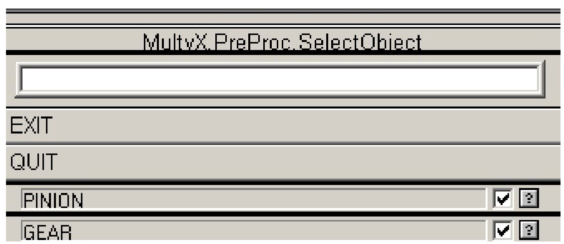


Figure 9.4: The body selection menu.

9.2 View parameters

The VIEW menu controls the appearance of the drawings. In the pre-processing view menu shown in Figure 9.5, 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 9.6) 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.

If the LOADS option is not checked (Figure 9.7), 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.

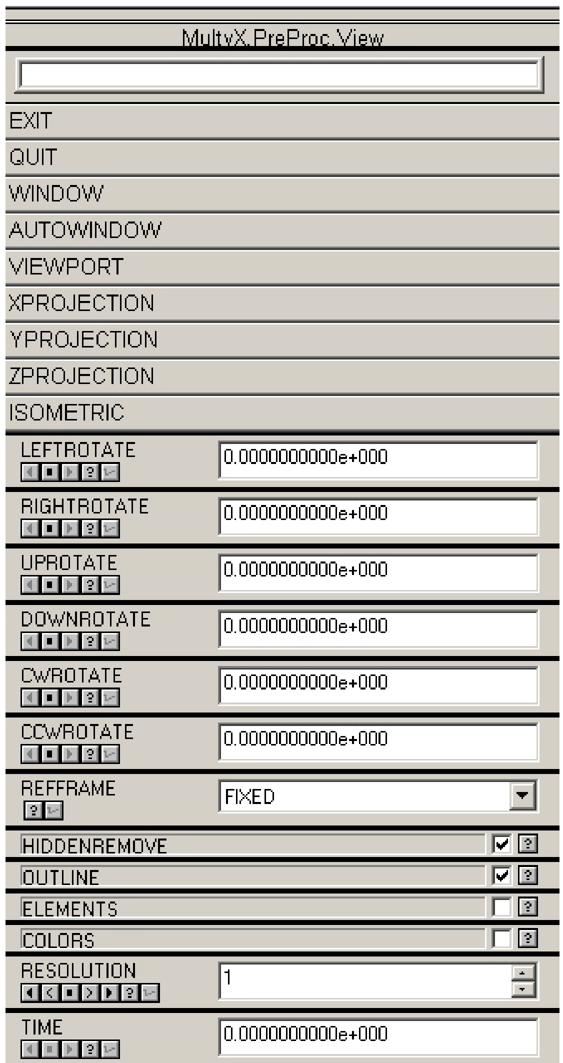


Figure 9.5: The view menu in pre-processing mode with the LOADS option disabled.

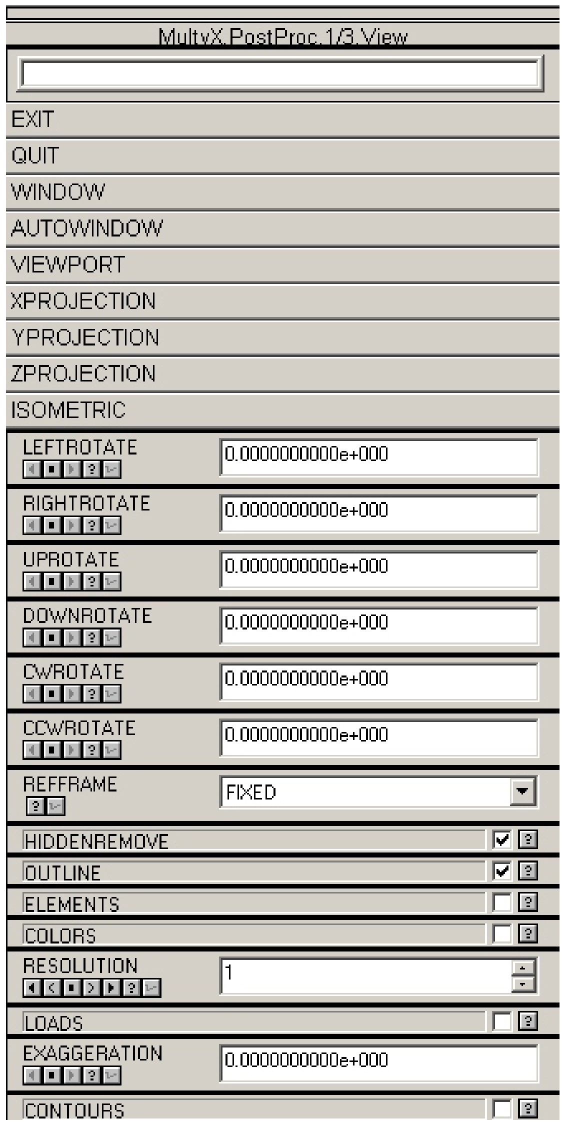


Figure 9.6: The view menu in post-processing mode.

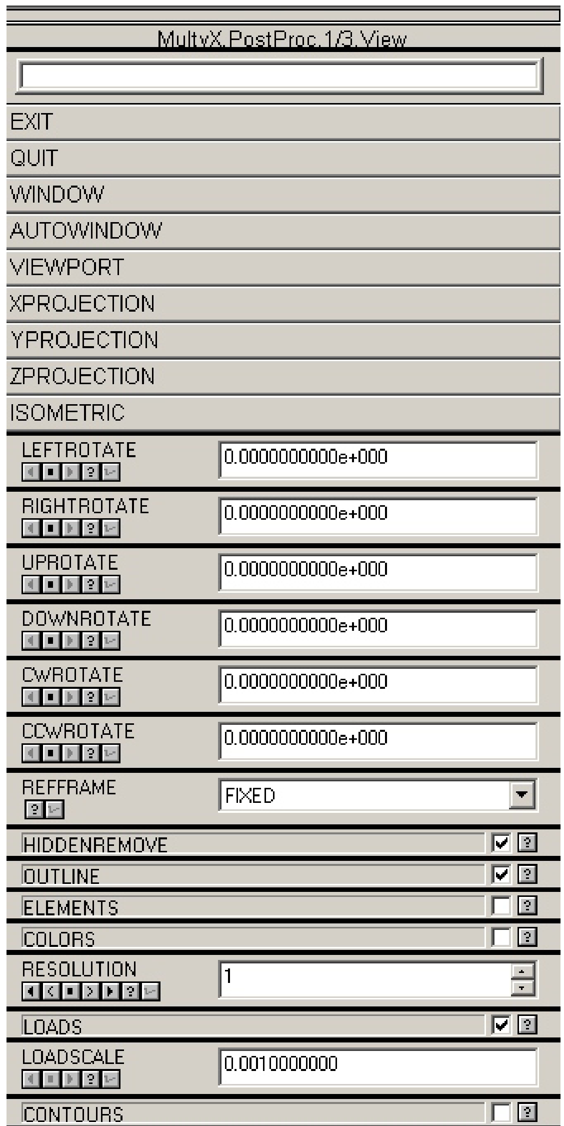


Figure 9.7: The view menu in post-processing mode with the LOADS option enabled.

9.3 The DRAWBODIES command

After an appropriate view and objects have been selected, the DRAWBODIES command in the pre- and post-processing menus (Figures 9.1 and 9.3) will generate a drawing. Figures 9.8 and 9.9 show examples of drawings generated by Multyx in the post-processing mode.

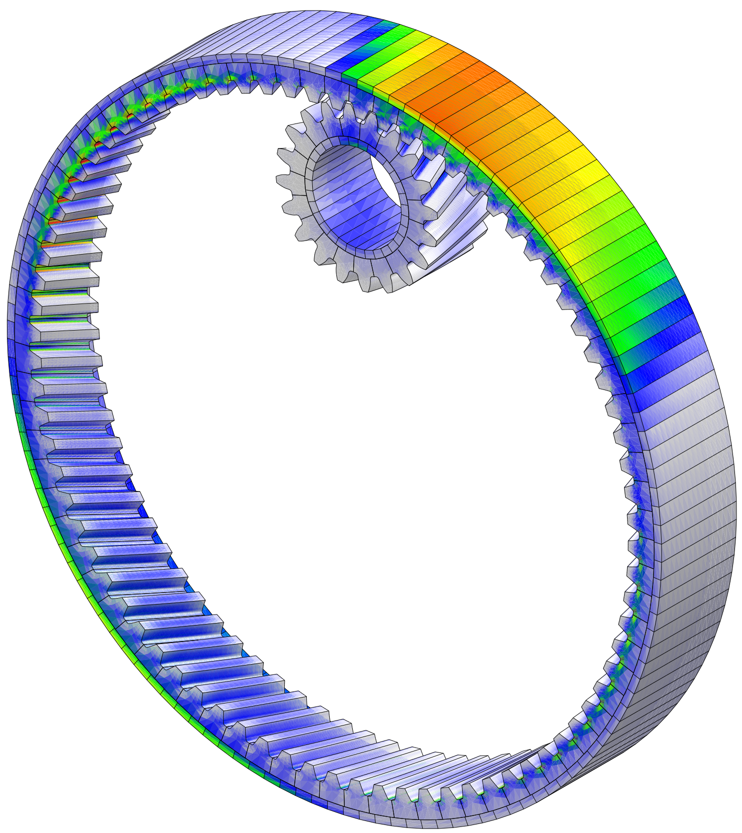


Figure 9.8: An example of a drawing made in post-processing mode.

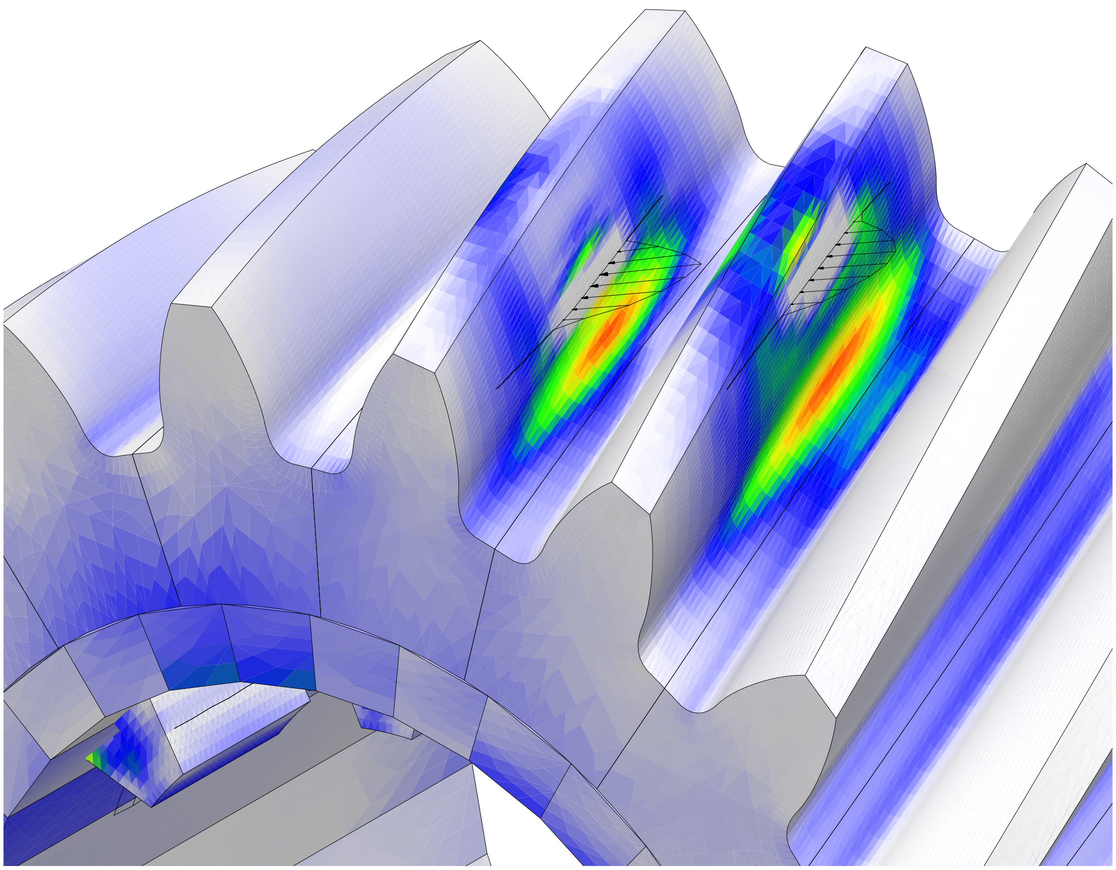


Figure 9.9: An example of a drawing made in post-processing mode.

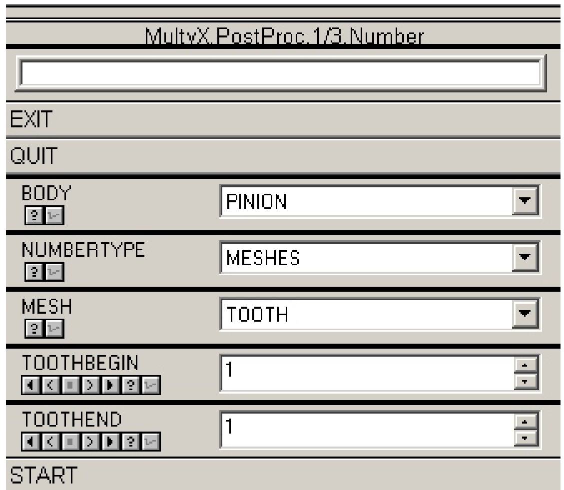


Figure 9.10: The NUMBER menu.

9.4 The NUMBER command

The NUMBER command in the pre- and post-processing menus (Figures 9.1 and 9.3) lead to the numbering menu shown in Figure 9.10. This menu is used to generate tooth and surface numbering, as shown in Figure 9.11.

9.5 The TOOTHLOAD command

The TOOTHLOAD command in the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.12. This menu is used to generate a graph of tooth load vs. time. The SURFACE-PAIR 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 9.13 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.

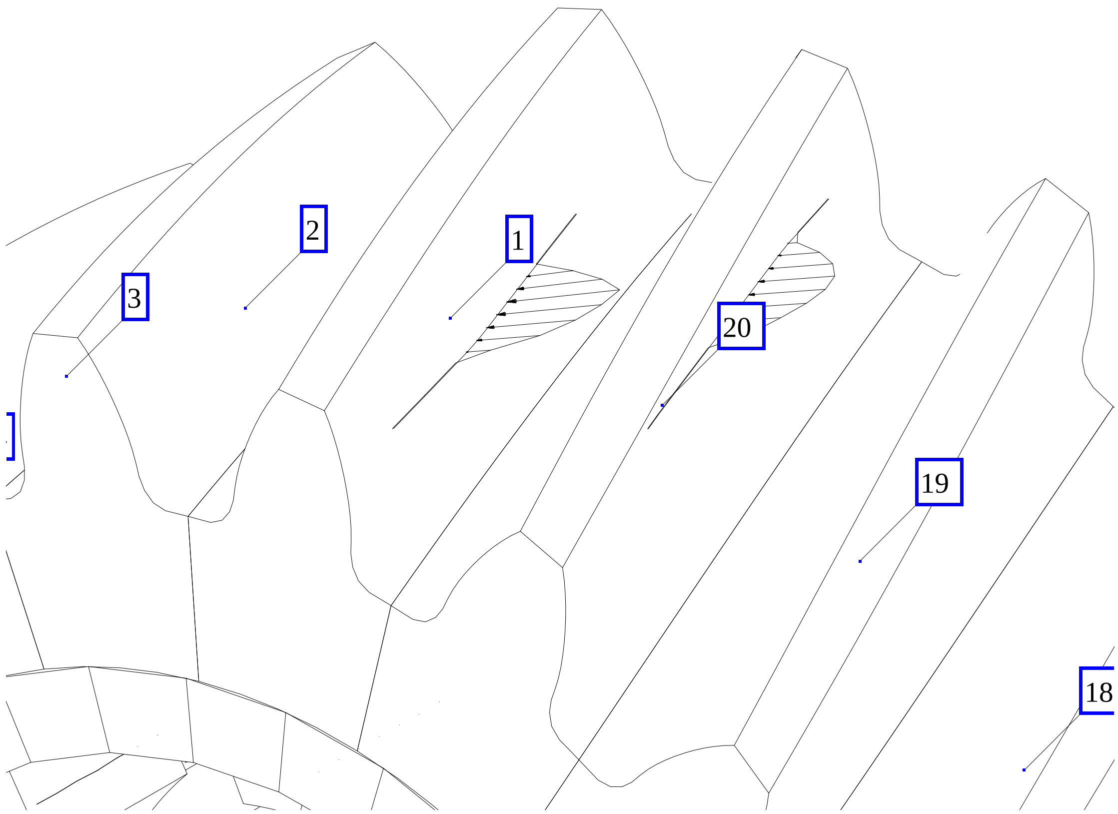


Figure 9.11: Tooth numbering superimposed on a pinion drawing, using the NUMBER command.

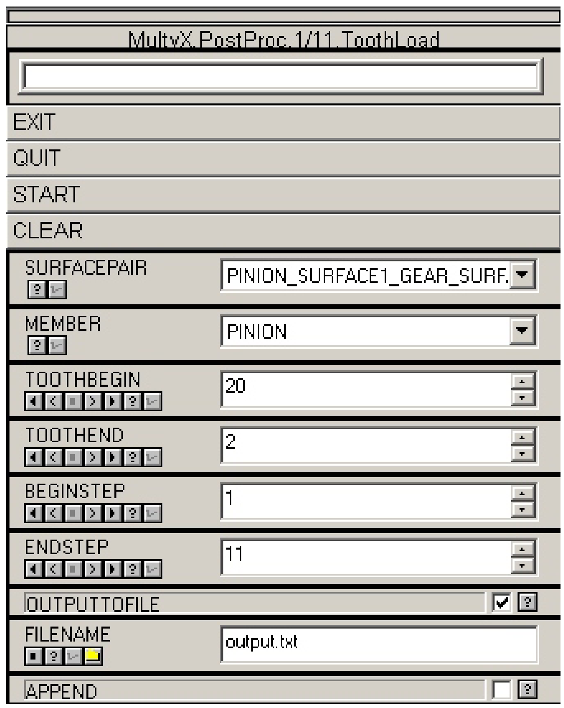


Figure 9.12: The TOOTHLOAD menu.

