MULTYX

Pre- and Post-processing User's Manual

Advanced Numerical Solutions

ADVANCED NUMERICAL SOLUTIONS, LLC

CONTRIBUTORS

Sandeep Vijayakar Ph.D., Hilliard OH
Karthikeyan Marambedu, Hilliard OH
Brett Baker, Hilliard OH
October 9, 2024

CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

PRE- AND POST-PROCESSING

The PREPROC command in the main menu leads to the pre-processing menu shown in Figure 1.1. The POSTPROC command leads to the dialog box shown in Figure 1.2, 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 1.3 comes up.

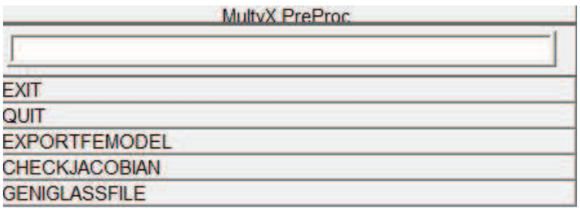

Figure 1.1 The pre-processing menu.

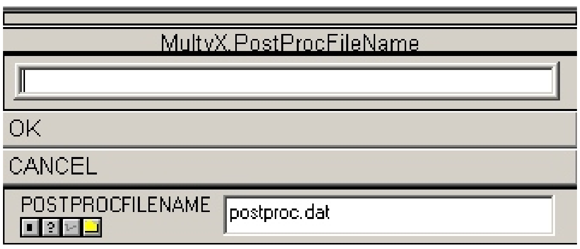

Figure 1.2 The post-processing file name dialog box.

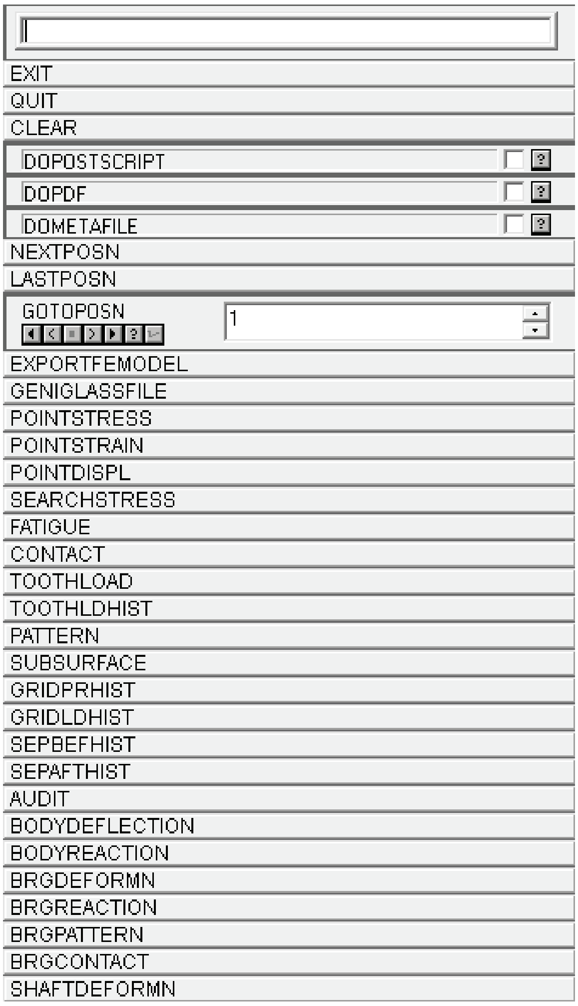

Figure 1.3 The post-processing menu.

1.1 The GETINERTIALPROPS command

The GETINERTIALPROPS command is available in the pre-processing menu and returns the mass, center of mass, and polar moment of inertia for a body given body.

1.2 The EXPORTFEMODEL command

The EXPORTFEMODEL menu is found in both the pre and post-processing menus. This menu allows the user to output a finite element model of a selected mesh in either the fixed or body reference frame. The input fields are described in 1.1.

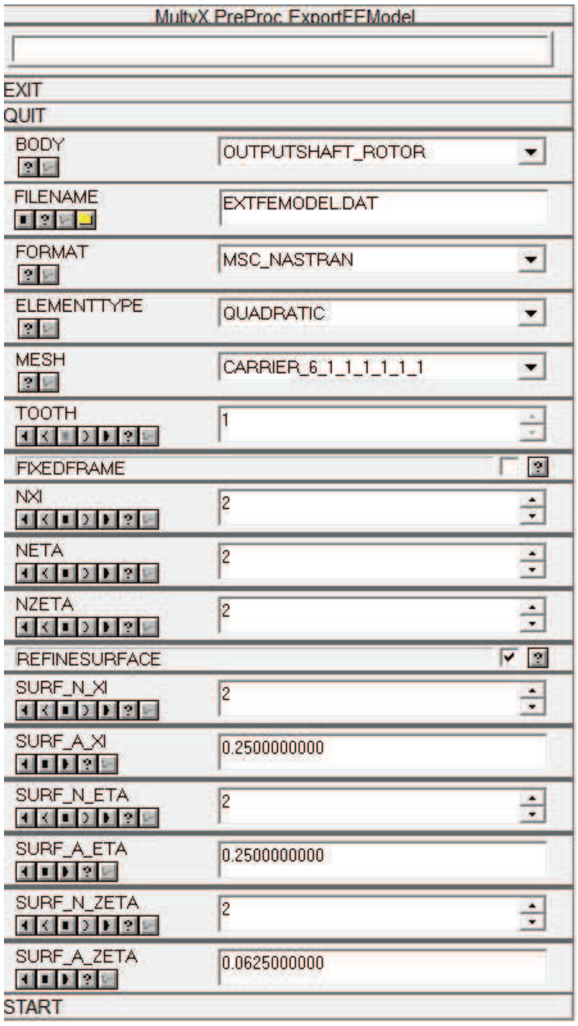

Figure 1.4 The export FE model menu.

Table 1.1 Export FE Model Menu Inputs

Item	Description	Condition
BODY	Body on which FE mesh to be exported lies.	
FILENAME	Filename to use for the exported body.	
FORMAT	FE file format. Options include FE mesh, stress invariants, or nodal loads	
ELEMENTTYPE	Type of elements use for exported FE file.	
MESH	Mesh to export to FE file.	
TOOTH	Tooth, or instance, of the mesh to export.	
FIXEDFRAME	If turned on, the file will be exported in the fixed reference frame. If turned off, the FE mesh will be exported in the body reference frame	
NXI	The number of elements to be generated in the cross, sectional direction Xi, of the original element.	
NETA	The number of elements to be generated in the cross, sectional direction Eta, of the original element.	
NZETA	The number of elements to be generated in the cross, sectional direction Zeta, of the original element.	
REFINESURFACE	If this flag is turned the mesh will be refined near the surface	
SURF_N_XI	This is the number of elements to be generated in cross sectional direction Xi of the original element, near the surface	IF REFINESURFACE_TRUE
SURF_A_XI	This is a factor which controls how much of the original element dimension is further subdivided to form the refined surface mesh in the Xi direction	IF REFINESURFACE_TRUE
SURF_N_ETA	This is the number of elements to be generated in cross sectional direction Eta of the original element, near the surface	IF REFINESURFACE_TRUE
SURF_A_ETA	This is a factor which controls how much of the original element dimension is further subdivided to form the refined surface mesh in the Eta direction	IF REFINESURFACE_TRUE
SURF_N_ZETA	This is the number of elements to be generated in cross sectional direction Zeta of the original element, near the surface	IF REFINESURFACE_TRUE
SURF_A_ZETA	This is a factor which controls how much of the original element dimension is further subdivided to form the refined surface mesh in the Zeta direction	IF REFINESURFACE_TRUE

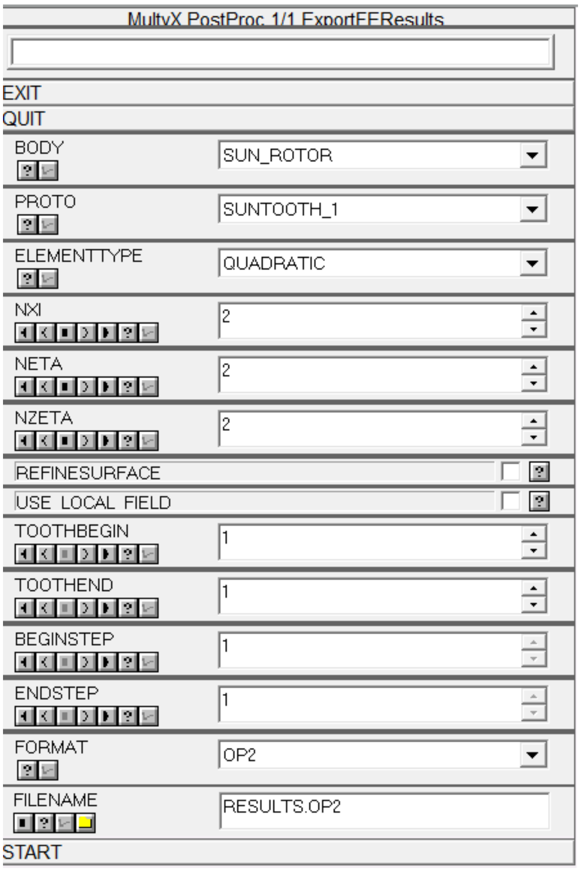

Figure 1.5 EXPORTFERESULTS postprocessing menu.

1.3 The EXPORTFERESULTS Command

The EXPORTFERESULTS post-processing command provides the user with the option of creating a *Nastran* results output file (`.OP2/.PCH`) containing stress and displacement results for a selected body. The *Guide* EXPORTFERESULTS postprocessing menu is displayed in Figure 1.5 and a description of its inputs are provided in Tables 1.2 and 1.3.

The refine surface option allows the user to refine the elements near the surface by specifying the number of elements in the element coordinate directions: xi, eta, zeta. Xi and eta are the directions normal and tangential to the surface, respectively. Zeta is in the direction of the rotational axis. The recommended values are provided in Figure 1.5. The local field option provides the user the option of calculating stresses near contact points using a local deformation field based on the analytical contact solution. The DISTMIN input specifies the distance from the contact point, up to which the local field is used. The *HyperView* load cases menu tree and contact results are shown in Figures 1.6 and 1.7.

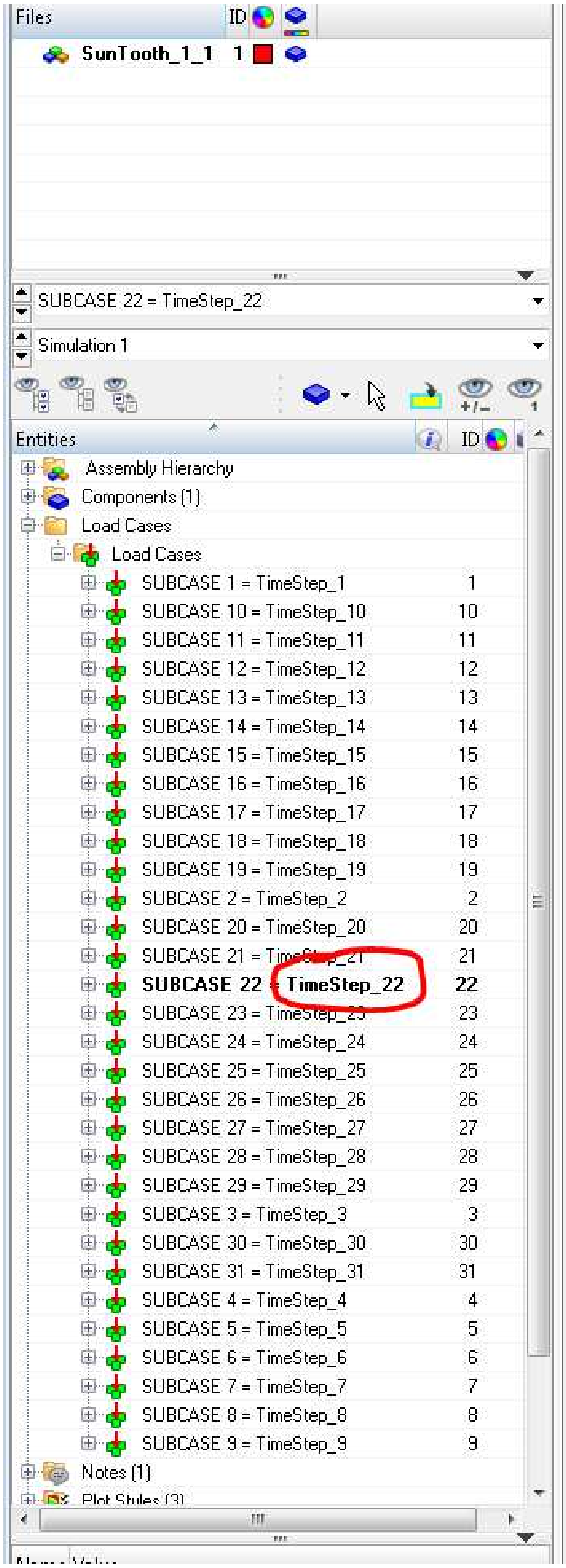

Figure 1.6 The load cases menu in HyperVeiw.

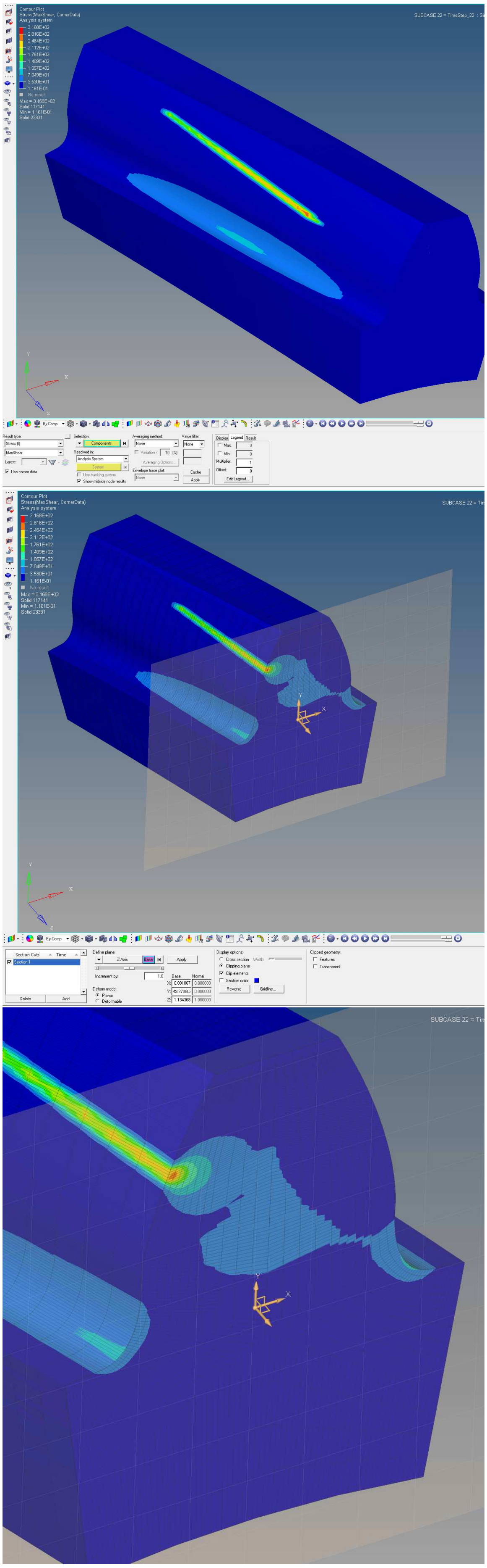

Table 1.2 Export FE Model Menu Inputs (1/2)

Item	Description	Condition
BODY	Body on which FE mesh to be exported lies.	
PROTO	Prototype of mesh to be exported.	
ELEMENTTYPE	Type of elements use for exported FE file.	
NXI	The number of elements to be generated in the cross, sectional direction Xi, of the original element.	
NETA	The number of elements to be generated in the cross, sectional direction Eta, of the original element.	
NZETA	The number of elements to be generated in the cross, sectional direction Zeta, of the original element.	
REFINESURFACE	If this flag is turned the mesh will be refined near the surface	
SURF_N_XI	This is the number of elements to be generated in cross sectional direction Xi of the original element, near the surface	IF REFINESURFACE_TRUE
SURF_A_XI	This is a factor which controls how much of the original element dimension is further subdivided to form the refined surface mesh in the Xi direction	IF REFINESURFACE_TRUE
SURF_N_ETA	This is the number of elements to be generated in cross sectional direction Eta of the original element, near the surface	IF REFINESURFACE_TRUE
SURF_A_ETA	This is a factor which controls how much of the original element dimension is further subdivided to form the refined surface mesh in the Eta direction	IF REFINESURFACE_TRUE
SURF_N_ZETA	This is the number of elements to be generated in cross sectional direction Zeta of the original element, near the surface	IF REFINESURFACE_TRUE
SURF_A_ZETA	This is a factor which controls how much of the original element dimension is further subdivided to form the refined surface mesh in the Zeta direction	IF REFINESURFACE_TRUE

Table 1.3 Export FE Model Menu Inputs (2/2)

Item	Description	Condition
USE_LOCAL_FIELD	If this flag is turned ON, the local deformation will be used instead of the FE field when the distance of the sampling point is less than the DISTMIN value	
DISTMIN	The distance used for local deformation if USE_LOCAL_FIELD is selected. Recommended value is $(toothheight)/3$	IF USE_LOCAL_FIELD=TRUE
TOOTHBEGIN	The beginning of the tooth instance range	
TOOTHEND	The end of the tooth instance range	
BEGINSTEP	The beginning of the time step range	
ENDSTEP	The end of the time step range	
FORMAT	The output file format	
FILENAME	The output file name	

1.4 The CHECKJACOBIAN command

The check Jacobian menu returns the element ID and location information for any elements with a negative Jacobian. The information is output to the information window.