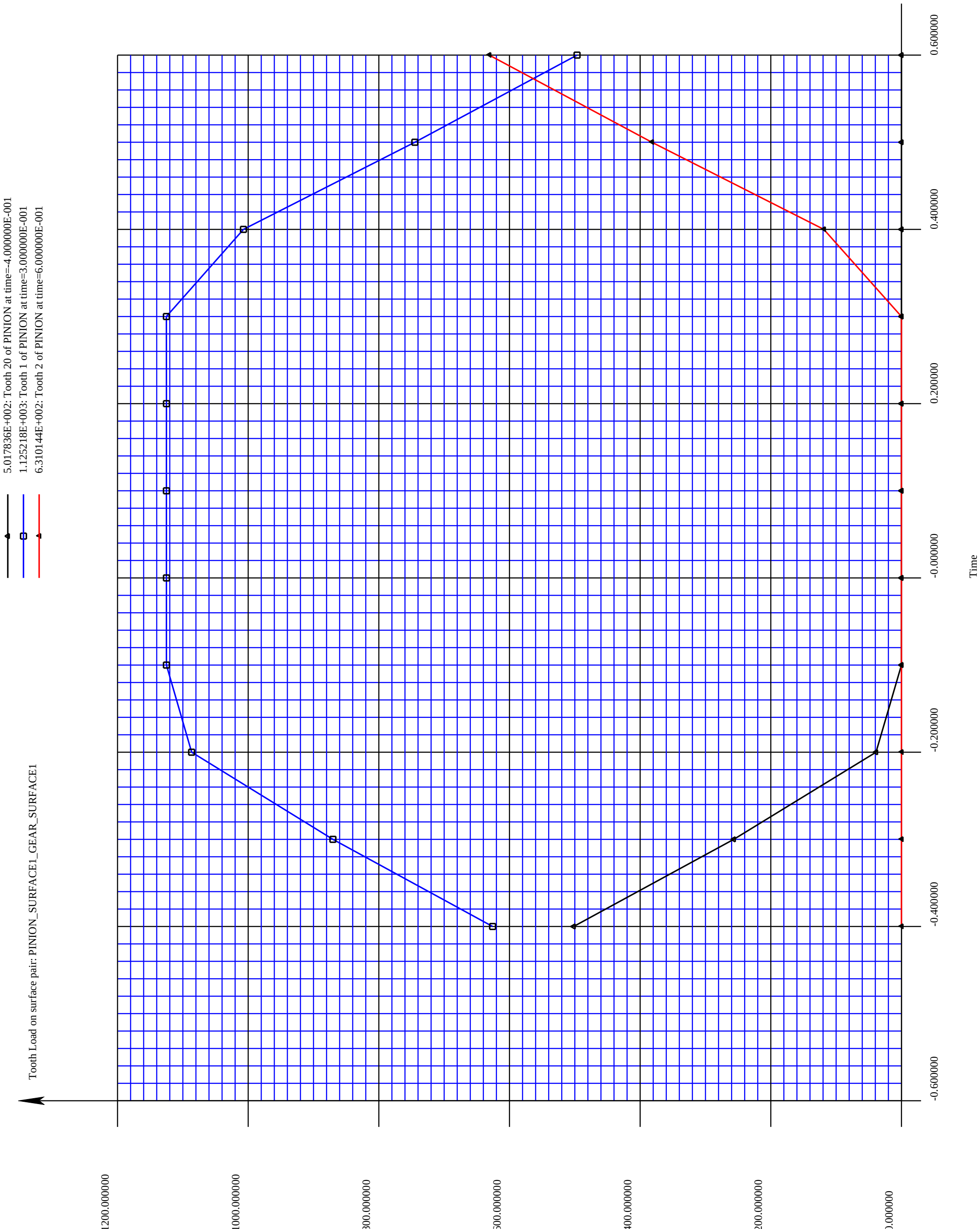


Figure 9.13: The tooth load vs. time graph generated by the TOOTHLOAD menu.

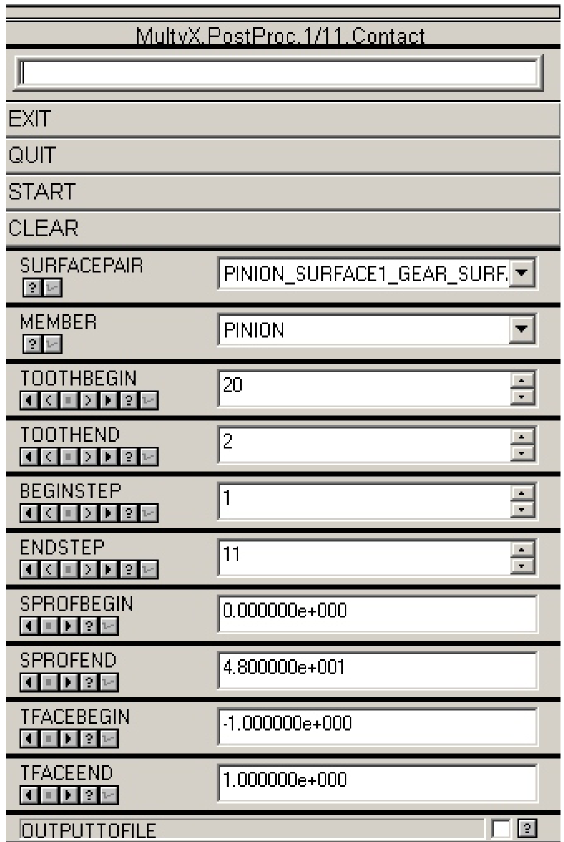


Figure 9.14: The CONTACT menu.

9.6 The CONTACT command

The CONTACT command in the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.14. 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. 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 9.15 shows a graph of contact pressure vs. time over the entire surface of a pinion tooth. Very high contact pressures are observed near the tips of the pinion and gear teeth. These high contact pressures near the tips may not be physically real and hence the results obtained could be misleading. The user should be careful when analysing the results at the tips or edges since the contact area is small which could lead to inaccurately high pressure values in some cases. This high contact pressure near the edges can be filtered out by restricting the search range, as shown in Figure 9.16.

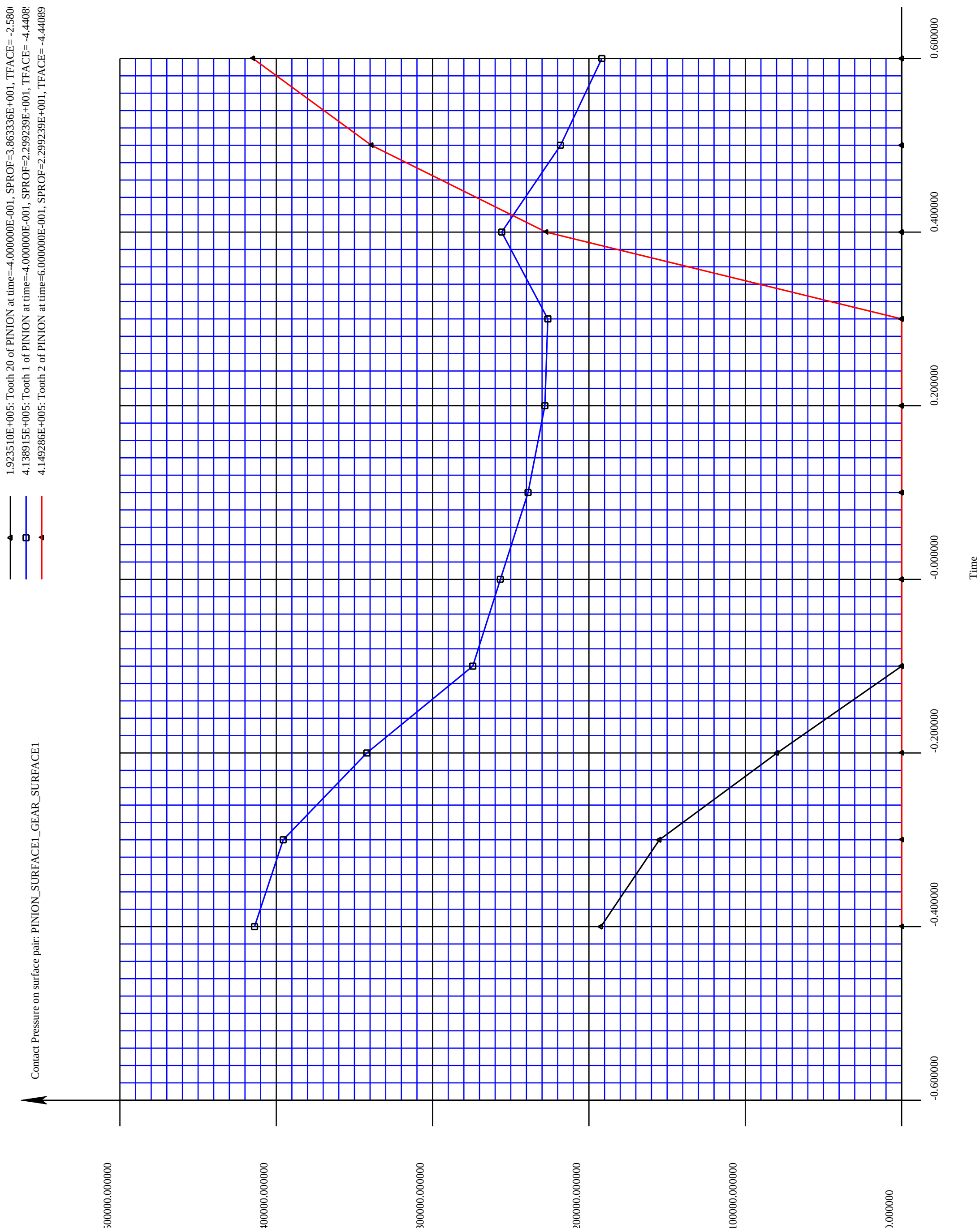


Figure 9.15: The tooth contact pressure vs. time graph generated by the CONTACT menu.

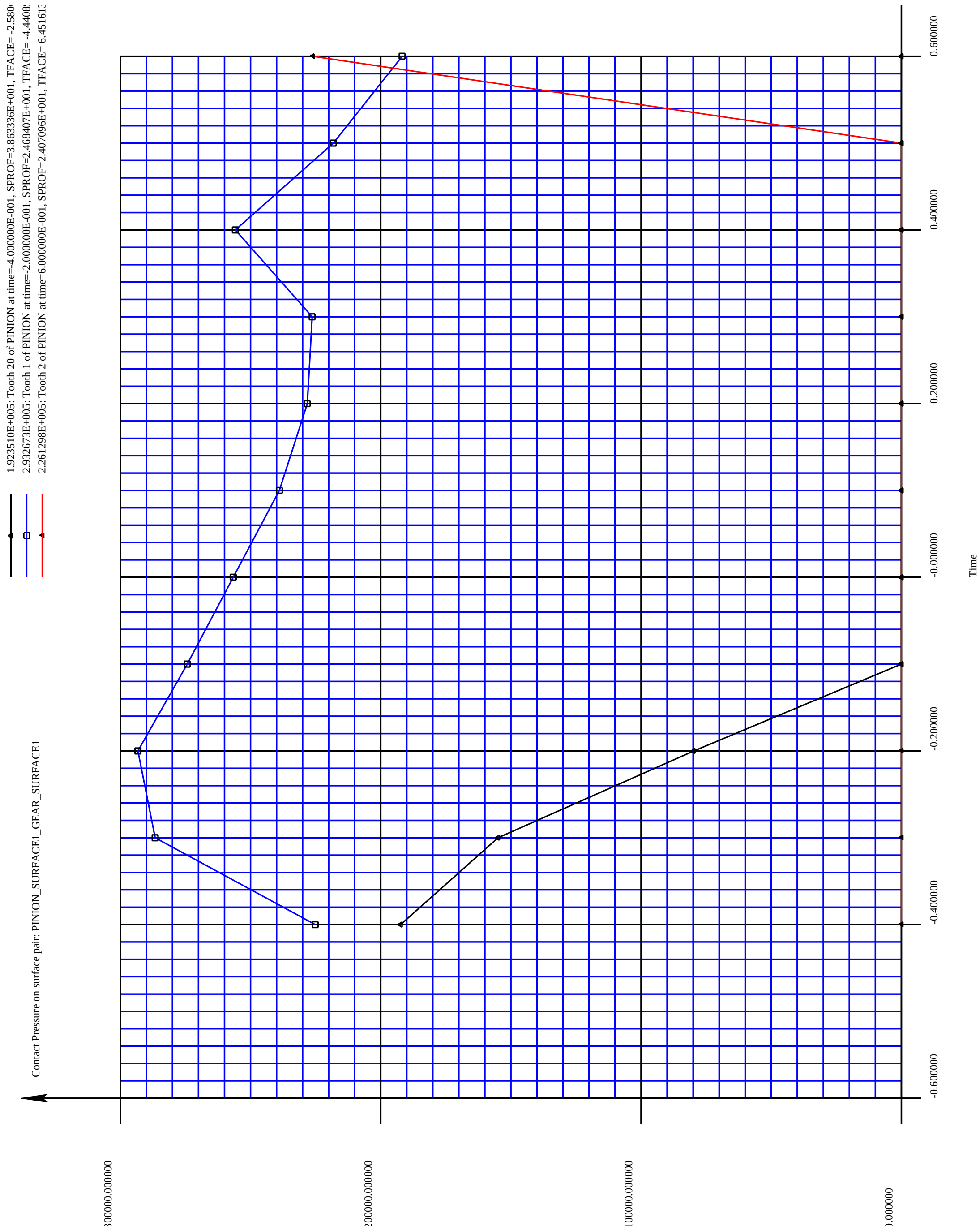


Figure 9.16: The tooth contact pressure vs. time graph generated by the CONTACT menu with the search range limited to $24 \leq s_{prof} \leq 46$.

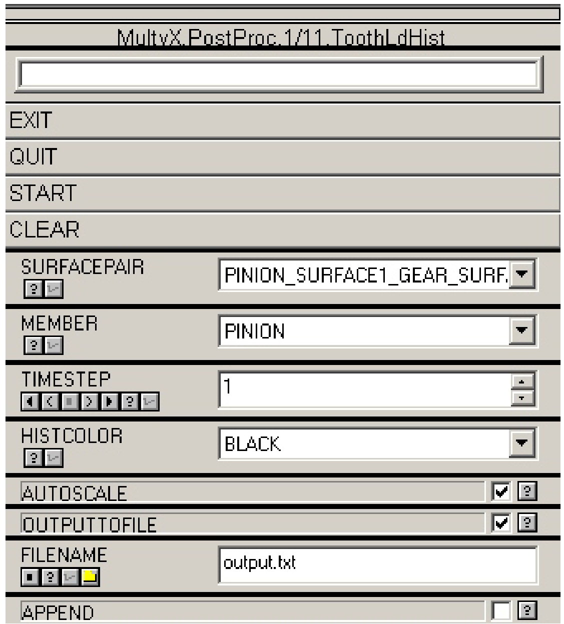


Figure 9.17: The TOOTHLDHIST menu.

9.7 The TOOTHLDHIST command

The TOOTHLDHIST command in the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.17. This menu is used 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 9.18.

9.8 The SUBSURFACE command

The SUBSURFACE command in the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.19. This menu is used to generate a graph of subsurface stresses vs. depth under the most critical point in the contact zone. Subsurface stresses are especially important in cases where there is failure away from the contacting surface. Shear stresses are present away from the surface. Also in some cases the gear material away from the surface is slightly weak compared to the material at or near the surface. In such cases also subsurface stresses are significant. 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. So DEPTHBEGIN should never be set to zero.

The stress component is selected in the COMPONENT box. Options available are MAXP-PLNORMAL (the maximum principal normal stress s_1), MINPPLNORMAL (the minimum

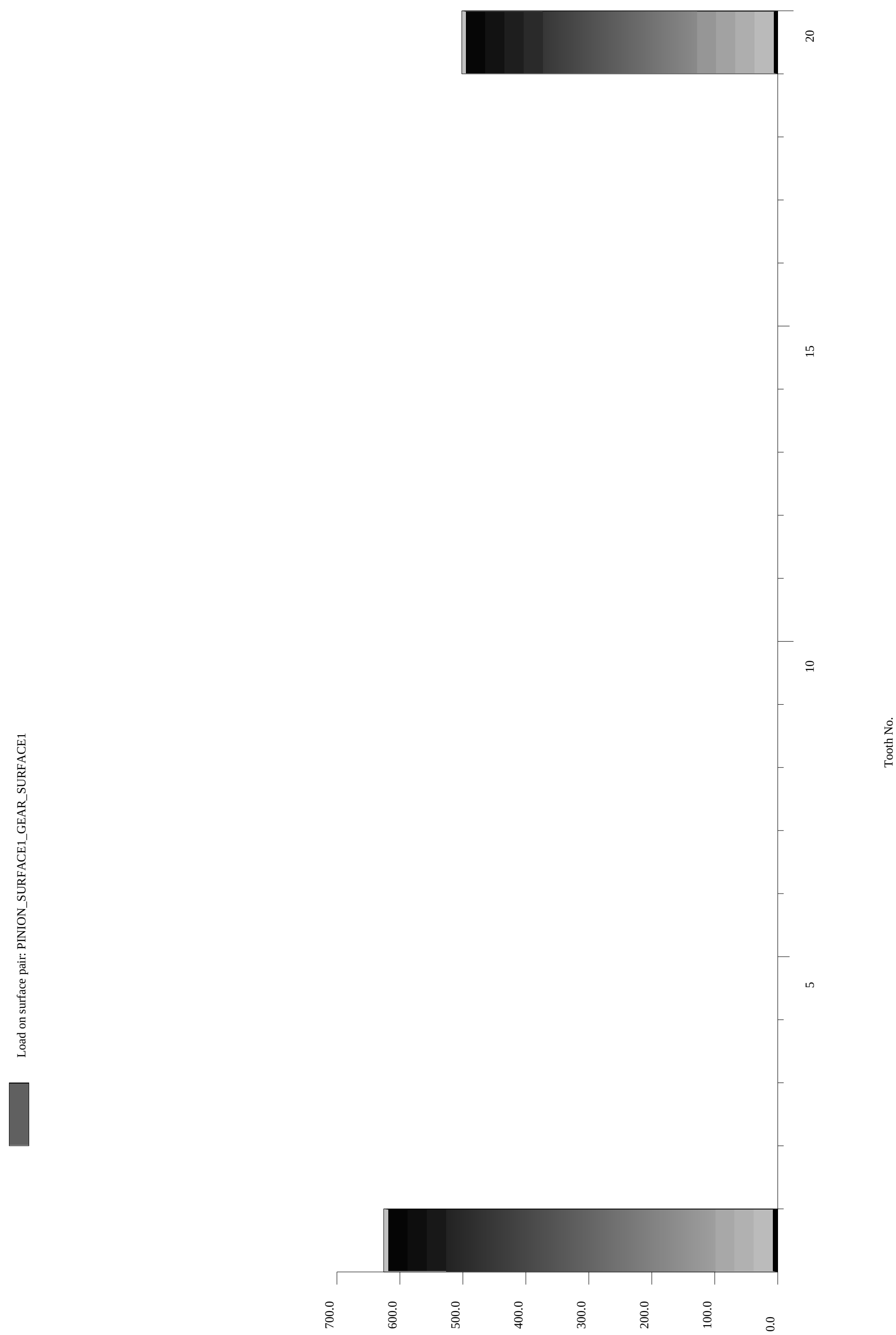


Figure 9.18: The tooth load histogram generated by the TOOTHLDHIST menu.

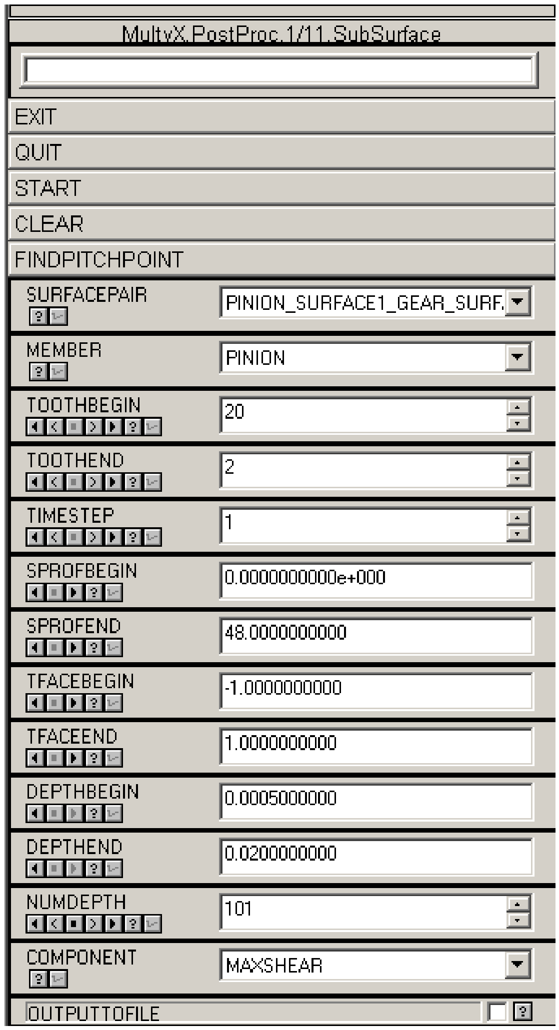


Figure 9.19: The SUBSURFACE menu.

principal normal stress s_3), MAXSHEAR (the maximum shear stress τ_{max}) and VONMISES (the Von Mises' octahedral shear stress s_{vm}).