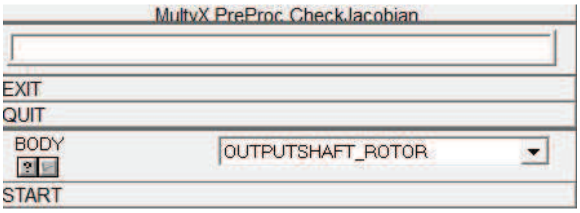

Figure 1.8 The export FE model menu.

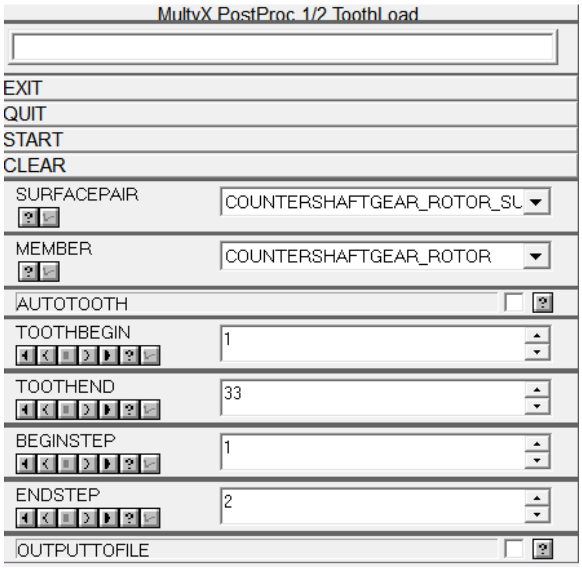

Figure 1.9 The TOOTHLOAD menu.

1.5 The TOOTHLOAD command

The TOOTHLOAD command in the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.9. This menu is used to generate a graph of tooth load vs. time. The SURFACEPAIR item selects the contact surface pair for which the load is of interest. Each surface pair has two contacting members or bodies. The MEMBER parameter selects one of these two bodies, and the TOOTHBEGIN and TOOTHEND items select a range of instance numbers (or tooth numbers) within that body. If TOOTHBEGIN is greater than TOOTHEND, then the range wraps around the last tooth of the surface. This range must contain 7 teeth or less. Selecting the AUTOTOOTH option automatically chooses the tooth range using the loaded teeth.

BEGINSTEP and ENDSTEP are used to select a range of time steps for which results have been stored in the post-processing file. Figure 1.10 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.

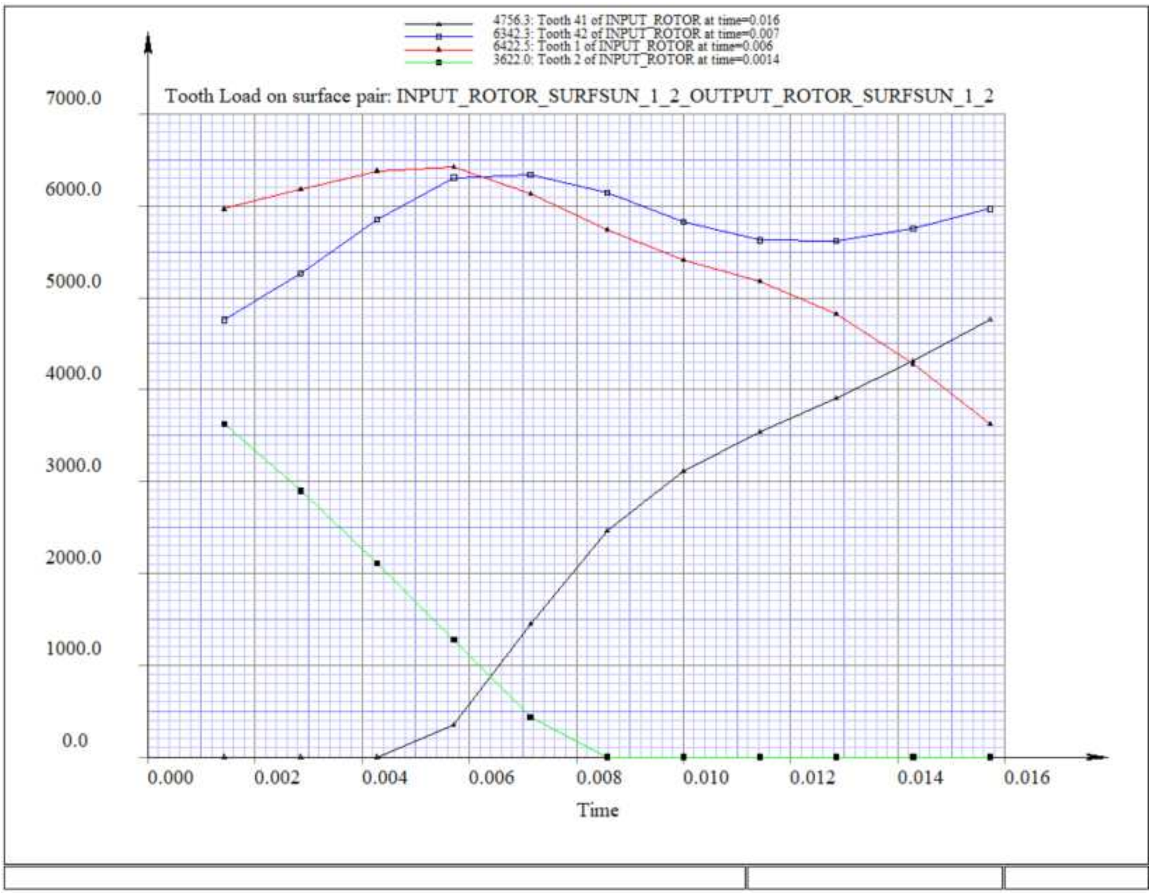

Figure 1.10 The tooth load vs. time graph generated by the TOOTHLOAD menu.

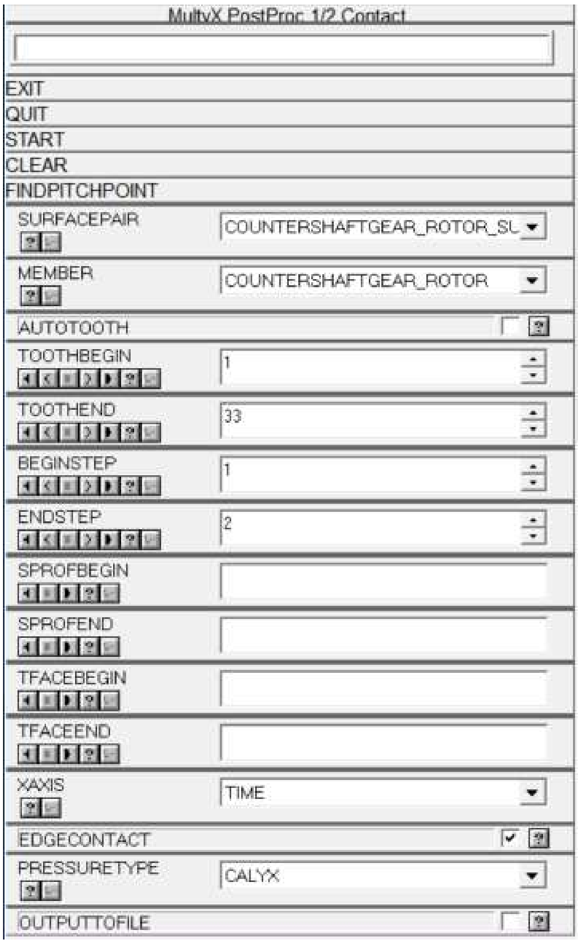

Figure 1.11 The CONTACT menu.

1.6 The CONTACT command

The CONTACT command in the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.11. 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. Selecting the AUTOTOOTH option automatically selects the tooth number range to cover the loaded teeth. 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 1.12 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. This high contact pressure near the edges can be filtered out by turning of EDGECONTACT. The plot without edge contact is shown in Figure 1.13.

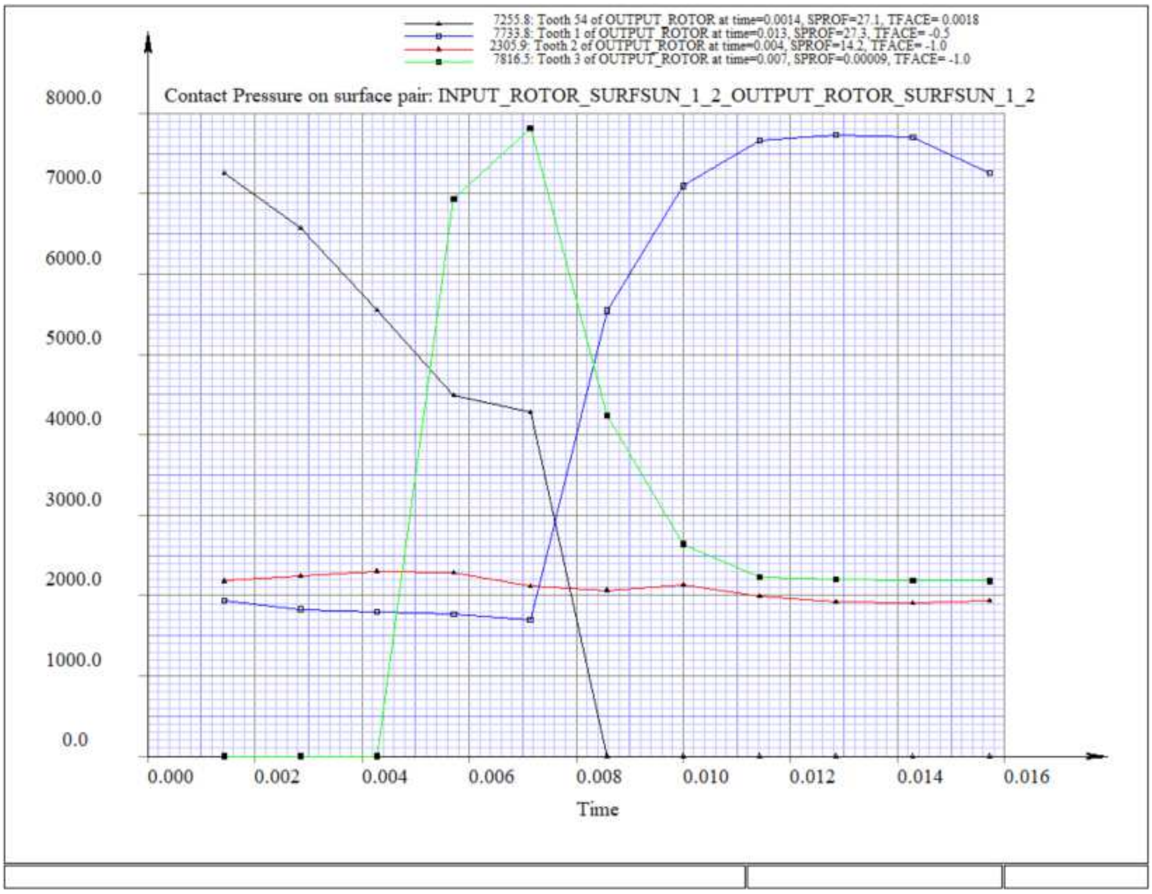

Figure 1.12 The tooth contact pressure vs. time graph generated by the CONTACT menu.

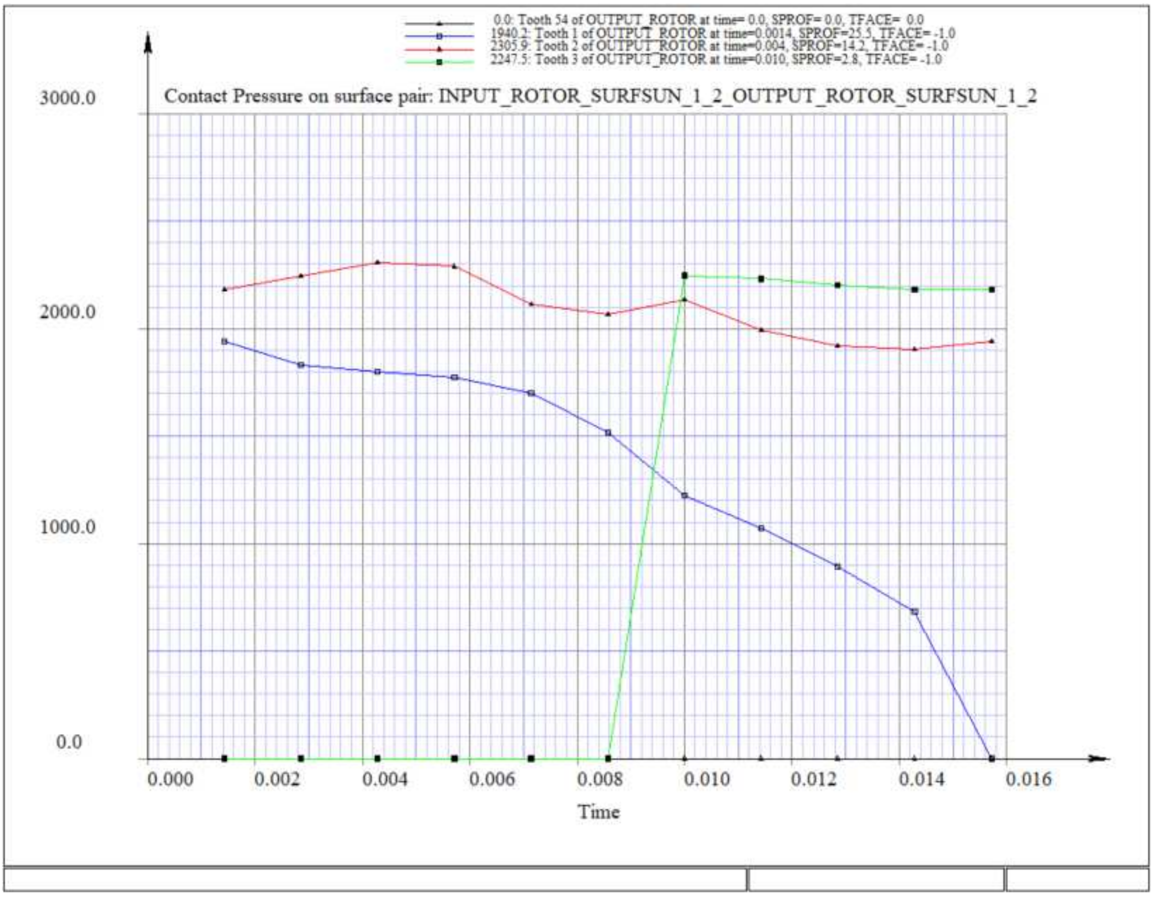

Figure 1.13 The tooth contact pressure vs. time graph generated by the CONTACT menu witHout EDGECONTACT.

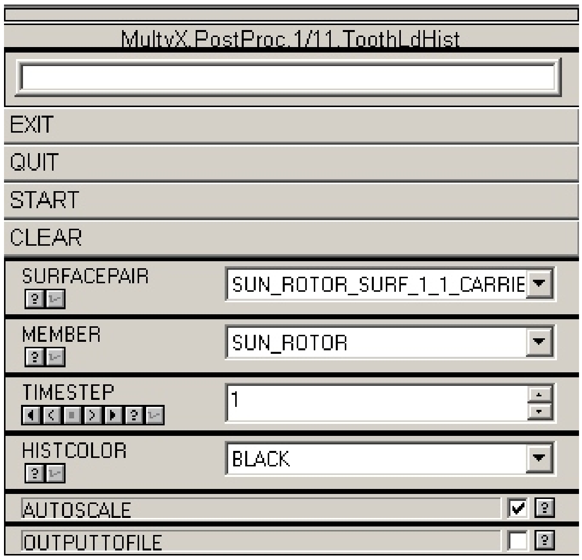

Figure 1.14 The TOOTHLDHIST menu.

1.7 The TOOTHLDHIST command

The TOOTHLDHIST command in the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.14. 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 1.15.

1.8 The SUBSURFACE command

The SUBSURFACE command in the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.16. This menu is used to generate a graph of subsurface stresses vs. depth under the most critical point in the contact zone. 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 MAXPPLNORMAL (the maximum principal normal stress s_1), MINPPLNORMAL (the minimum principal normal stress s_3), MAXSHEAR (the maximum shear stress τ_{max}) and VONMISES (the Von Mises' octahedral shear stress s_{VM}).

Figure 1.17 shows an example of a graph of sub-surface stress vs. depth.