Figure 9.20 shows an example of a graph of sub-surface stress vs. depth.

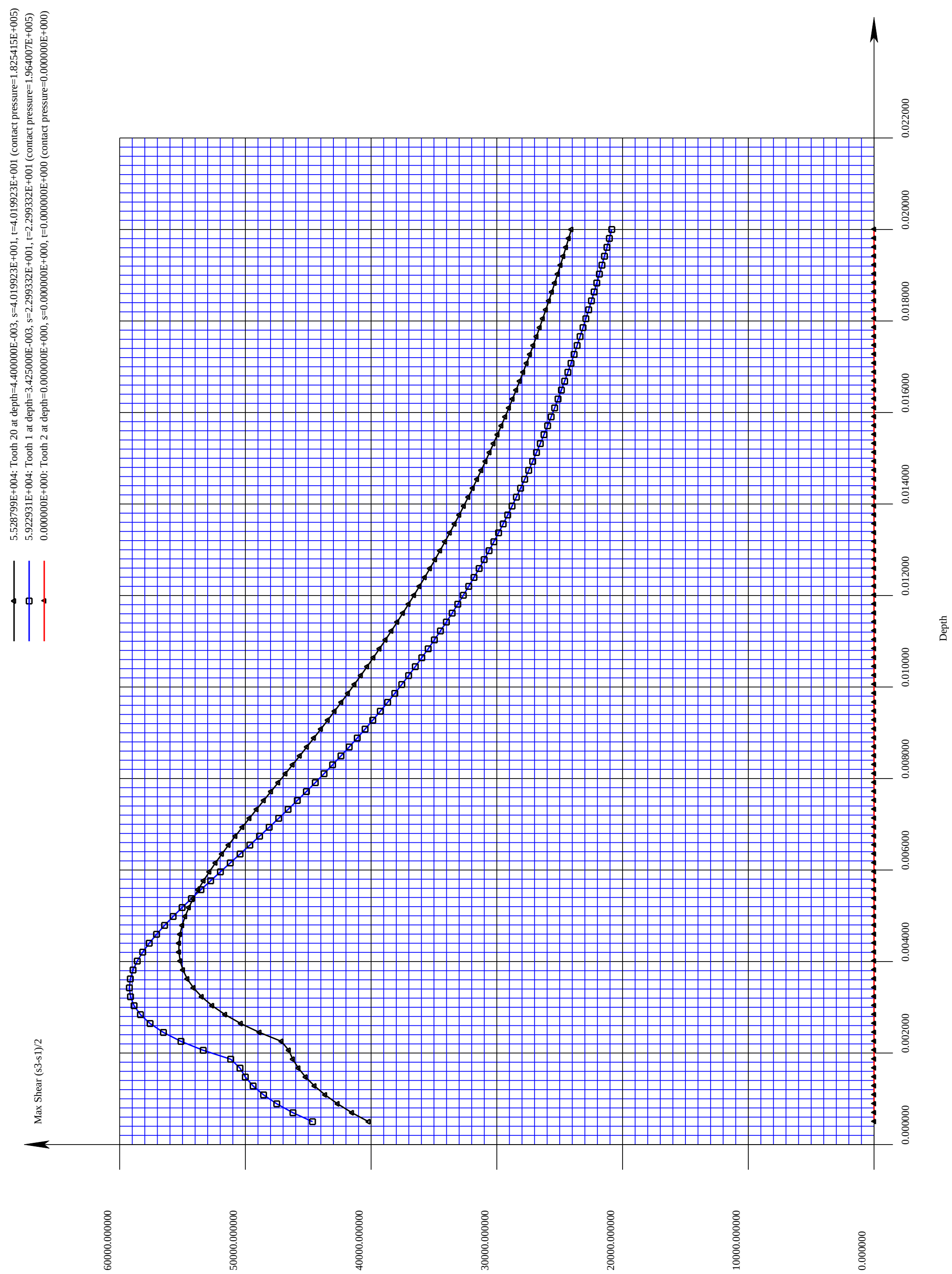


Figure 9.20: The sub-surface shear graph generated by the SUBSURFACE menu.

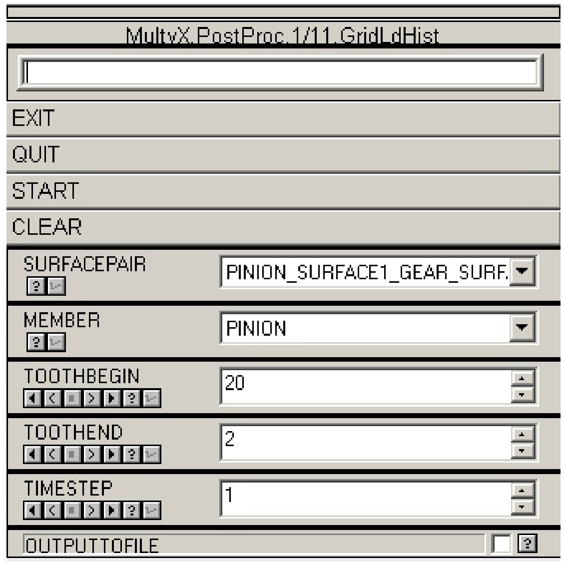


Figure 9.21: The GRIDLDHIST menu.

9.9 The GRIDLDHIST command

The GRIDLDHIST command in the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.21. 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 9.22 shows an example of a grid load histogram.

9.10 The GRIDPRHIST command

The GRIDPRHIST command in the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.23. This menu is used 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 9.24 shows an example of a grid pressure histogram. Figures 9.25 through 9.27 show the application of the GRIDPRHIST menu in obtaining a correct contact grid.

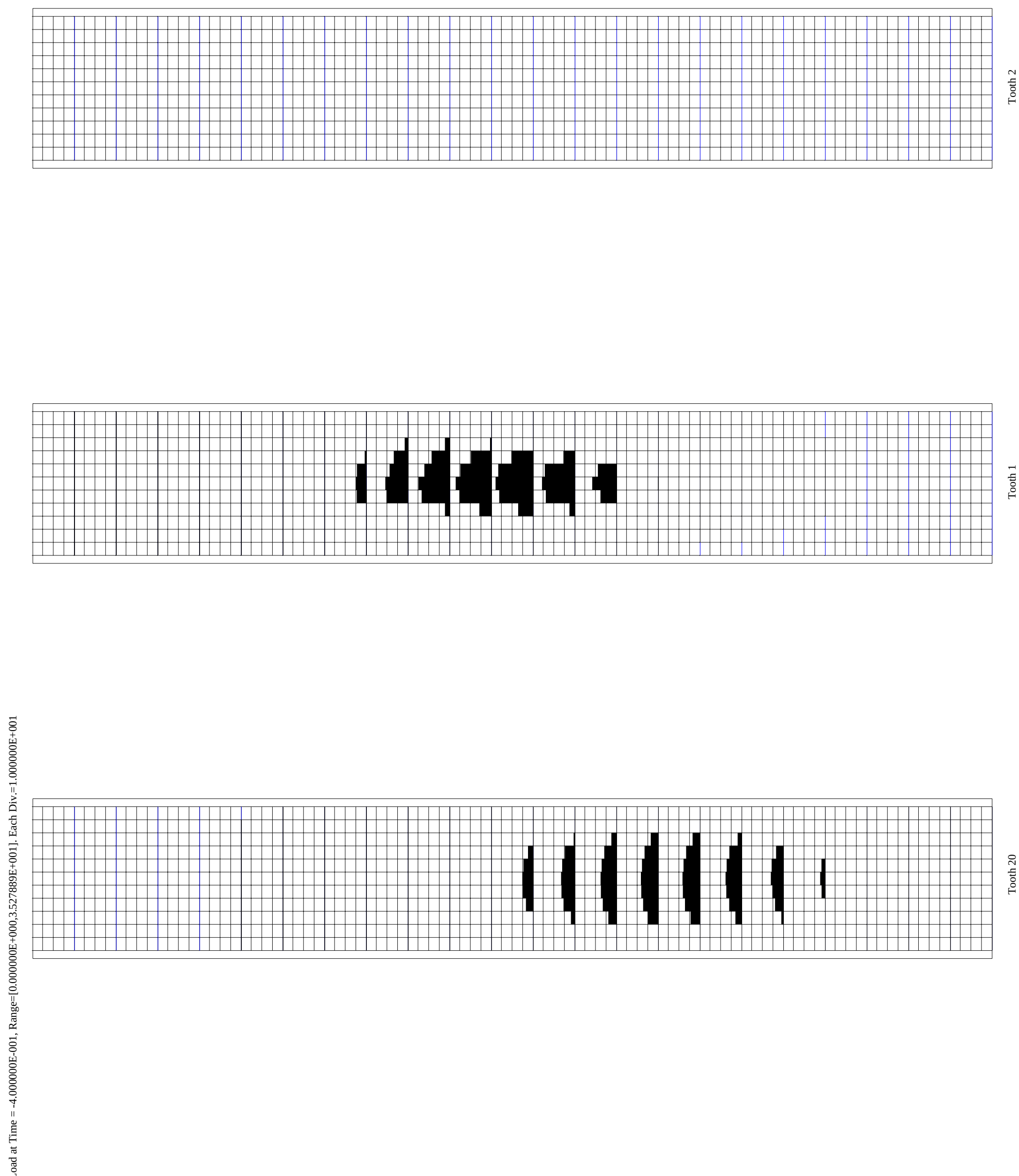


Figure 9.22: The grid load histogram generated by the GRIDLDHIST menu.

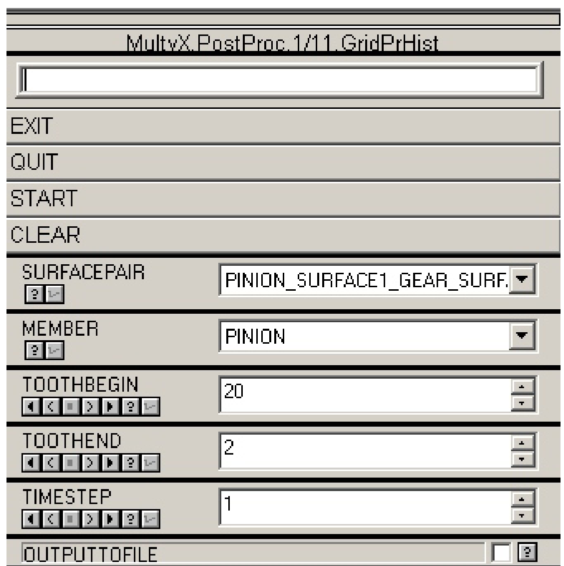


Figure 9.23: The GRIDPRHIST menu.

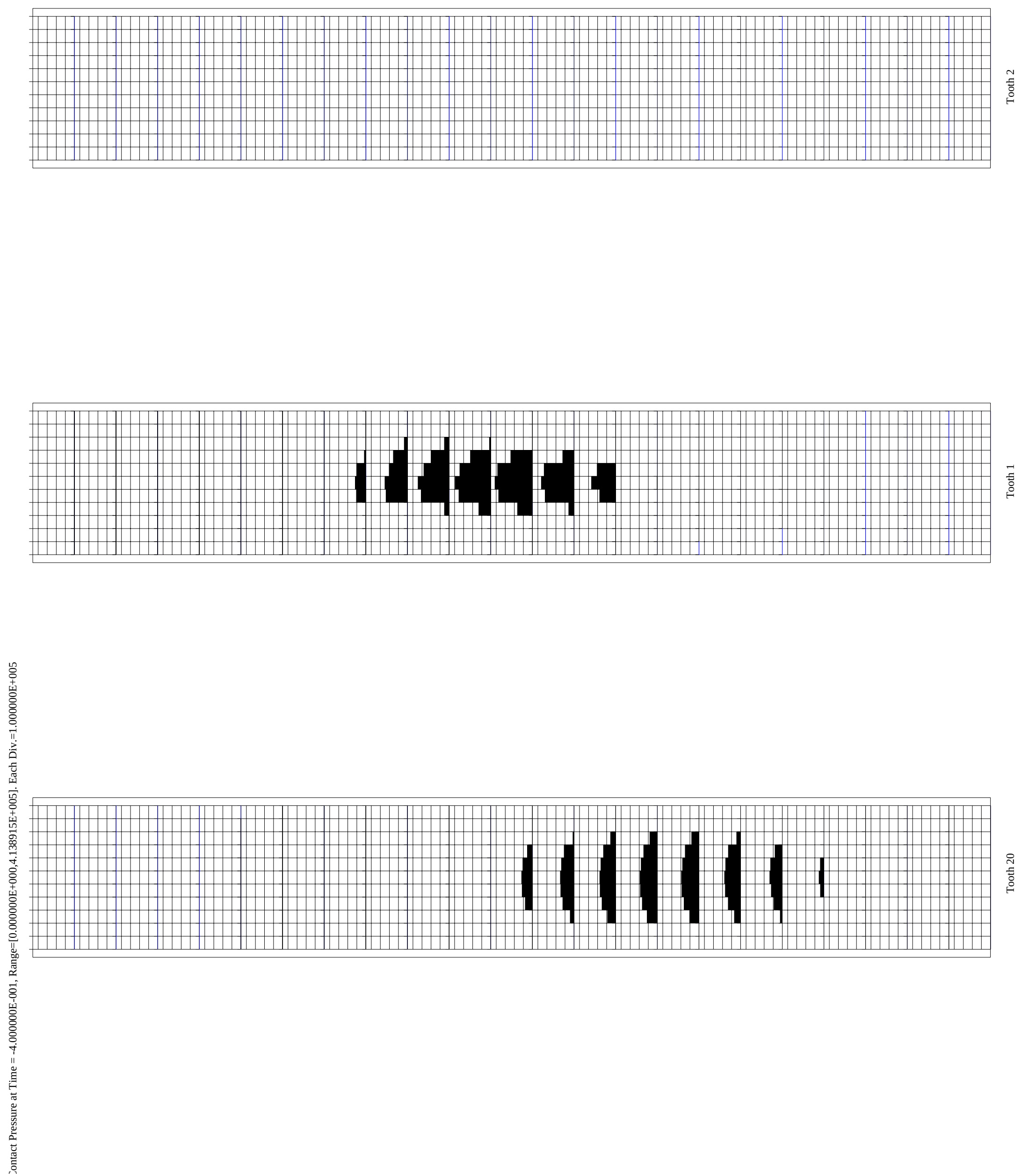


Figure 9.24: The grid pressure histogram generated by the GRIDPRHIST menu.

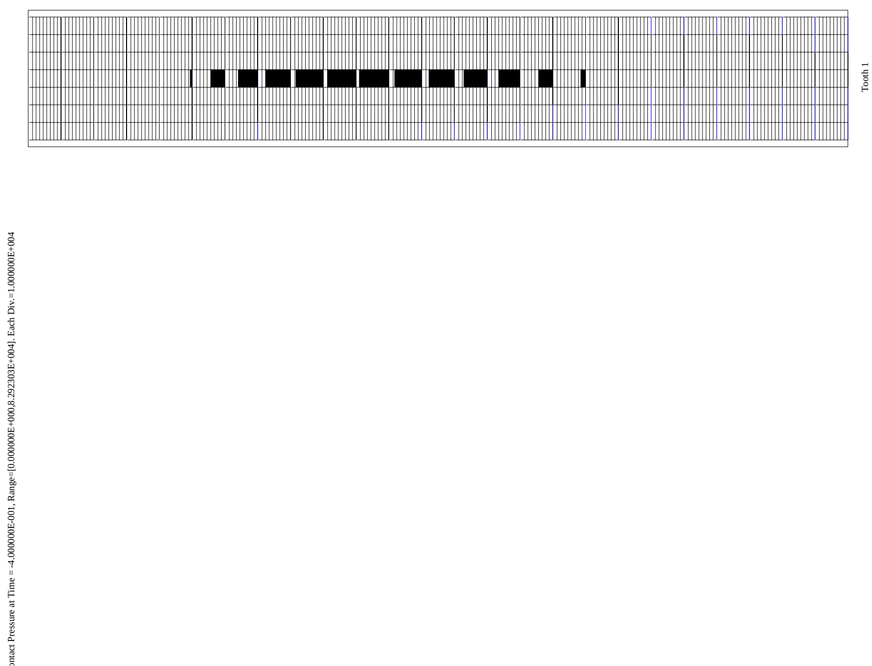


Figure 9.25: The grid pressure histogram generated by the GRIDPRHIST menu for DSPROF= 0.02 (too wide contact grid).

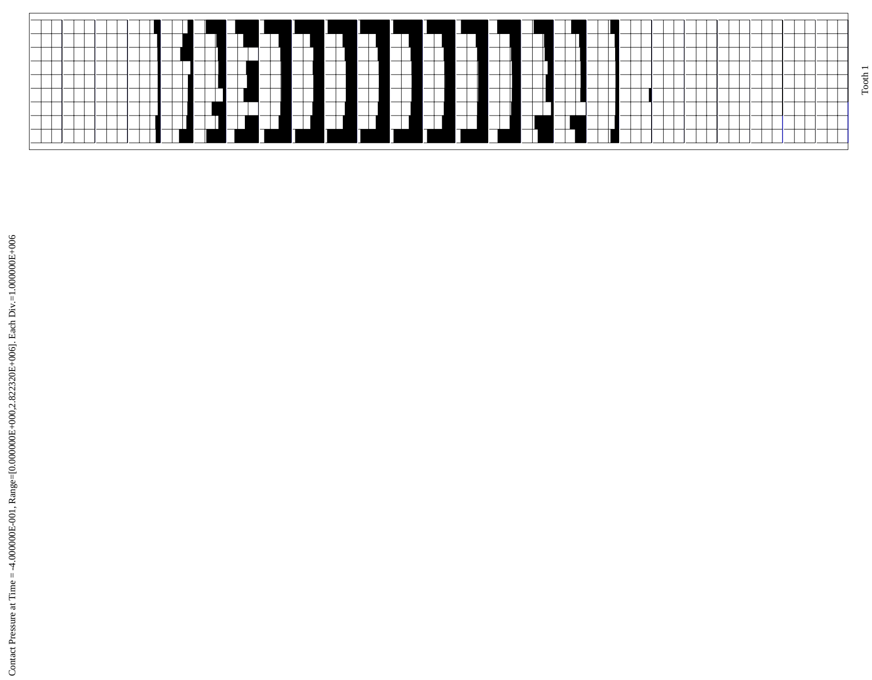


Figure 9.26: The grid pressure histogram generated by the GRIDPRHIST menu for DSPROF= 0.0001 (too narrow contact grid).