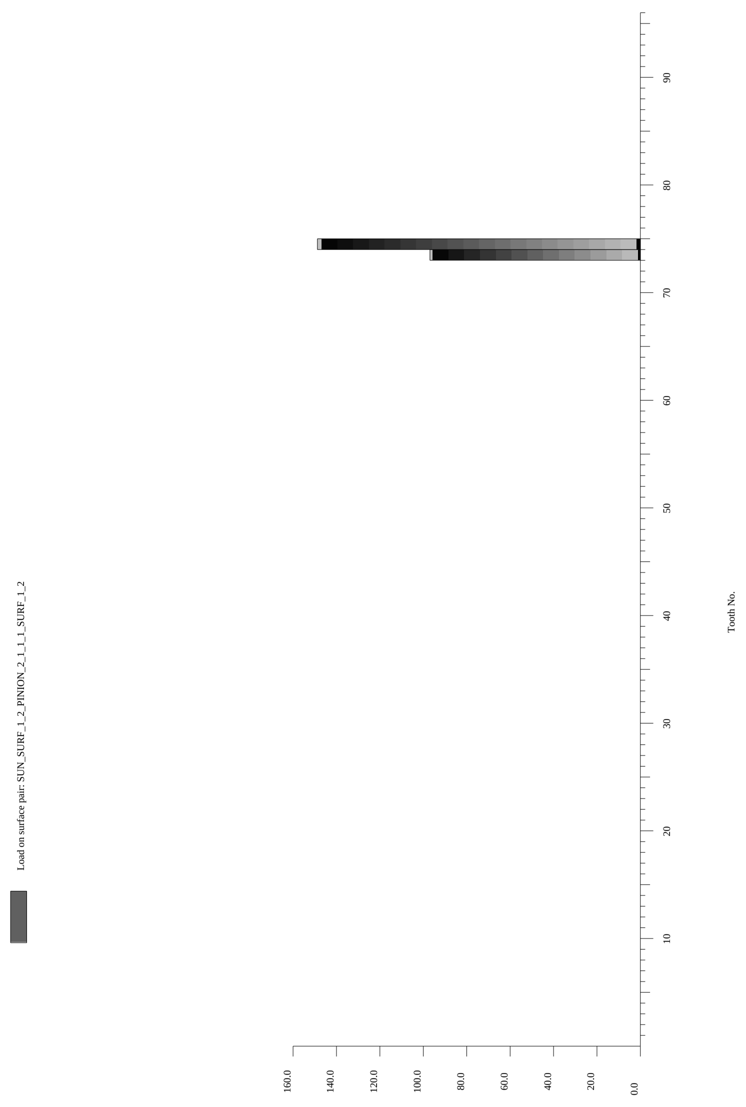

Figure 1.15 The tooth load histogram generated by the TOOTHLDHIST menu.

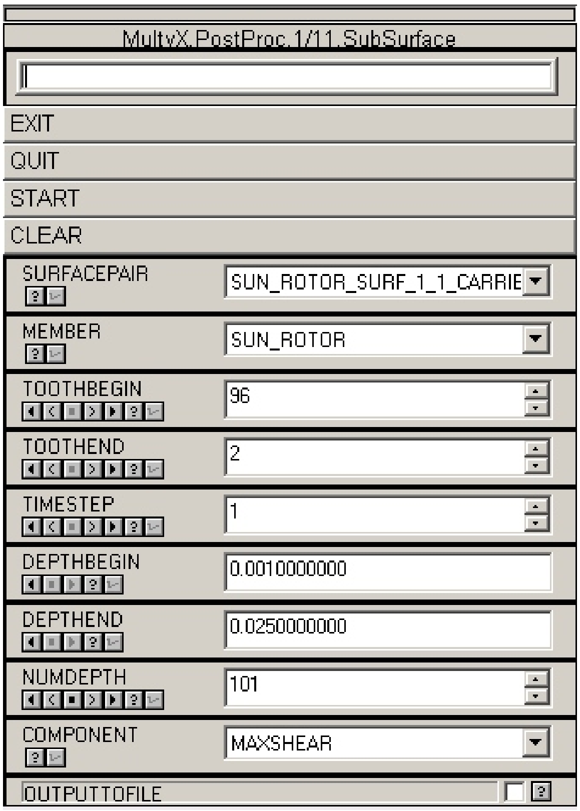

Figure 1.16 The SUBSURFACE menu.

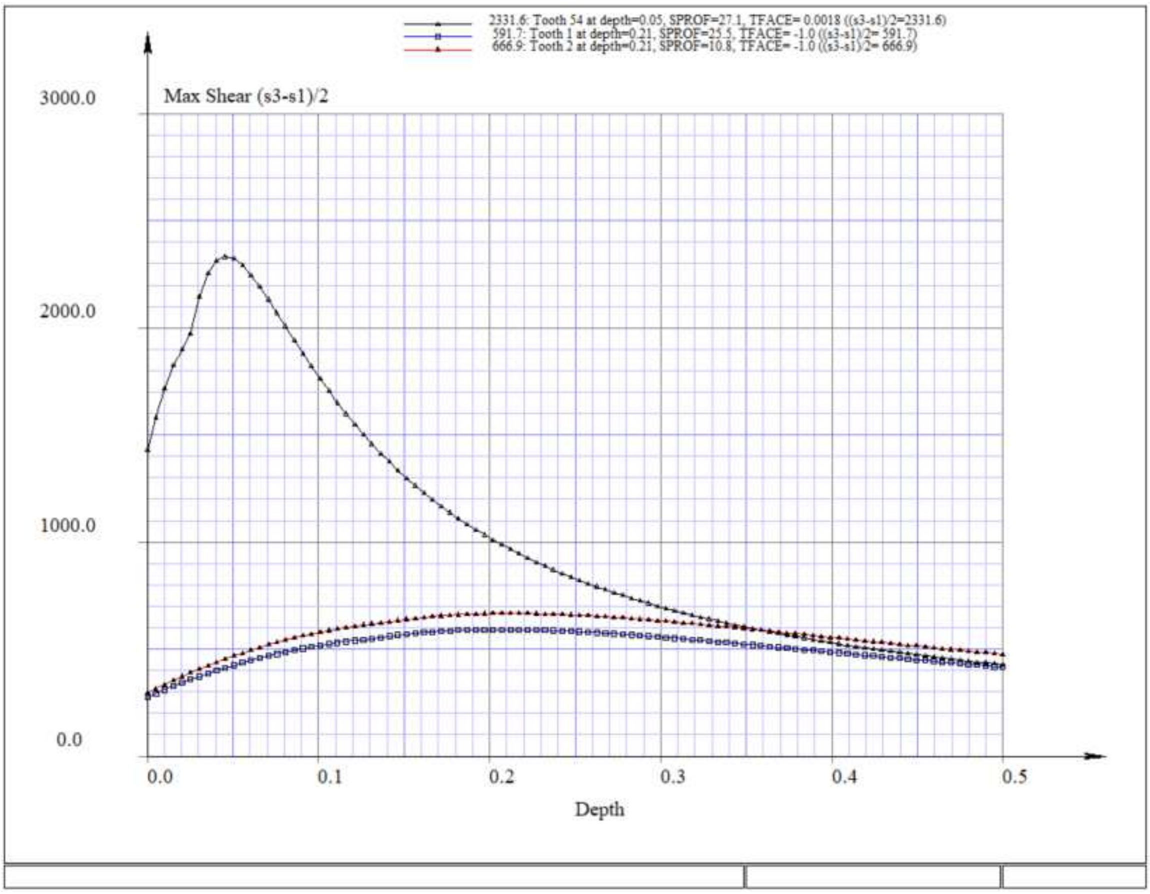

Figure 1.17 The sub-surface shear graph generated by the SUBSURFACE menu.

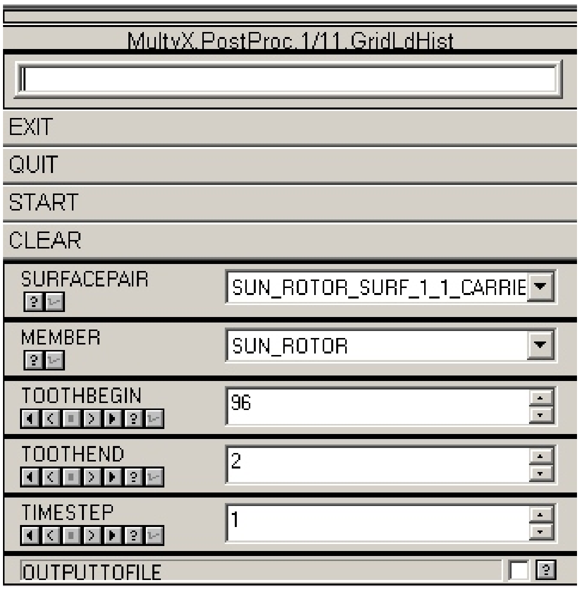

Figure 1.18 The GRIDLDHIST menu.

1.9 The GRIDLDHIST command

The GRIDLDHIST command in the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.18. 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 1.19 shows an example of a grid load histogram.

1.10 The GRIDPRHIST command

The GRIDPRHIST command in the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.20. 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 1.21 shows an example of a grid pressure histogram.