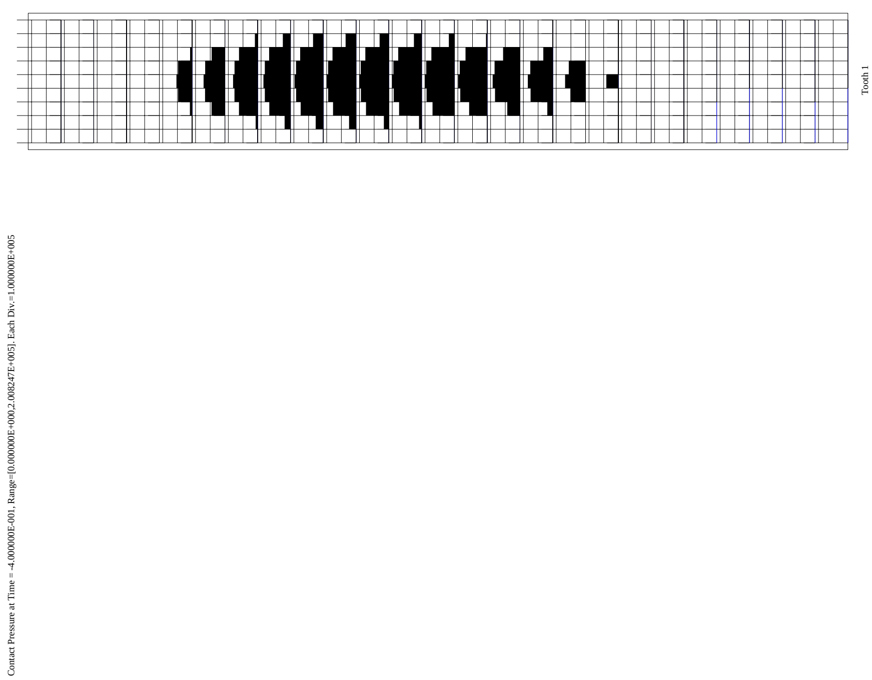


Figure 9.27: The grid pressure histogram generated by the GRIDPRHIST menu for DSPROF= 0.0015 (correct contact grid).

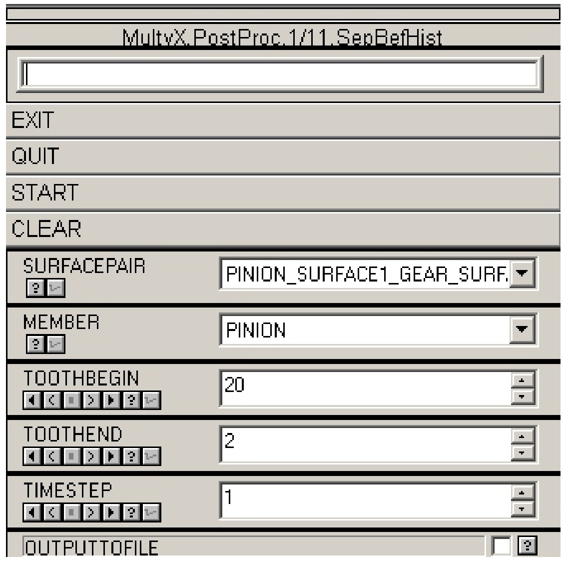


Figure 9.28: The SEPBEFHIST menu.

9.11 The SEPBEFHIST command

The SEPBEFHIST command in the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.28. This menu is used to generate a histogram of the distribution of normal separation over individual contact grid cells, in the unloaded and undeformed state.

Figure 9.29 shows an example of a histogram of separation in the unloaded state. Negative separation values are possible in this histogram which means there may be interference between the contacting surfaces. In such cases the program will still go ahead and compute the contacting loads.

9.12 The SEPAFTHIST command

The SEPAFTHIST command in the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.30. This menu is used to generate a histogram of the distribution of normal separation over individual contact grid cells, in the loaded and deformed state.

Figure 9.31 shows an example of a histogram of separation in the loaded state. These separation values must be either zero or positive.

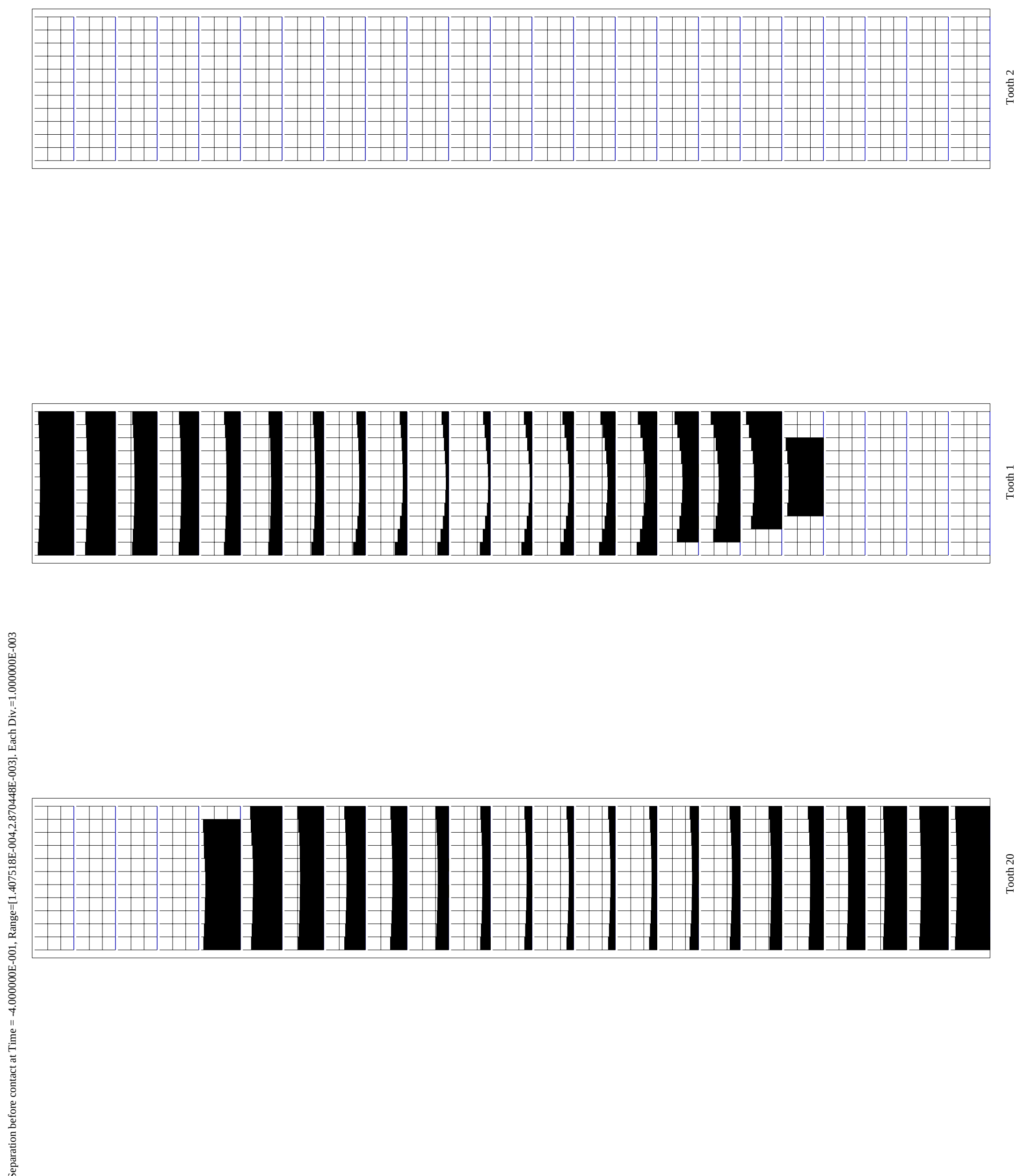


Figure 9.29: The histogram of grid separation before contact, generated by the SEPBEFHIST menu.

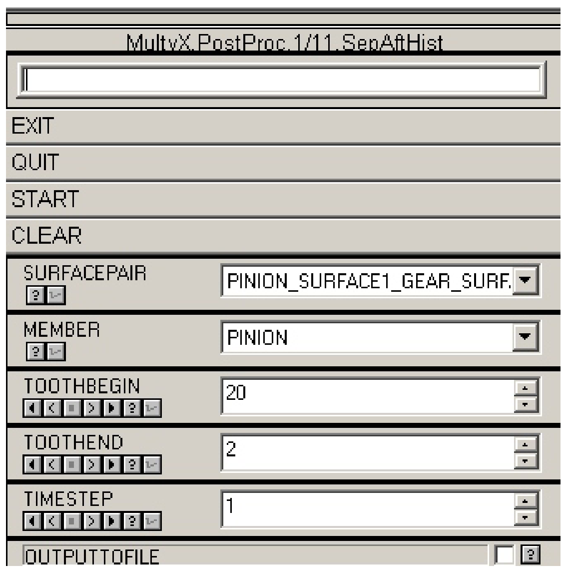


Figure 9.30: The SEPAFTHIST menu.

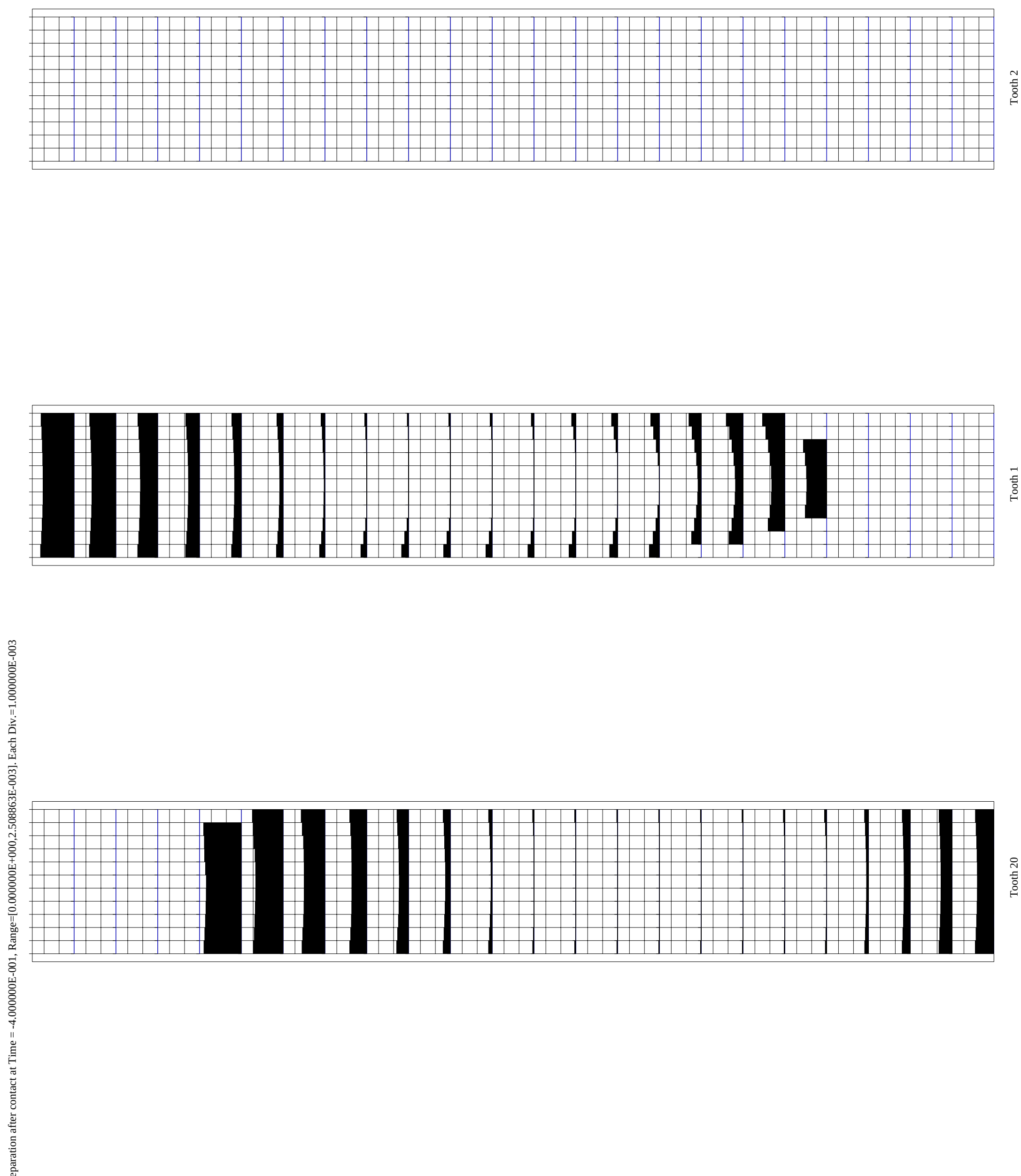


Figure 9.31: The histogram of grid separation after contact, generated by the SEPAFTHIST menu.

9.13 The SEARCHSTRESS command

The SEARCHSTRESS command of the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.32. 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 9.33 shows an example of stress as a function of time, Figure 9.34 shows stress as a function of profile position. Sharp oscillations can be seen in this graph in the vicinity of the concentrated contact loads. Finite element analysis results underneath a concentrated load may not always be reliable. Hence we feel that these oscillations may not be physically real as they are present near the concentrated loads. Figure 9.35 shows a graph of stress vs. face.

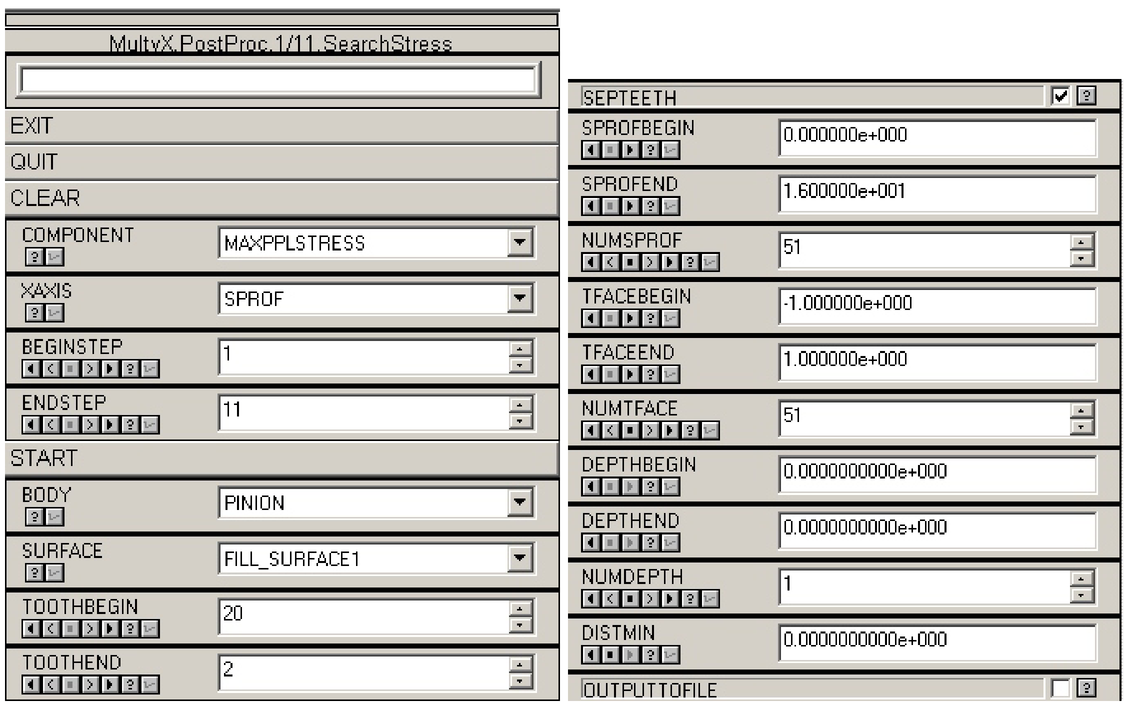


Figure 9.32: The SEARCHSTRESS menu

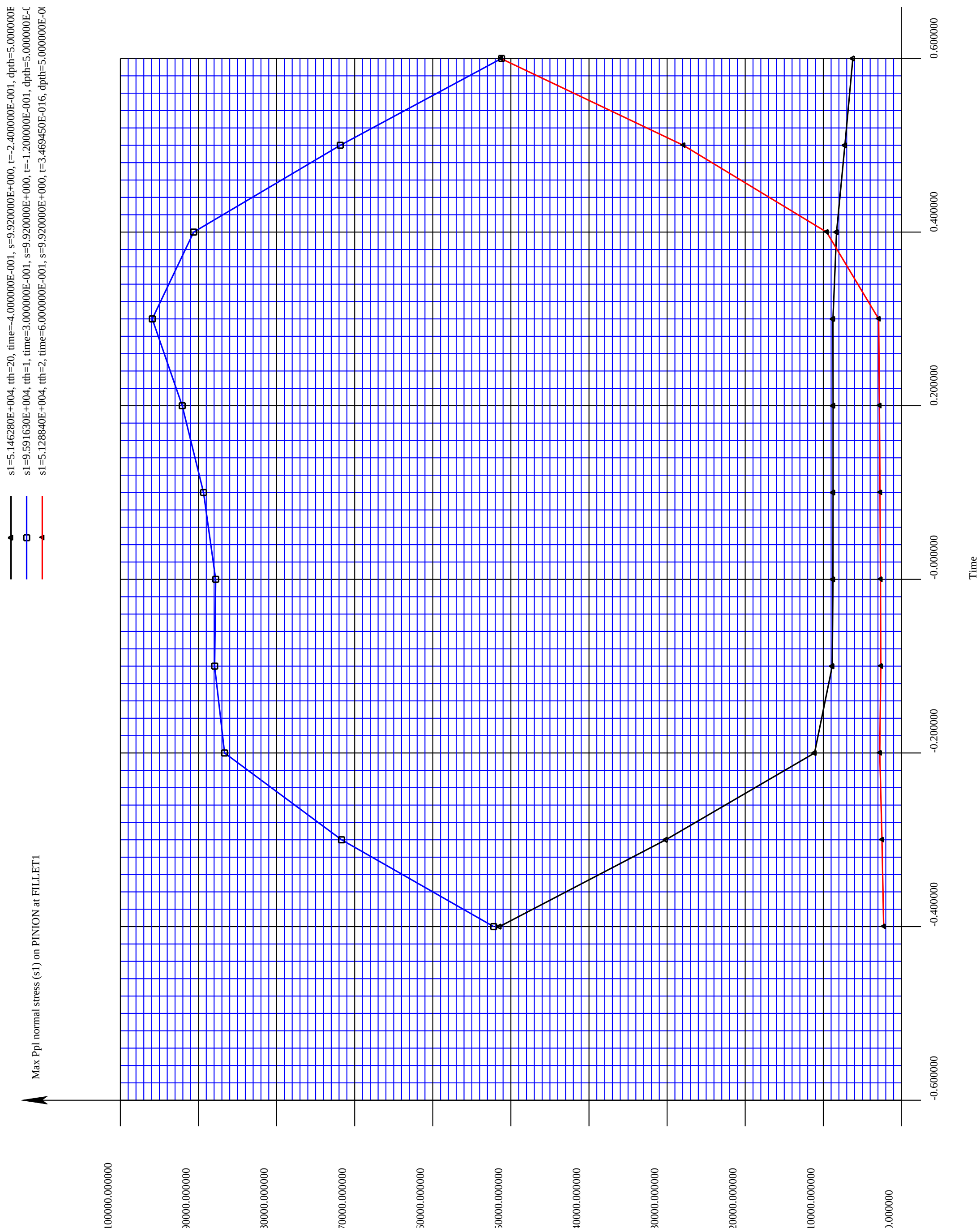


Figure 9.33: The graph of root stress vs. time, generated by the SEARCHSTRESS menu.

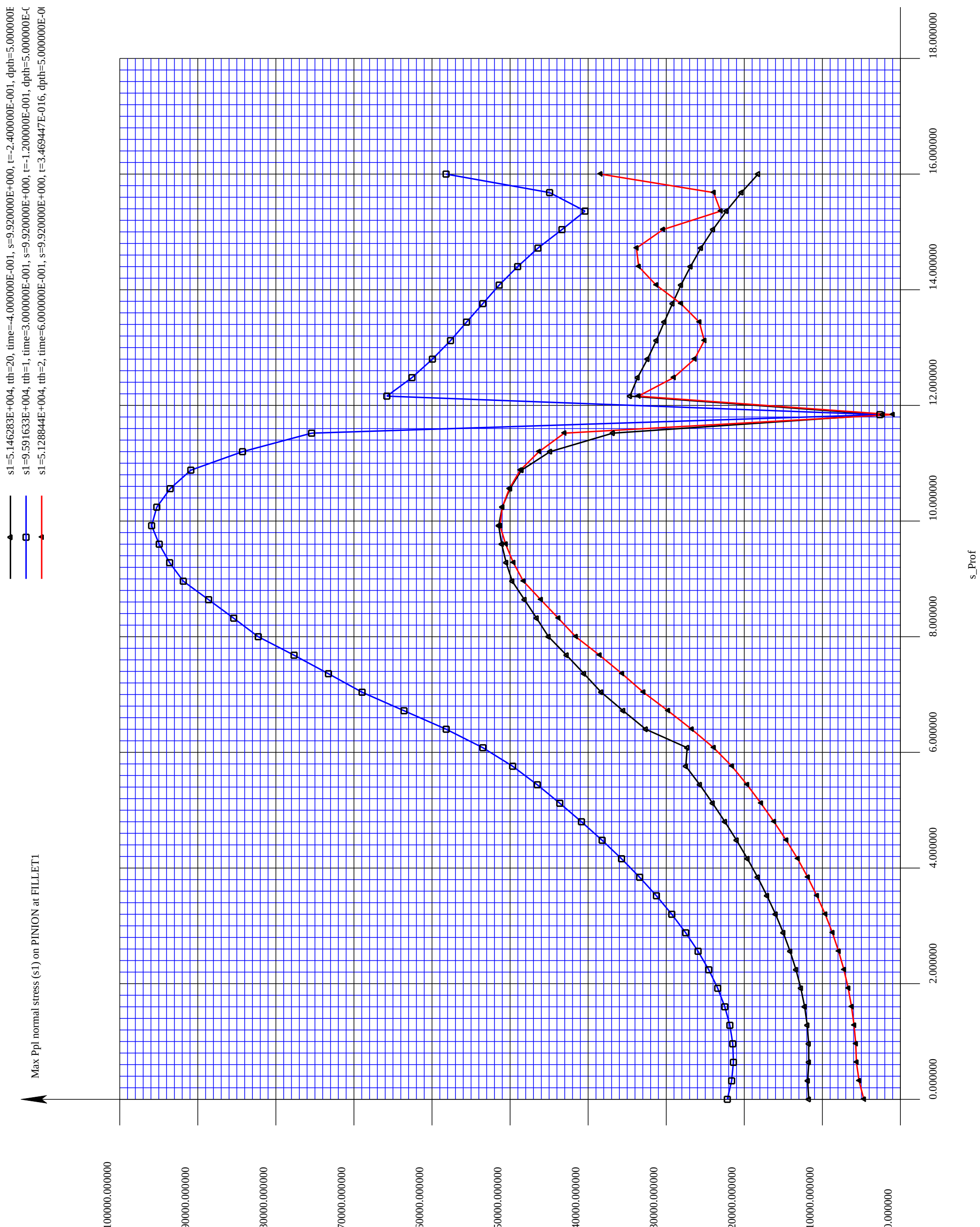


Figure 9.34: The graph of root stress vs. profile, generated by the SEARCHSTRESS menu.

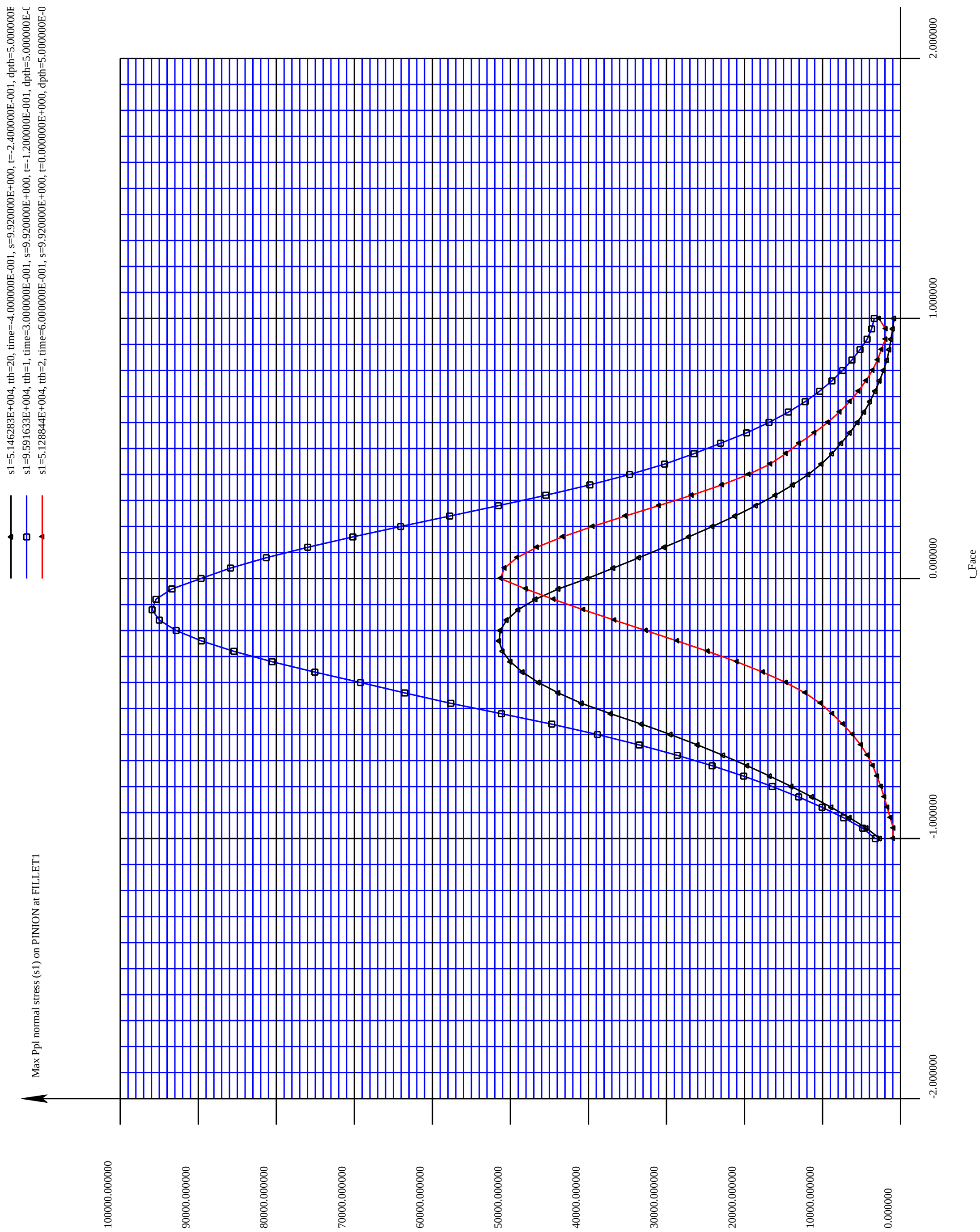


Figure 9.35: The graph of root stress vs. face, generated by the SEARCHSTRESS menu.

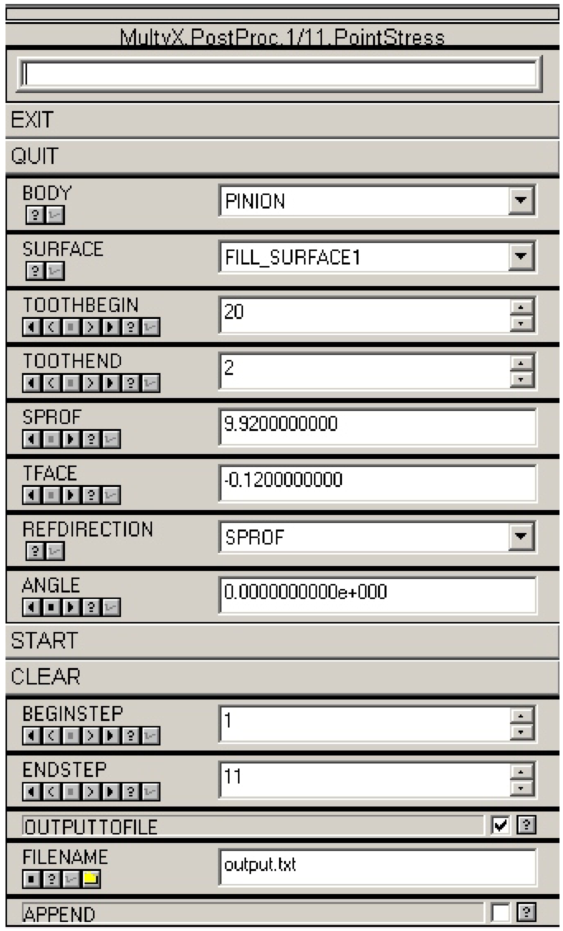


Figure 9.36: The POINTSTRESS menu.

9.14 The POINTSTRESS command

The POINTSTRESS command of the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.36. 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 9.37 shows an example of the graph generated by this menu.

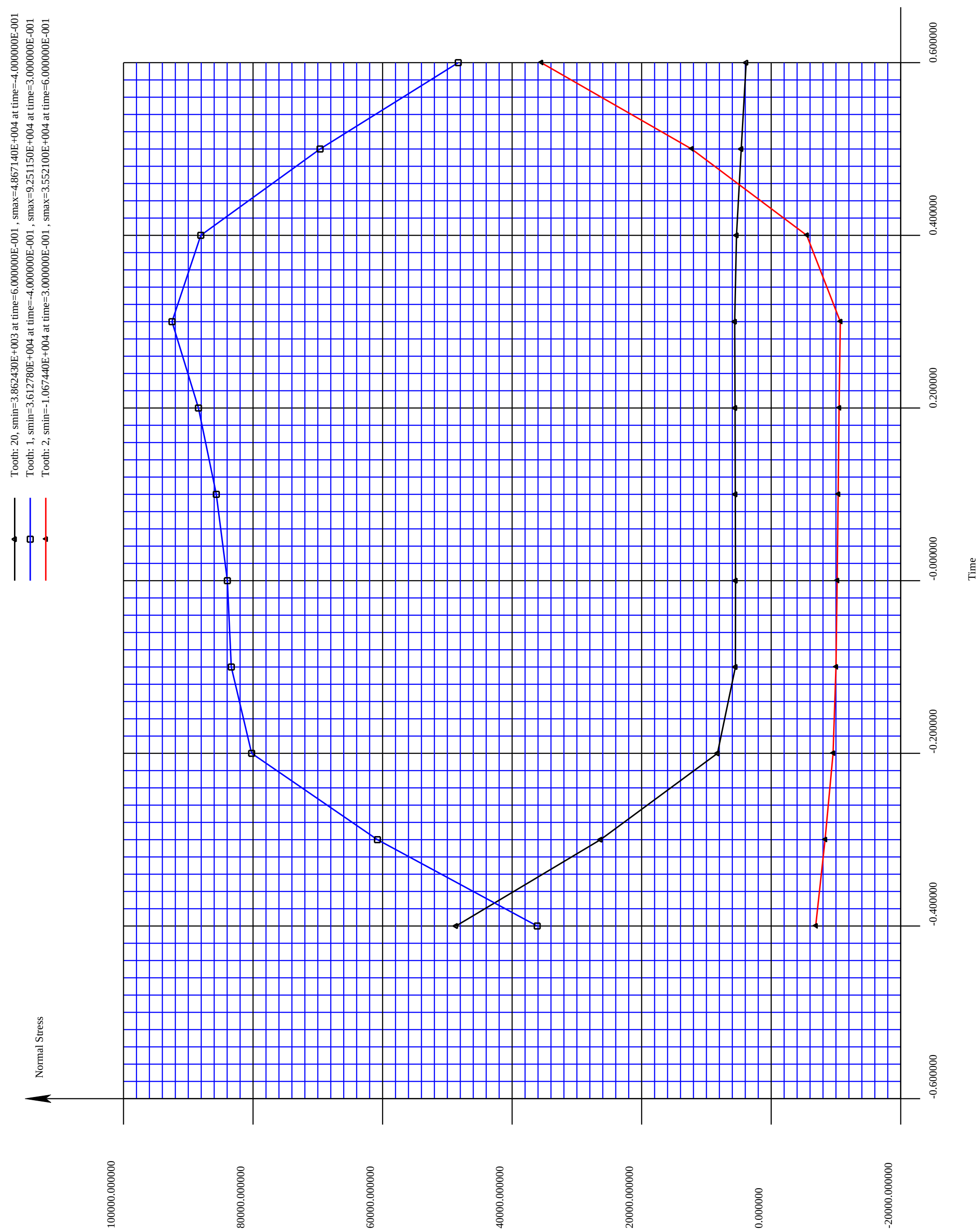


Figure 9.37: The graph of root stress vs. face, generated by the POINTSTRESS menu.

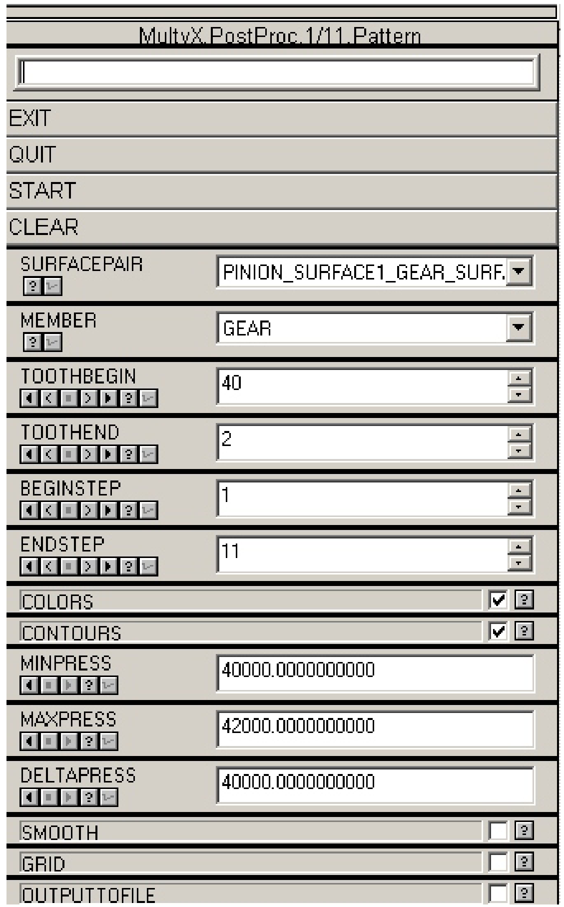


Figure 9.38: The PATTERN menu.

9.15 The PATTERN command

The PATTERN command of the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.38. This menu is used to create a drawing of the contact pattern on a tooth.

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 9.39 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.

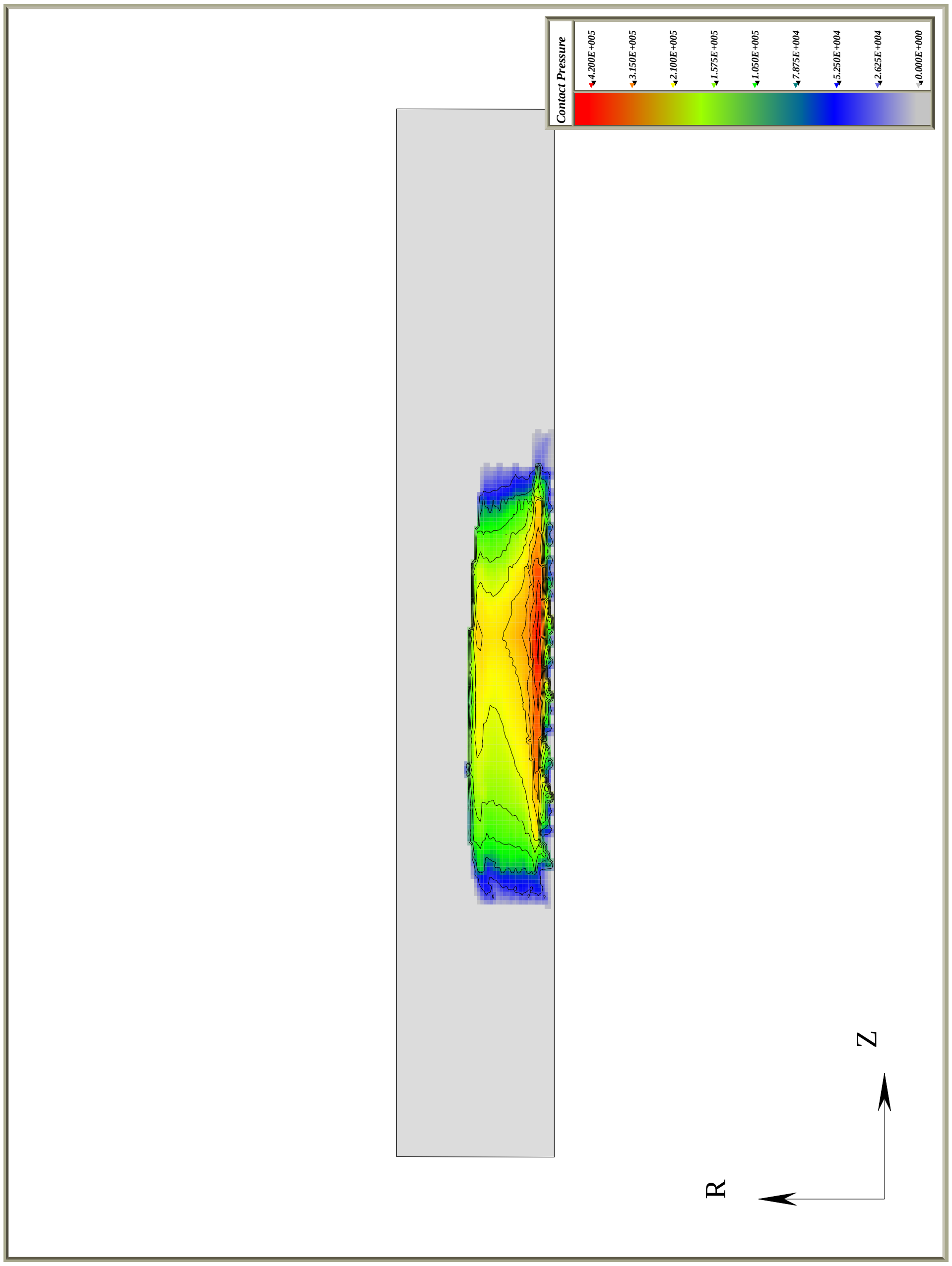


Figure 9.39: The contact pattern generated by the PATTERN menu.