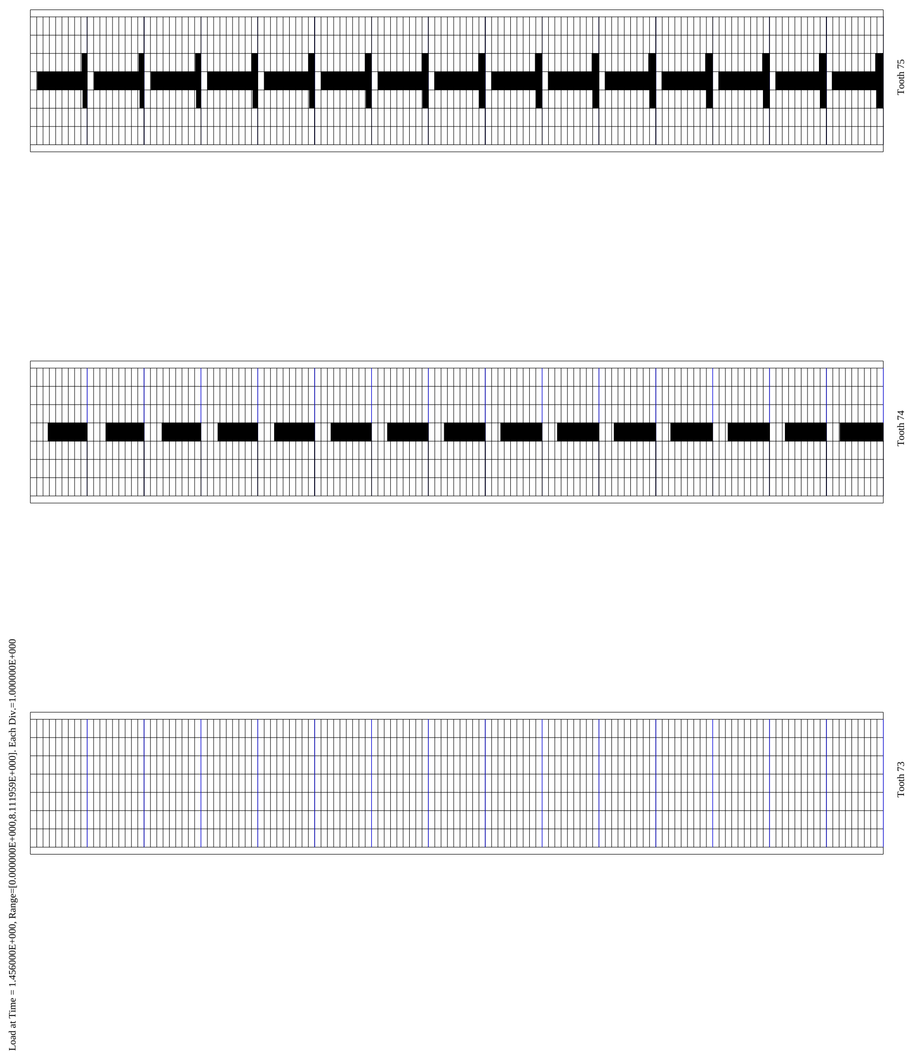

Figure 1.19 The grid load histogram generated by the GRIDLDHIST menu.

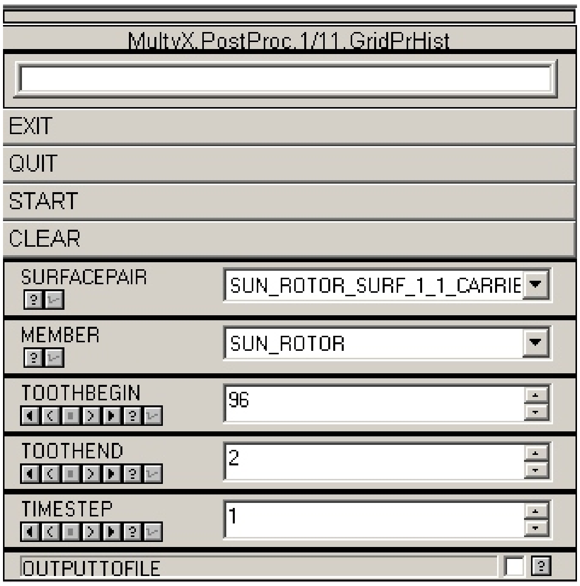

Figure 1.20 The GRIDPRHIST menu.

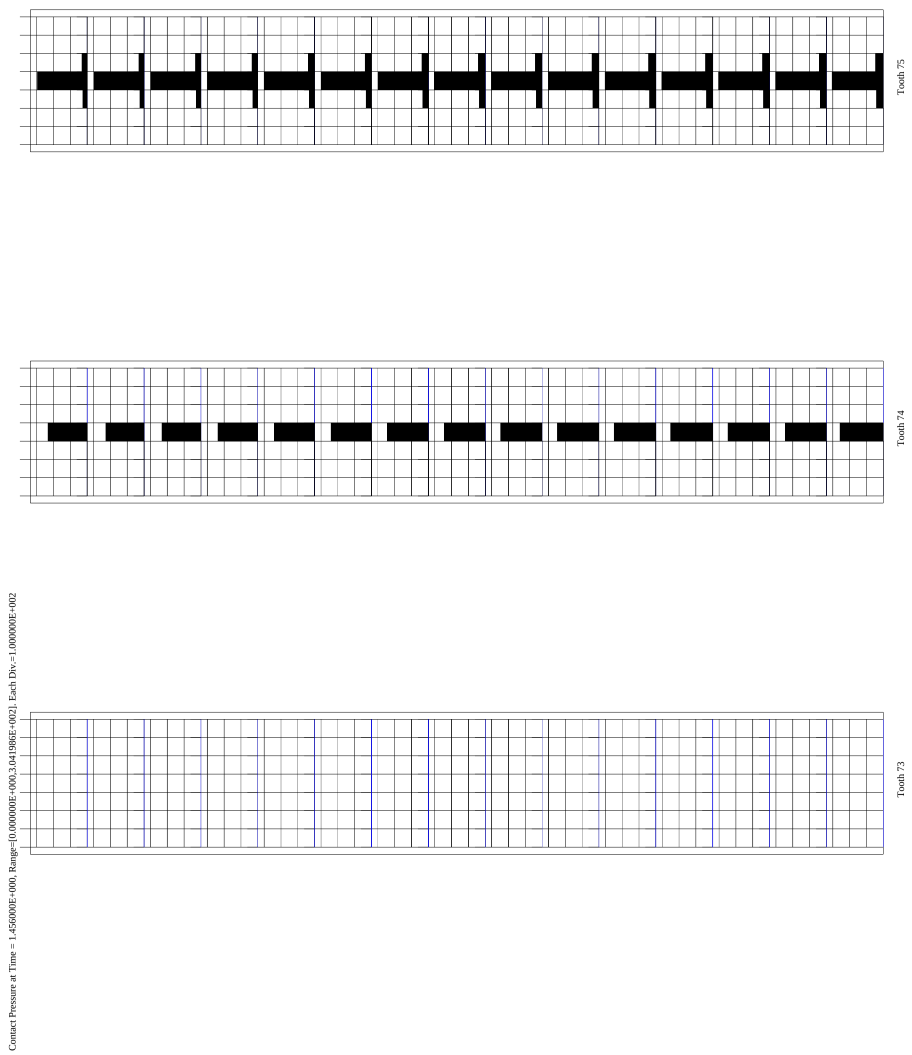

Figure 1.21 The grid pressure histogram generated by the GRIDPRHIST menu.

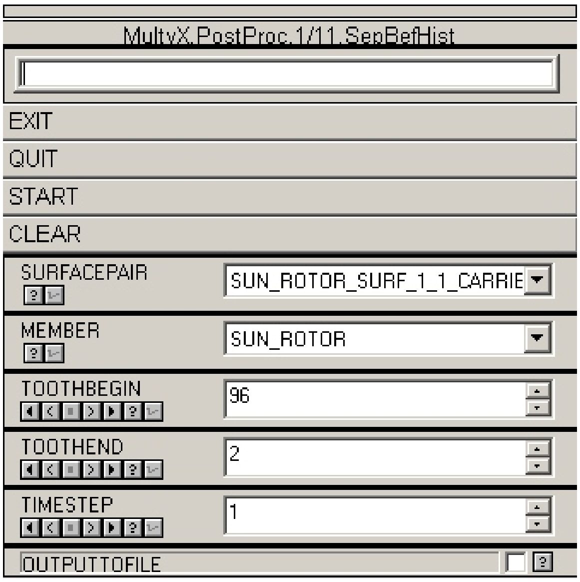

Figure 1.22 The SEPBEFHIST menu.

1.11 The SEPBEFHIST command

The SEPBEFHIST command in the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.22. This menu is used to to generate a histogram of the distribution of normal separation over individual contact grid cells, in the unloaded and undeformed state.

Figure 1.23 shows an example of a histogram of separation in the unloaded state. Negative separation values are possible in this histogram.

1.12 The SEPAFTHIST command

The SEPAFTHIST command in the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.24. This menu is used to to generate a histogram of the distribution of normal separation over individual contact grid cells, in the loaded and deformed state.

Figure 1.25 shows an example of a histogram of separation in the loaded state. These separation values must be either zero or positive.

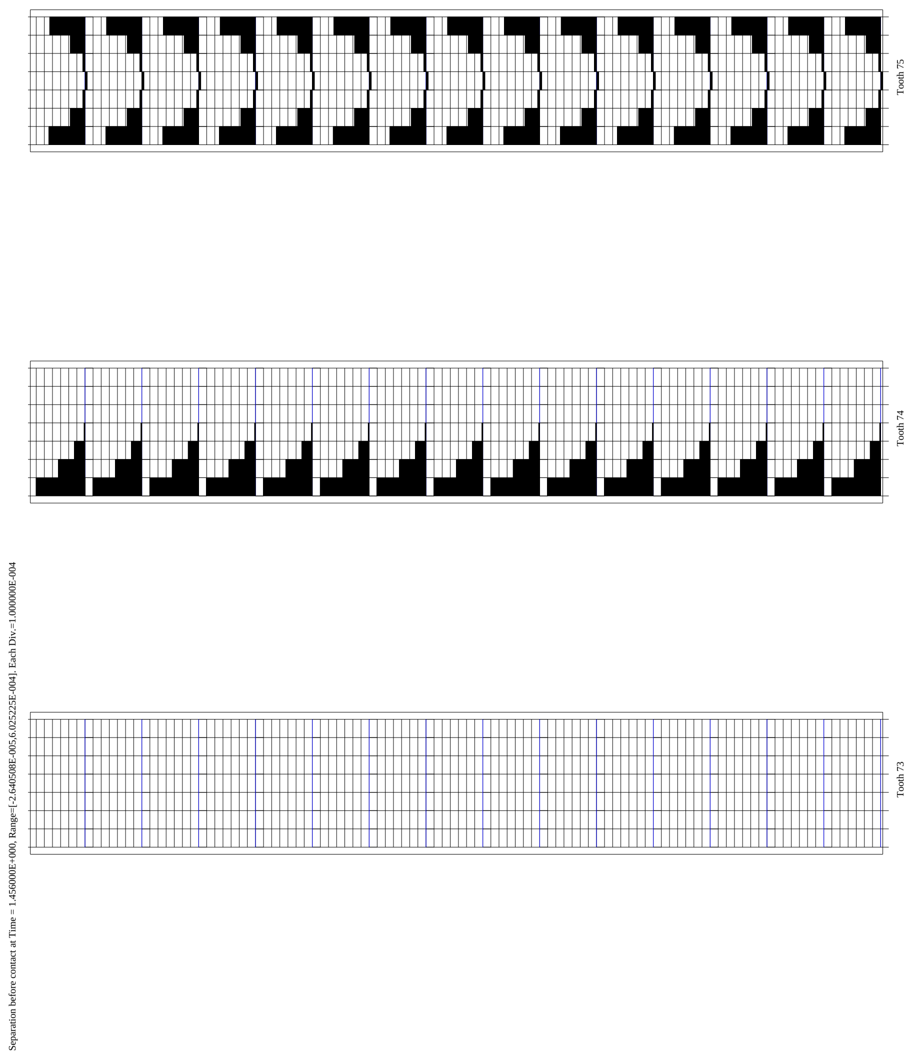

Figure 1.23 The histogram of grid separation before contact, generated by the SEPBEFHIST menu.

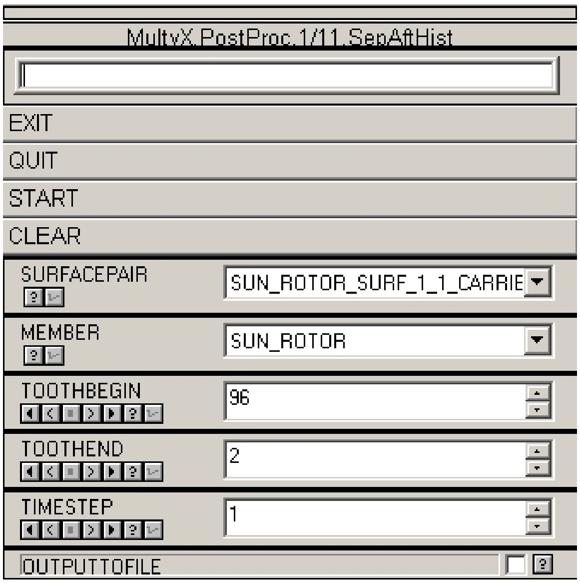

Figure 1.24 The SEPAFTHIST menu.

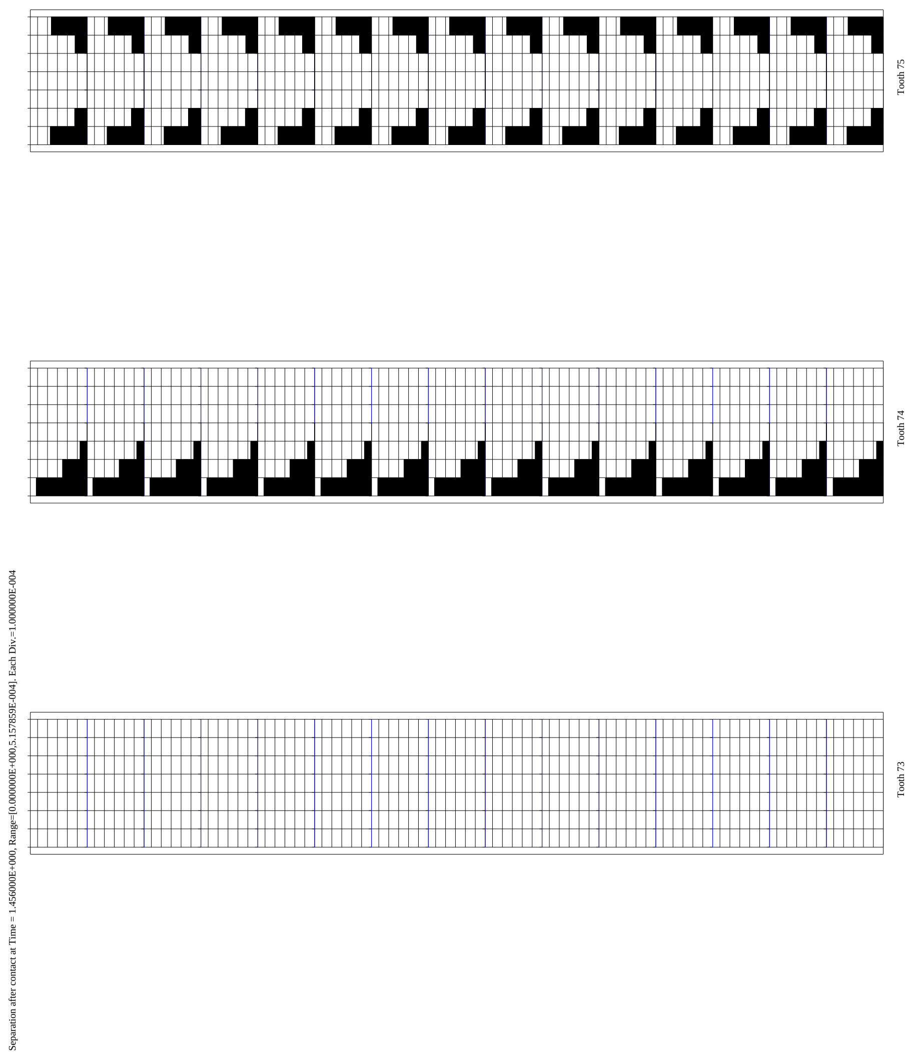

Figure 1.25 The histogram of grid separation after contact, generated by the SEPAFTHIST menu.

1.13 The SEARCHSTRESS command

The SEARCHSTRESS command of the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.26. 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.

Selection of the AUTOTOOTH option automatically selects the tooth range for a given surface pair. The surface pair is selected from a drop-down menu that appears upon selection of the AUTOTOOTH flag. The AUTOTOOTH option is not visible for conformal surface pairs as all teeth in conformal pairs are loaded.

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.

The FOCUS NEXT SEARCH option allows the user to focus the next run on the critical point only. USE LOCAL FIELD uses the local deformation field to compute the stress values near the contact area. DISTMIN specifies the minimum distance from a contact point that is used for the local field.

File output is controlled by the OUTPUTTOFILE, FILENAME and APPEND items. Figure 1.27 shows an example of stress as a function of time, Figure 1.28 shows stress as a function of profile position. Sharp oscillations can be seen in this graph in the vicinity of the concentrated contact loads. Figure 1.29 shows a graph of stress vs. face.

1.14 The POINTSTRESS command

The POINTSTRESS command of the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.30. 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 1.31 shows an example of the graph generated by this menu.