9.16 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 9.3) generates an equilibrim 'audit' of all the forces and moments acting on each body. Figure 9.40 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 of 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=-0.4
Body no.2:PINIONSHAFT (Origin at:[0,-1,0])
==================================================================

Contact forces:
    Exerted by:PINION
    Total  :f [-974.3496506,-360.2120942,-1.704639161e-012],
            mo[310.218819,-837.6780654,1000]
            m [310.218819,-837.6780654,25.65034942]
Total contact force=f [-974.3496506,-360.2120942,-1.704639161e-012]
                    mo[310.218819,-837.6780654,1000]
                    m [310.218819,-837.6780654,25.65034942]


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 [-974.3496506,-360.2120942,-1.704639161e-012],
                                           mo[310.218819,-837.6780654,1000]
                                           m [310.218819,-837.6780654,25.65034942]
Total bearing  force                       :f [0,0,0],
                                           mo[0,0,0]
                                           m [0,0,0]
Total reaction force                       :f [974.3496506,360.2120942,1.704639161e-012],
                                           mo[-310.218819,837.6780654,-1000]
                                           m [-310.218819,837.6780654,-25.65034942]
==================================================================================
Residual force (error)                     :f [-5.684341886e-013,0,0],
                                           mo[-5.684341886e-014,1.136868377e-013,-1.813305062e-010]
                                           m [-5.684341886e-014,1.136868377e-013,-1.818989404e-010]
```

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.

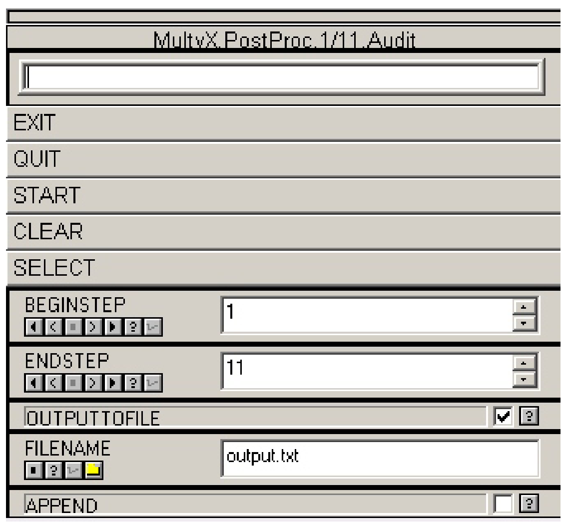


Figure 9.40: The AUDIT menu.

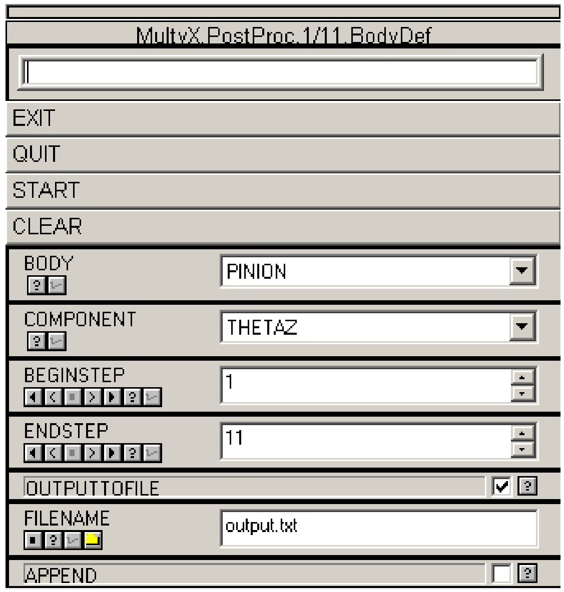


Figure 9.41: The BODYDEFLECTION menu.

9.17 The BODYDEFLECTION command

The BODYDEFLECTION command of the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.41. This menu is used to generate a graph (Figure 9.42) 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.

9.18 The BODYREACTION command

The BODYREACTION command of the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.43. This menu is used to generate a graph (Figure 9.44) 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.

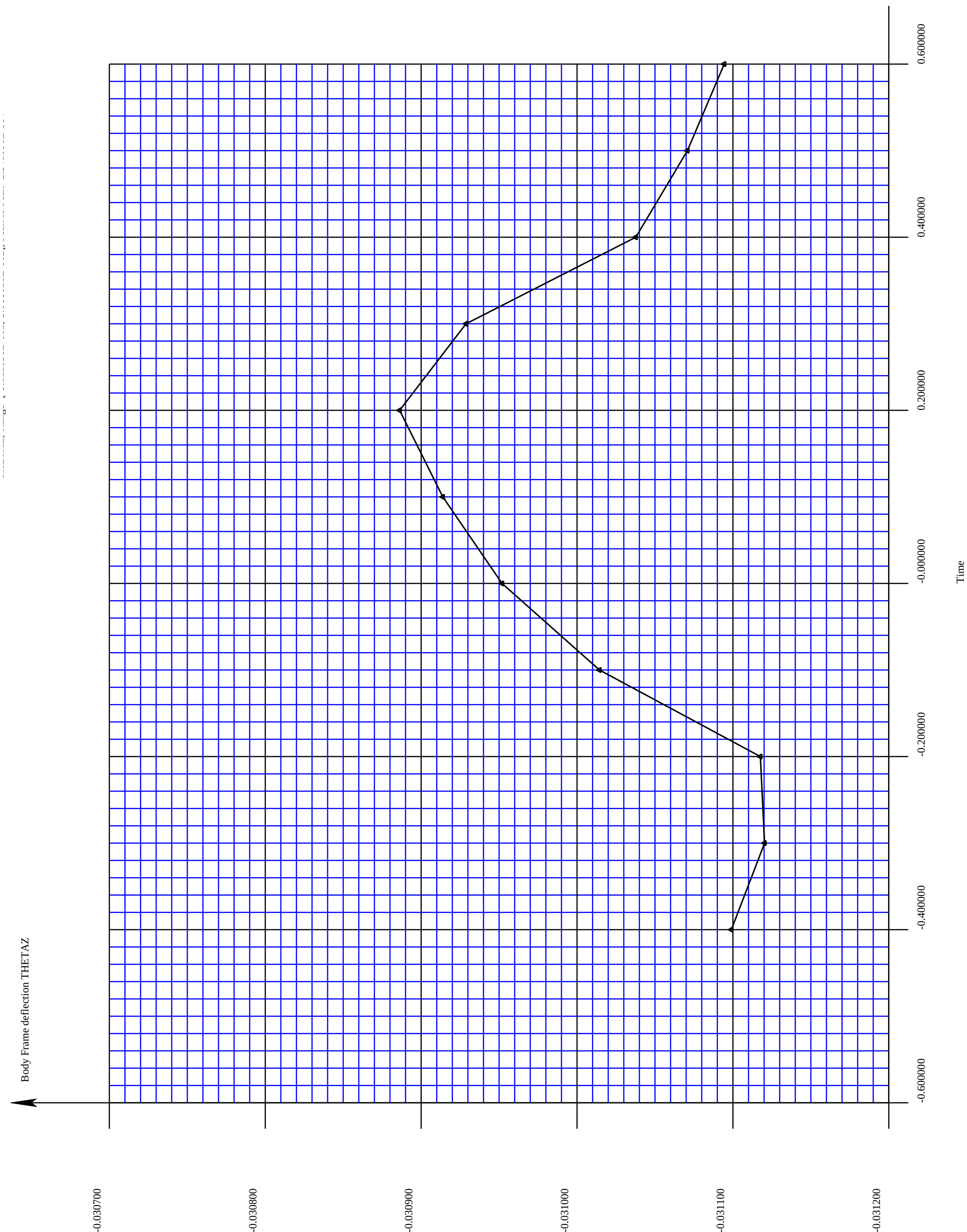


Figure 9.42: The graph generated by the BODYDEFLECTION menu.

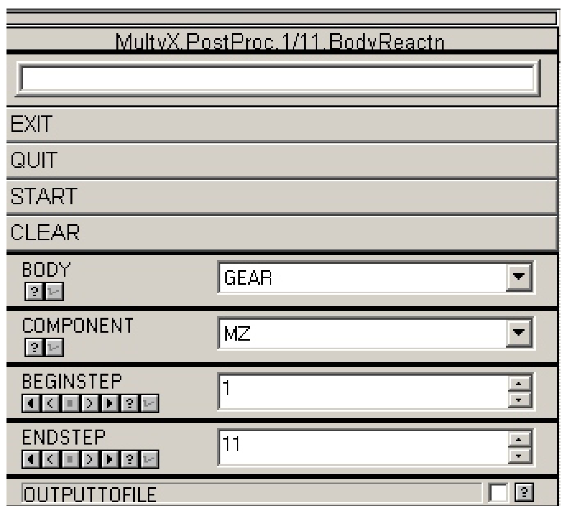


Figure 9.43: The BODYREACTION menu.

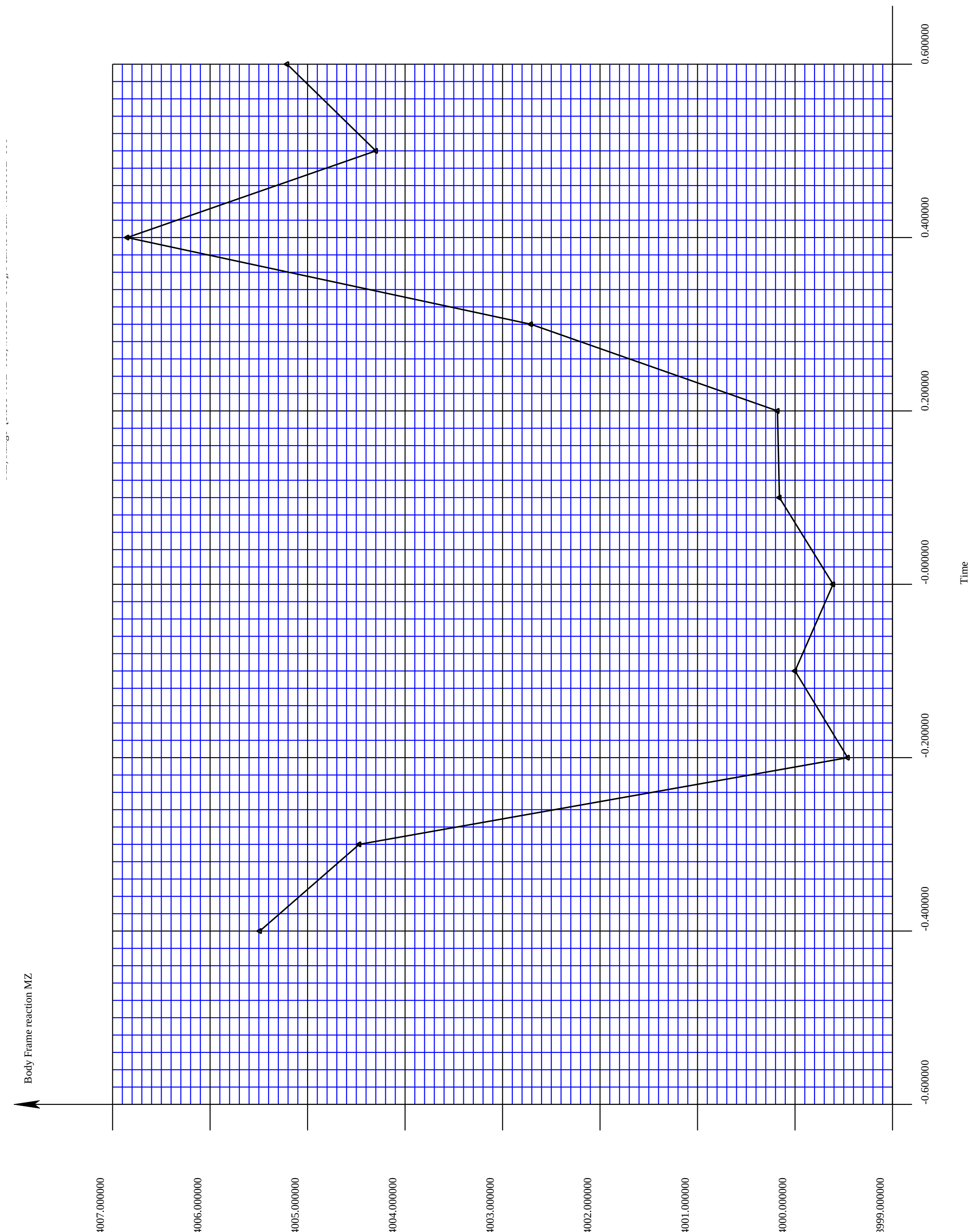


Figure 9.44: The graph generated by the BODYREACTION menu.

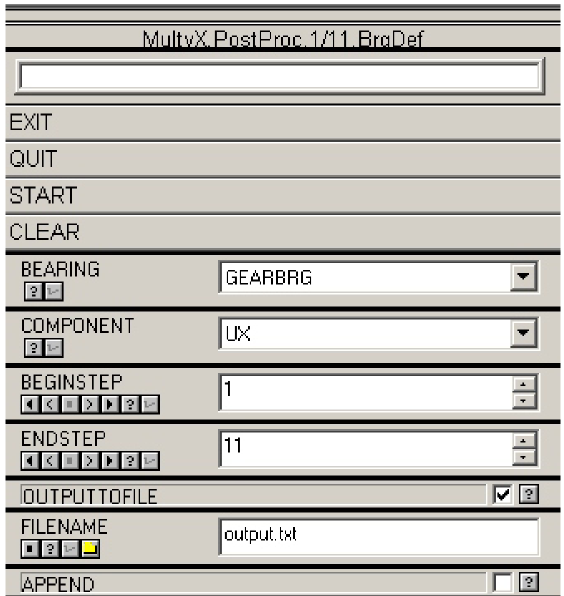


Figure 9.45: The BRGDEFORMN menu.

9.19 The BRGDEFORMN command

The BRGDEFORMN command of the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.45. This menu is used to generate a graph (Figure 9.46) of a component of the bearing deformation 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 of bearing race 1 with respect to bearing race 2. The components are measured in bearing race 2. In *Helical3D*, bearing race 2 for the pinion and gear bearings are attached to the fixed body (ground). So the components are the same as they would appear when measured in the fixed frame.

The rotation components are displayed in radians.

9.20 The BRGREACTION command

The BRGREACTION command of the post-processing menu (Figure 9.3) leads to the menu shown in Figure 9.47. This menu is used to generate a graph (Figure 9.48) of a component of the bearing 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 the forces and moments exerted by race 1 on race 2. The components are calculated in the race 2 reference frame. The moments are about the origin of race 2. In *Helical3D*, race 2 for the pinion bearing, as well as for the gear bearing is attached to the fixed body (ground). So the components are the same as they would appear when measured in the fixed reference frame.

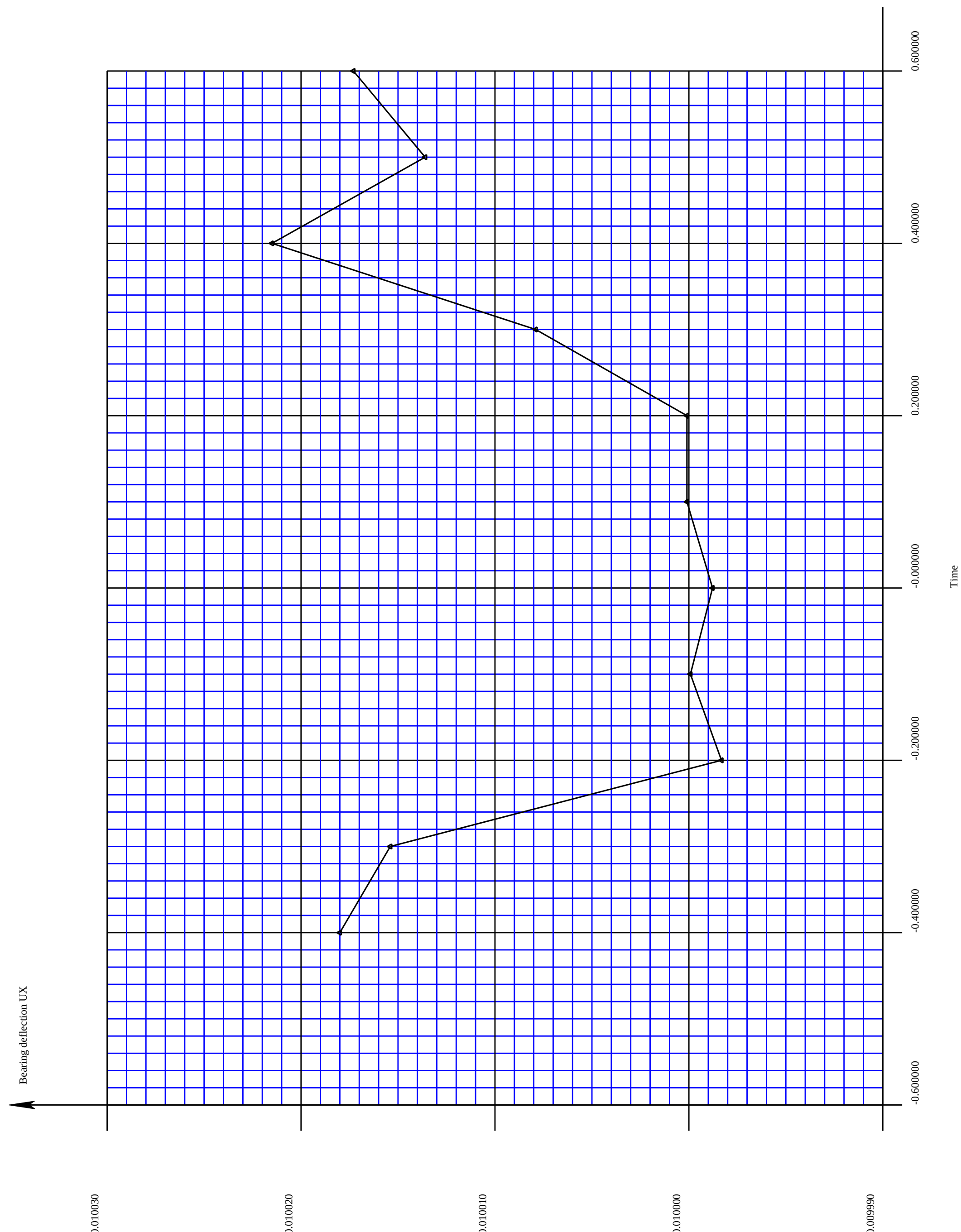


Figure 9.46: The graph generated by the BRGDEFORMN menu.

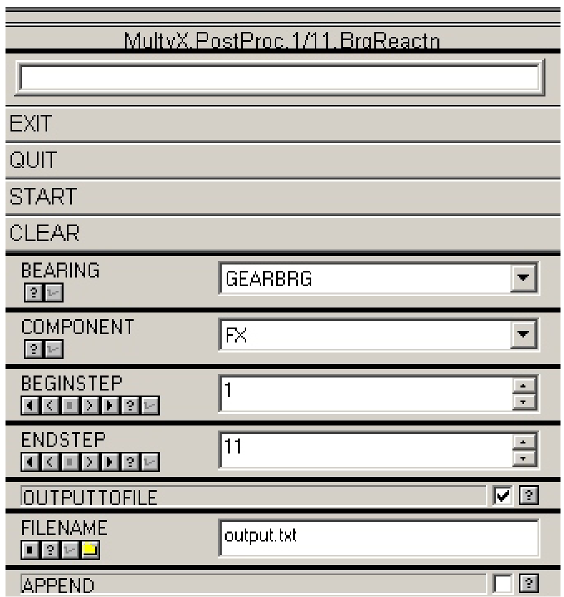


Figure 9.47: The BRGREACTION menu.

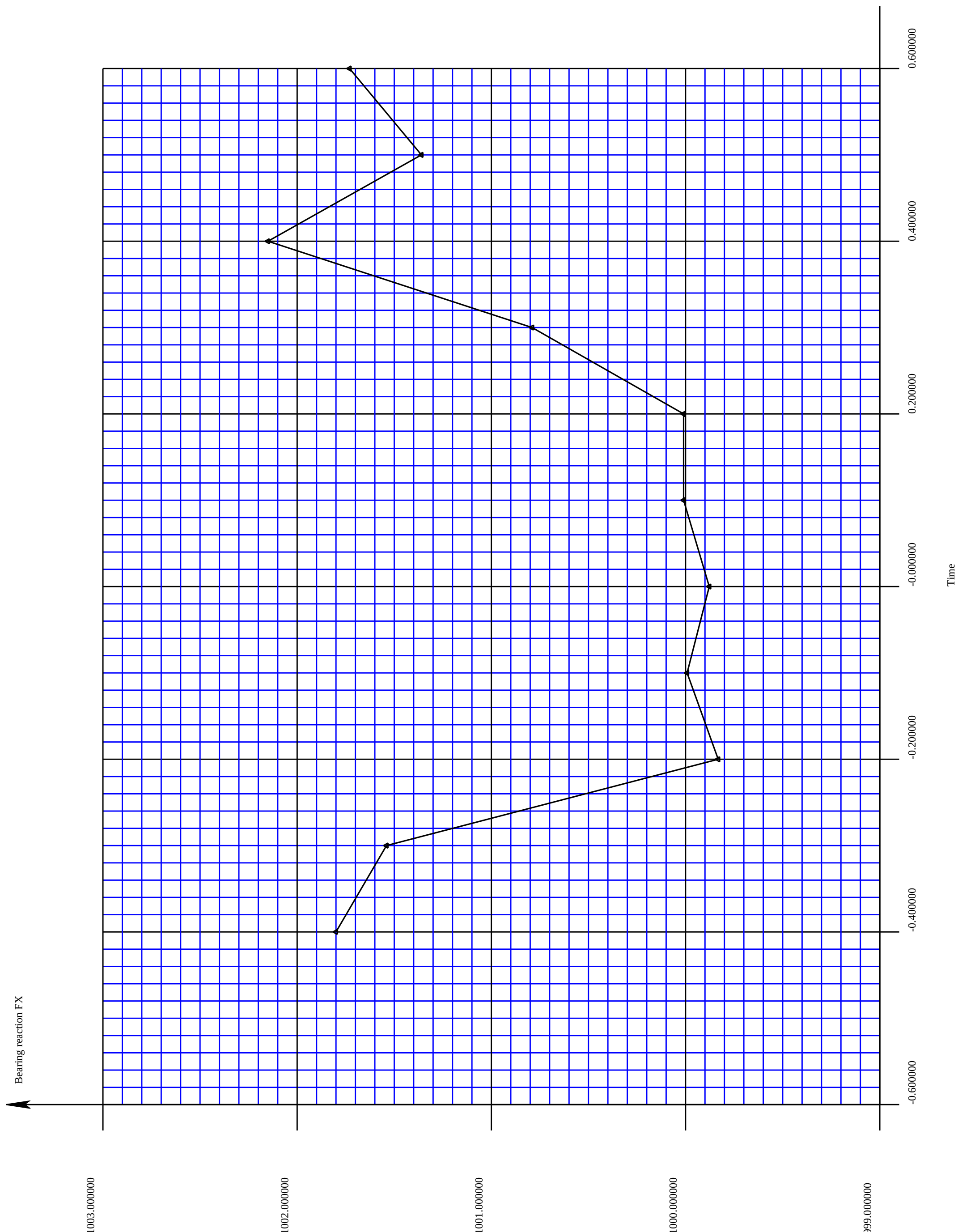


Figure 9.48: The graph generated by the BRGREACTION menu.

Chapter 10

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.

10.1 Generating an Iglass file for preprocessing

The GENIGLASSFILE command in Figure 9.1 will lead to a menu shown in Figure 10.1 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 10.1 the user can select the bodies to be displayed in the Iglass graphical window. Click on the START button in Figure 10.1 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 Helical gear pair is shown in Figure 10.2. 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.

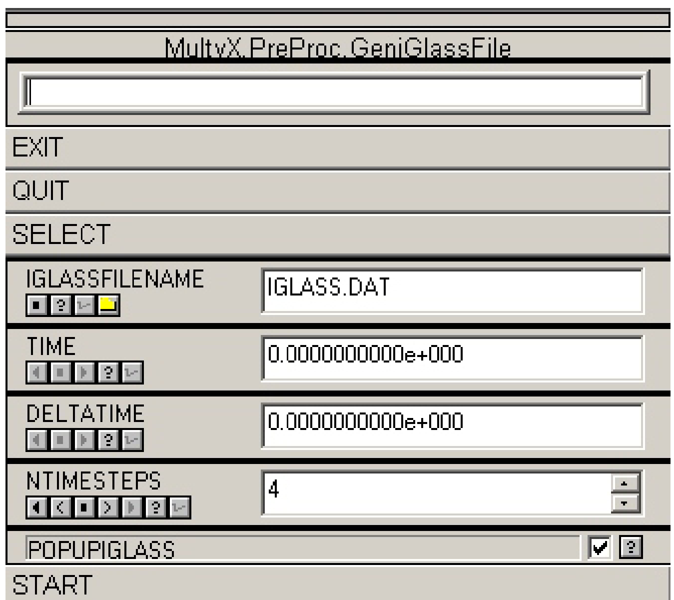


Figure 10.1: The generate Iglass file menu

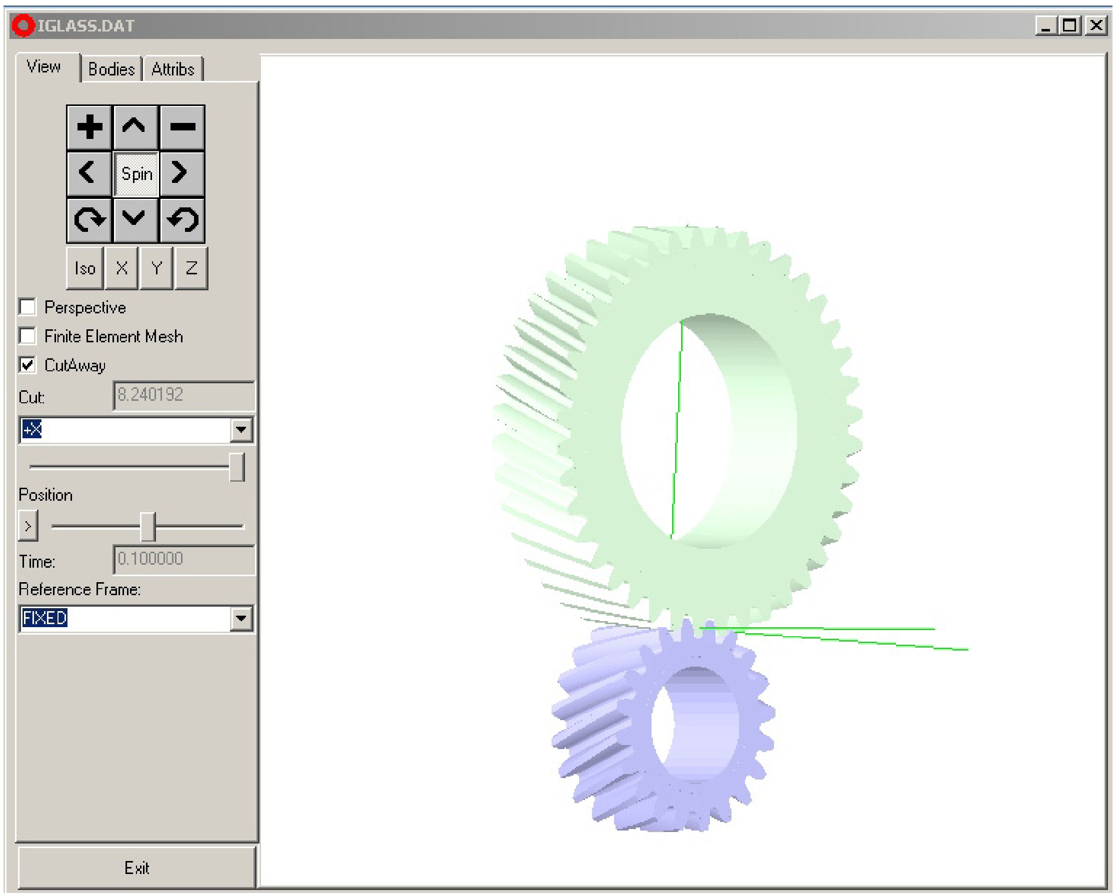


Figure 10.2: An example of an Iglass preprocessing window.

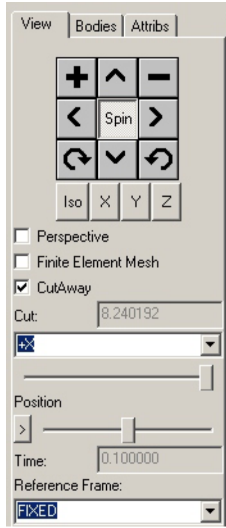


Figure 10.3: Iglass preprocessing view menu

10.2 View menu

The View menu is shown in Figure 10.3. Table 10.1 shows the common tasks performed by some of the buttons displayed in this menu.

Apart from all the features shown in Table 10.1 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.

10.2.1 Finite element mesh

The finite element mesh model can be visualised if the 'Finite Element Mesh' item is selected. Figure 10.4 shows the finite element mesh model of the gear bodies in iglass preprocessing.

10.2.2 Cutting plane

Using the cutting plane switch shown in Figure 10.5 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.

10.2.3 Selecting the time step

User can visualise the model at a particular time step in iglass pre-processing using the 'Position' slider shown in Figure 10.6. 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 10.7.

Table 10.1: Common buttons in Iglass pre and postprocessing window

Button	Purpose
	Zoom In
	Zoom Out
	Move the model upwards (If Spin is turned OFF)
	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)
	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)
	View the model in an isometric view
	View the model in the $Y - Z$ plane
	View the model in the $X - Z$ plane
	View the model in the $X - Y$ plane

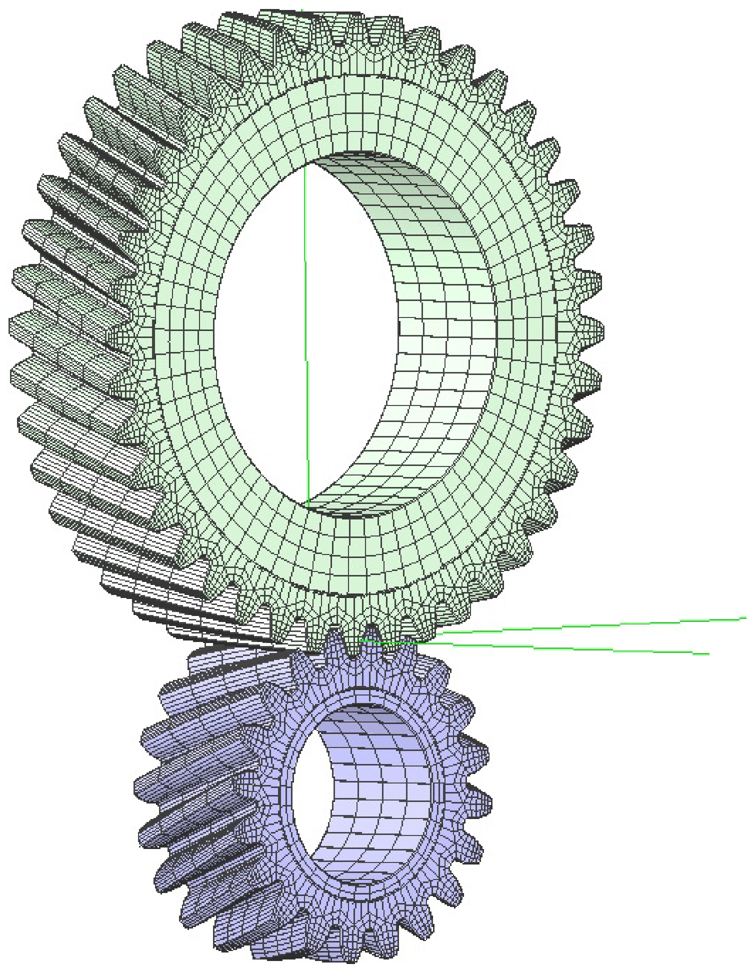


Figure 10.4: Finite element mesh model of the gear bodies

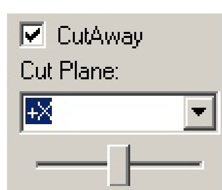


Figure 10.5: The cutting plane switch.

Figure 10.6: The position slider.

Figure 10.7: The time menu.

Figure 10.8: The reference frame switch.

10.2.4 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 10.8. 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.

10.3 The Bodies menu

The 'Bodies' menu is shown in Figure 10.9. 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.

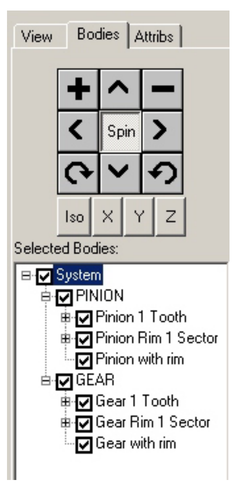


Figure 10.9: Iglass preprocessing Bodies menu

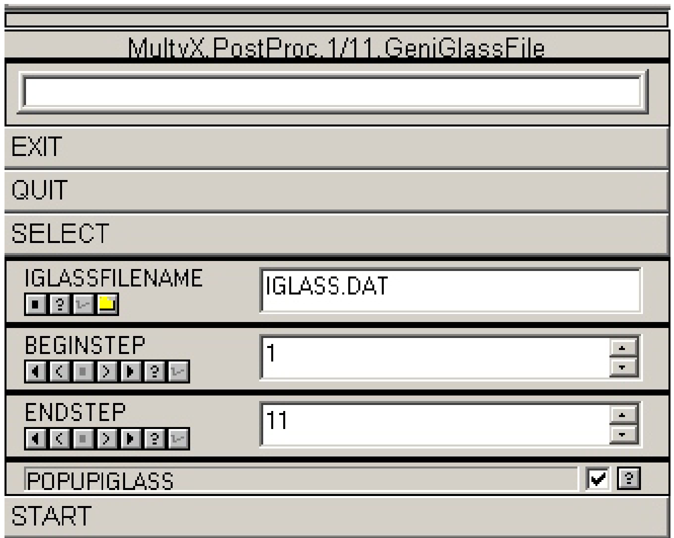


Figure 10.10: The generate iglass file menu for post processing.

10.4 Post processing using iglass

The GENIGLASSFILE command in Figure 9.3 leads to the generate iglass file menu shown in Figure 10.10 for post processing in iglass. BEGINSTEP and ENDSTEP menus shown in Figure 10.10 define the range for which you want to check for results. Note that these menus are independent of the GOTOPOSN menu shown in Figure 9.3.

An example of an iglass post processing window is shown in Figure 10.11.

10.5 Features specific to iglass post processing

The position switch shown in Figure 10.12 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 10.13 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 10.14 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 10.15. The magnification scale of the forces and the moments can be adjusted using the respective sliders.

The 'Attribs' menu is shown in Figure 10.16. The attribute menu shown in Figure 10.17 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.

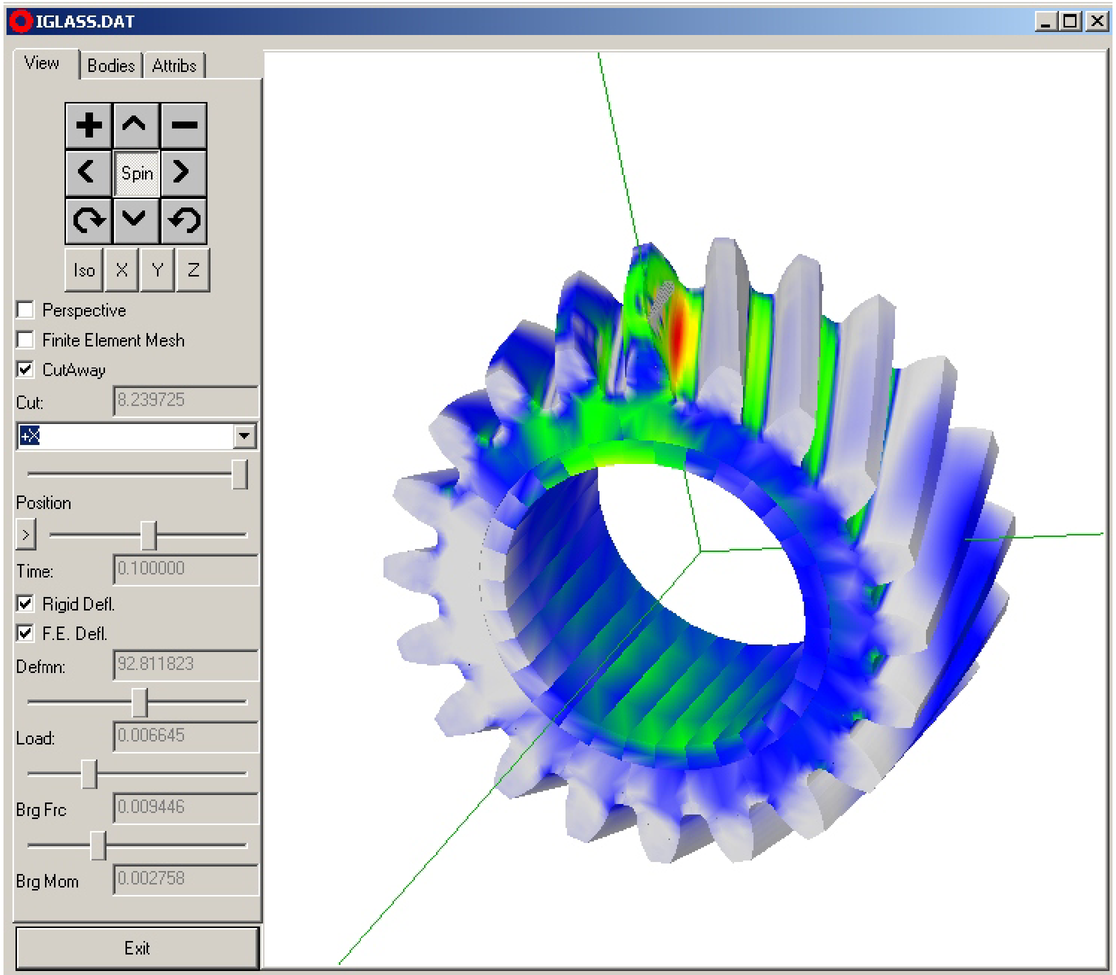


Figure 10.11: An example of an iglass post processing window.

Figure 10.12: The position slider.

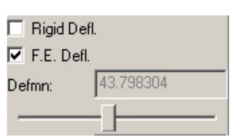


Figure 10.13: The deformation slider.

Figure 10.14: The load slider.

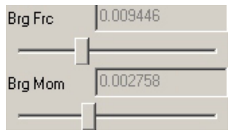


Figure 10.15: The bearing forces and moments sliders.

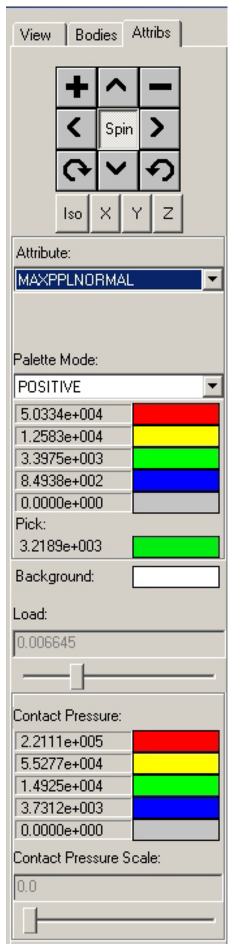


Figure 10.16: The iglass postprocessing attribute menu.

Figure 10.17: The attribute switch.

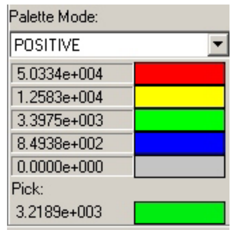


Figure 10.18: The palette switch.

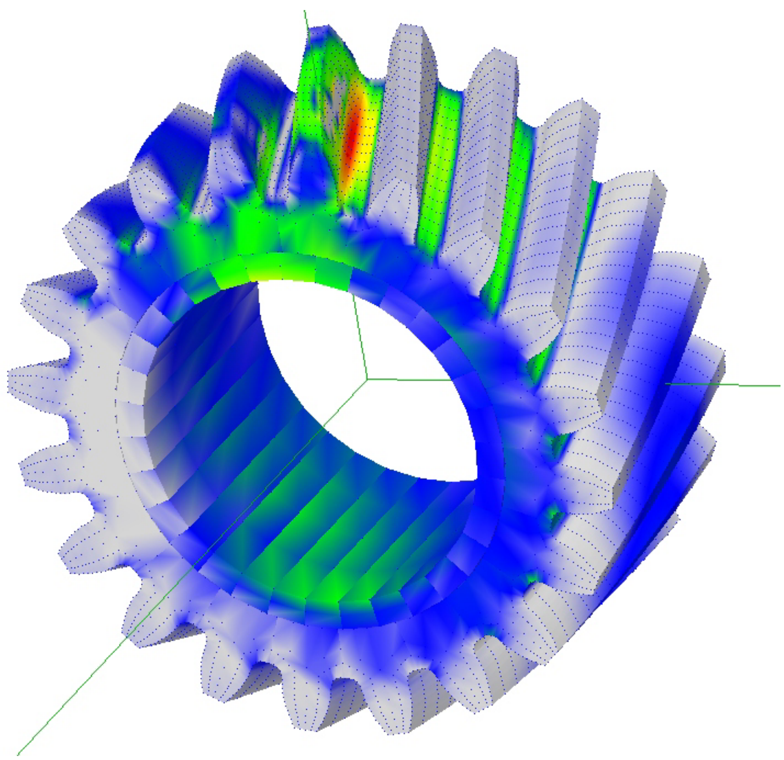


Figure 10.19: Finite element mesh so as to find the stress at a nodal point

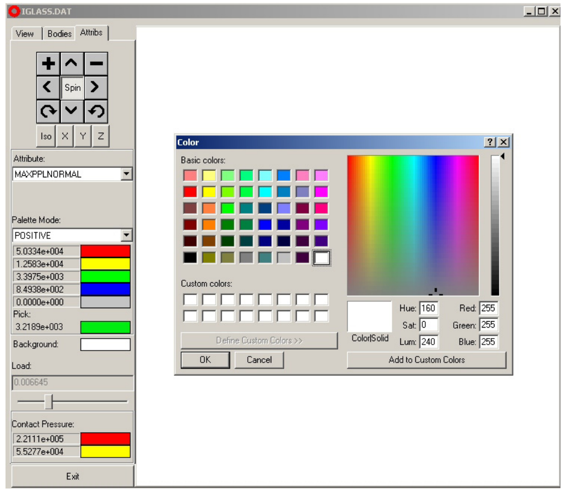


Figure 10.20: The background color popup window switch.

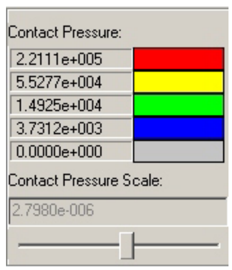


Figure 10.21: The Contact pattern menu.

The colors for minimum and maximum stress contours can be controlled using the palette mode menu shown in Figure 10.18. 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 10.19. 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 10.20 using which you can change the background color of the iglass graphics window.

The Contact pattern menu shown in Figure 10.21 is used to view the contact pressure pattern on the contacting surfaces. Figure 10.22 shows an example of a contact pattern on the gear tooth.

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

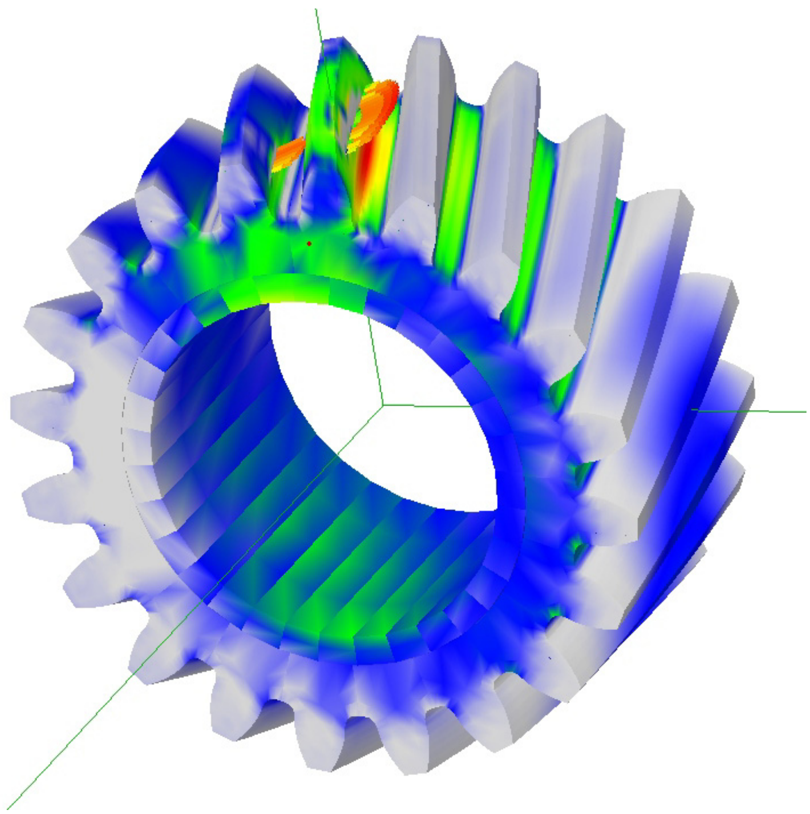


Figure 10.22: Example of a contact pattern on a gear tooth

Appendix A

Tooth Mesh Templates

The finite element meshes in the *Helical3D* 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 ζ direction is pointing outside the page.

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

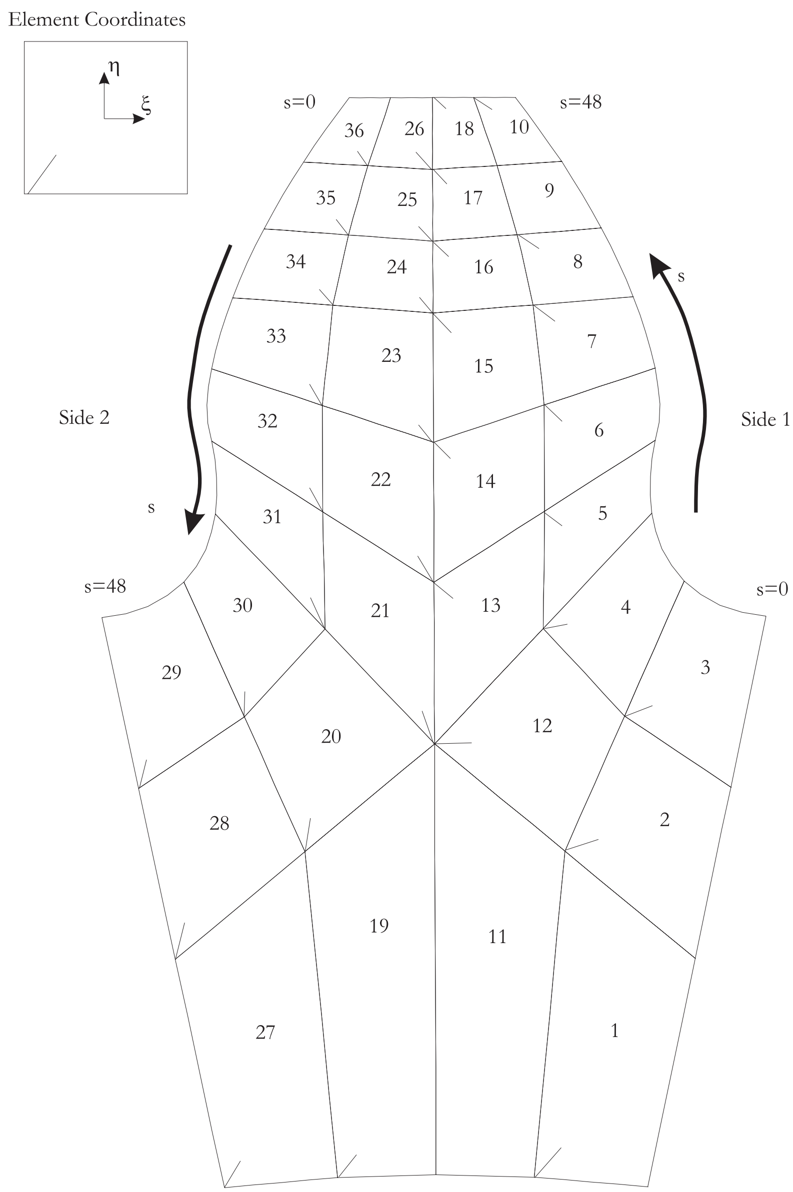


Figure A.1: The MEDIUM.TPL template file.

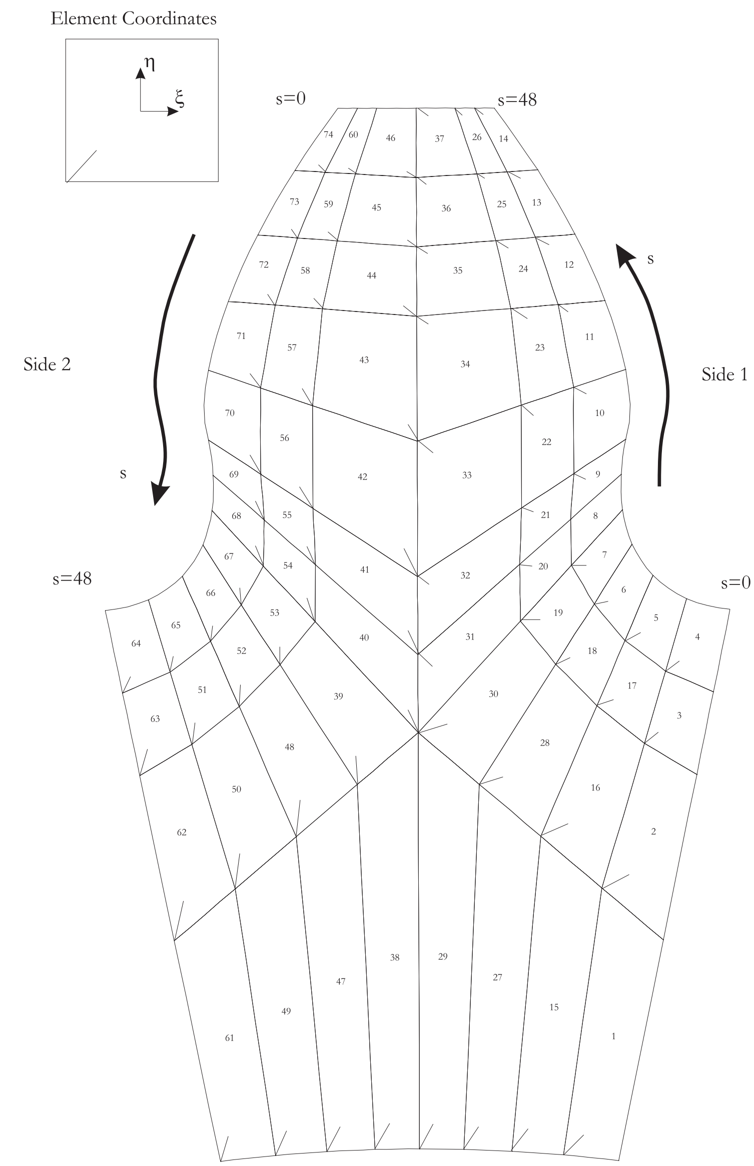


Figure A.2: The FINEROOT.TPL template file.

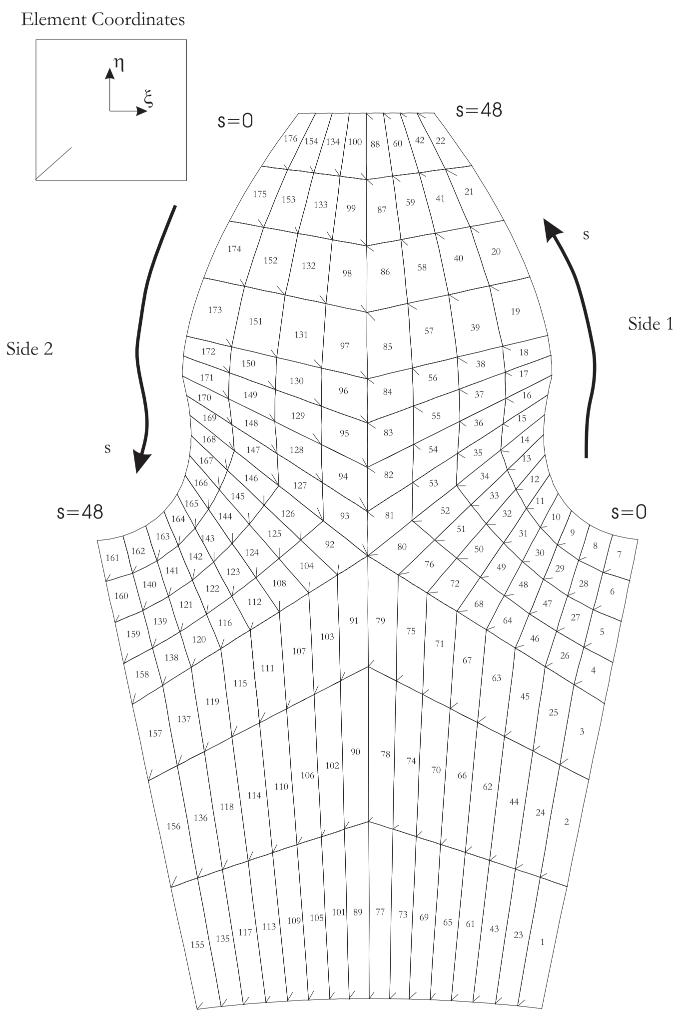


Figure A.3: The FINEST.TPL template file.

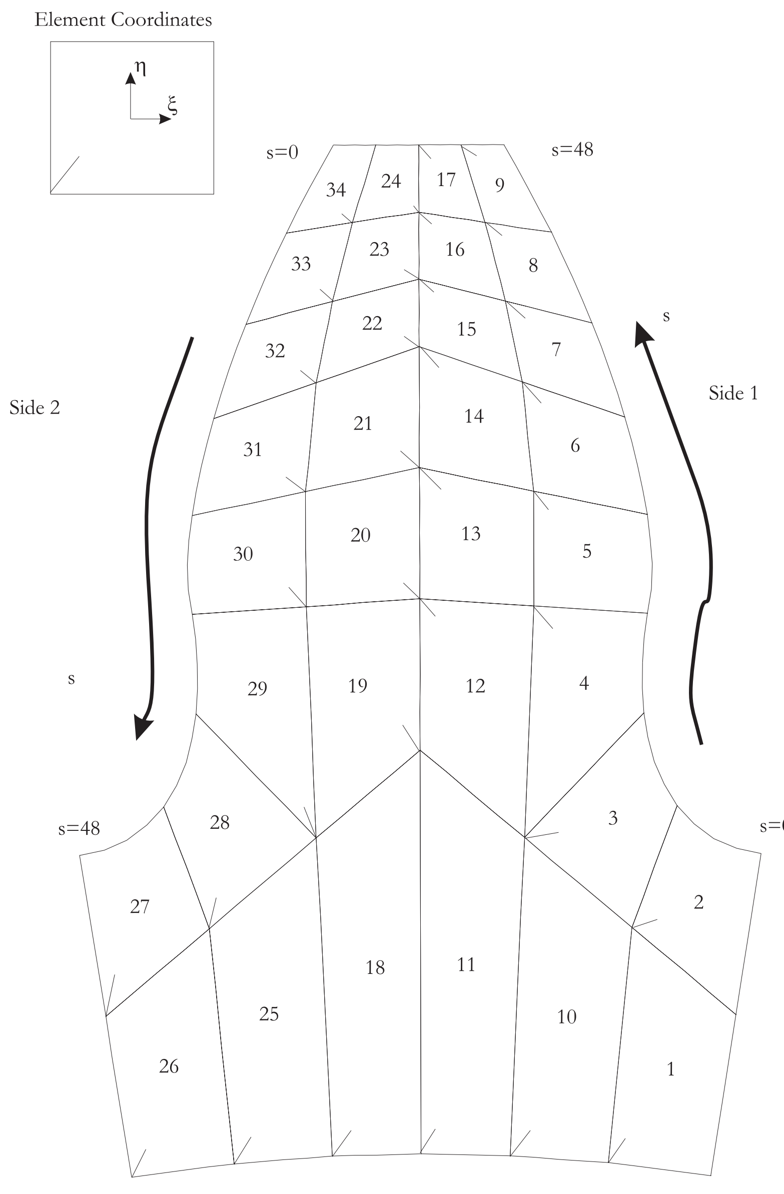


Figure A.4: The THINRIM.TPL template file.

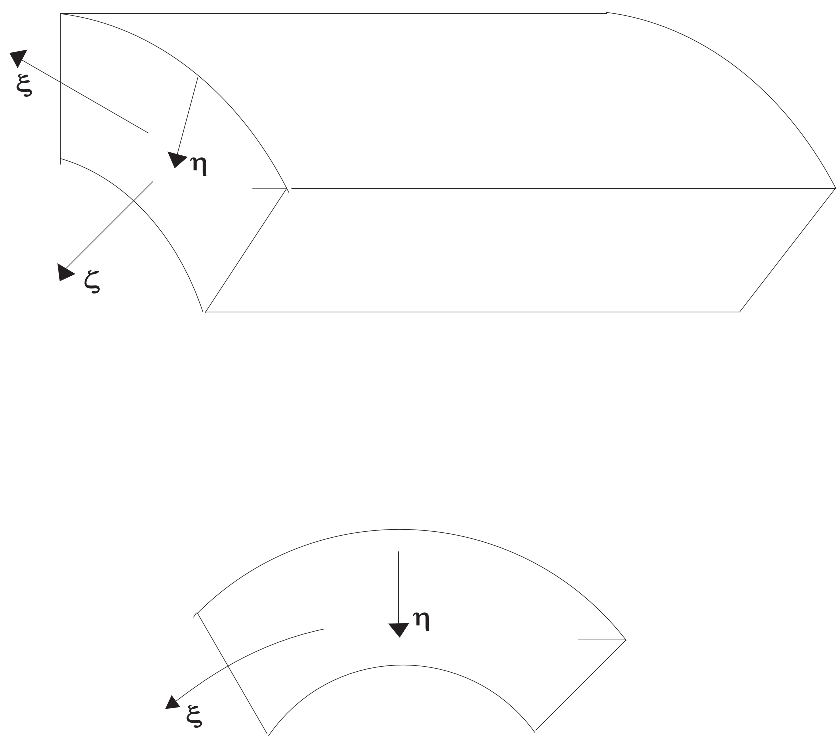


Figure A.5: Element orientation for the rim sector

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.