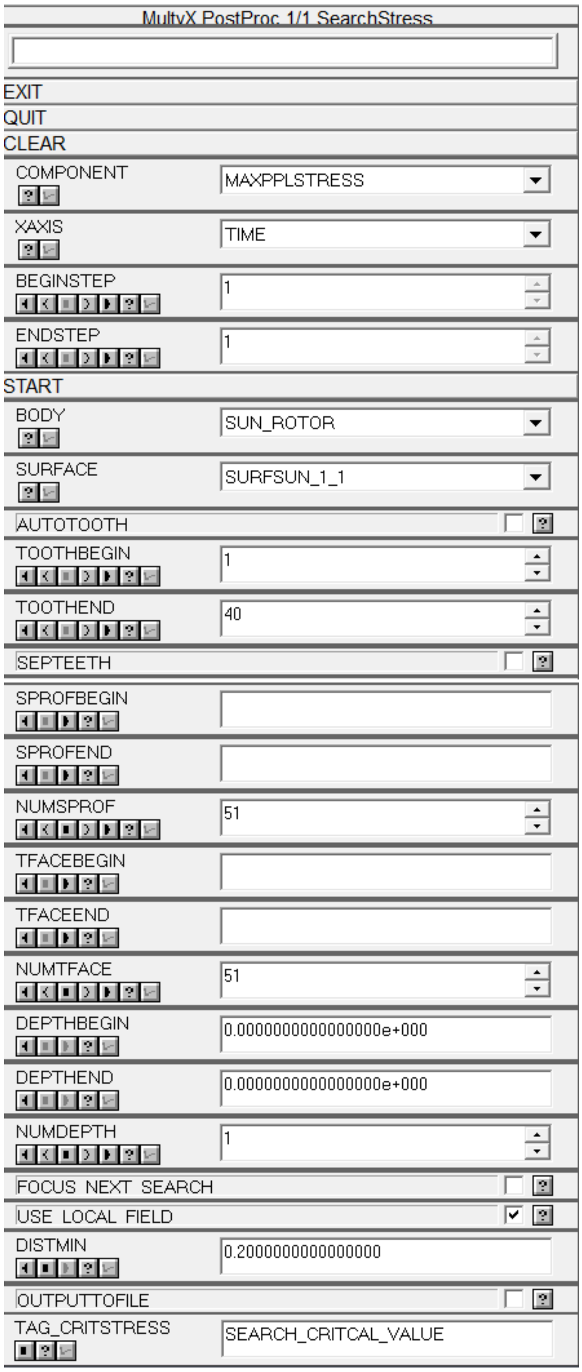

Figure 1.26 The SEARCHSTRESS menu

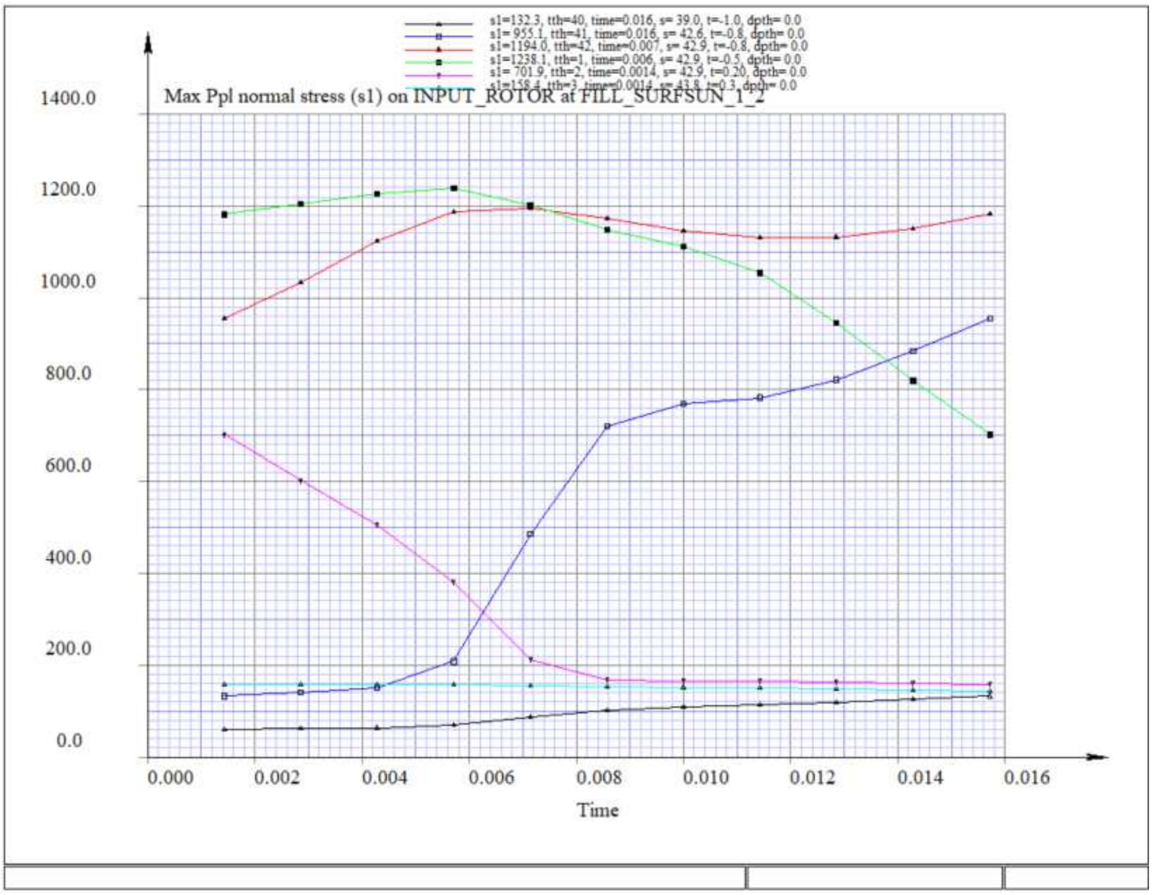

Figure 1.27 The graph of root stress vs. time, generated by the SEARCHSTRESS menu.

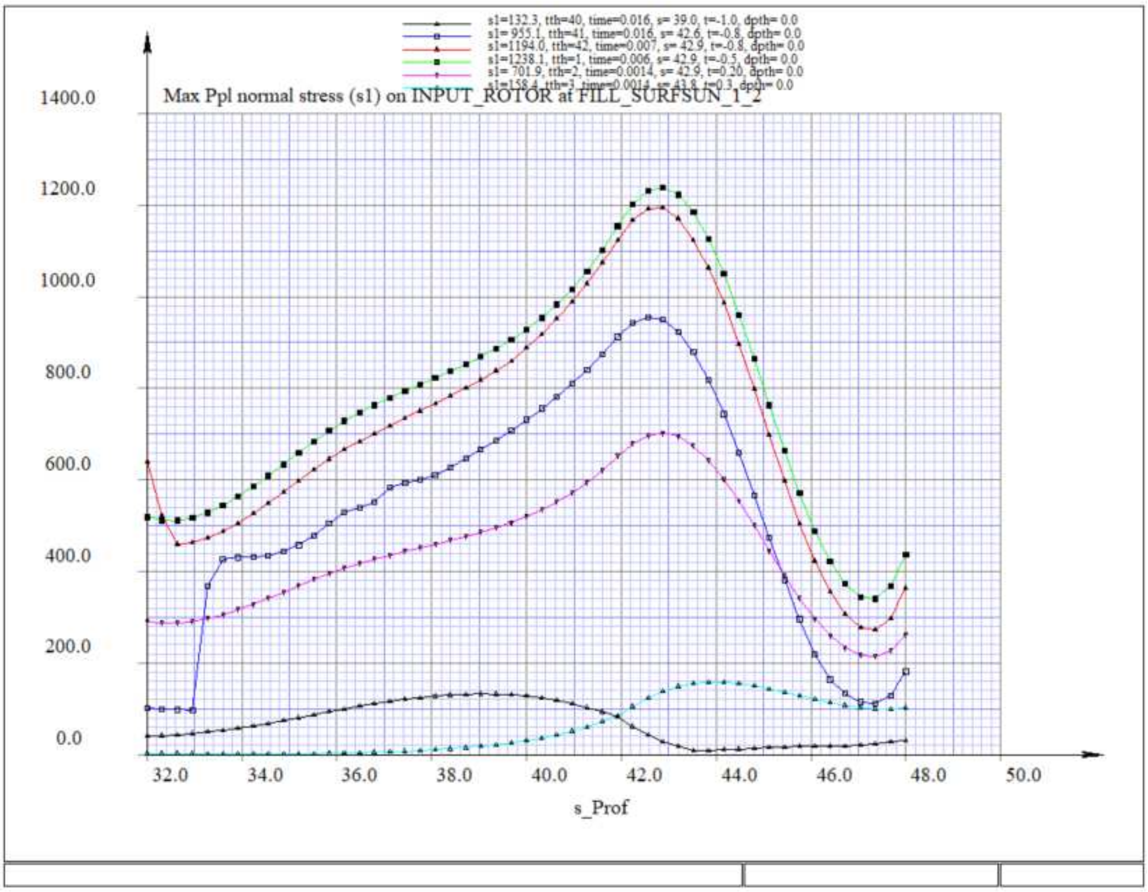

Figure 1.28 The graph of root stress vs. profile, generated by the SEARCHSTRESS menu.

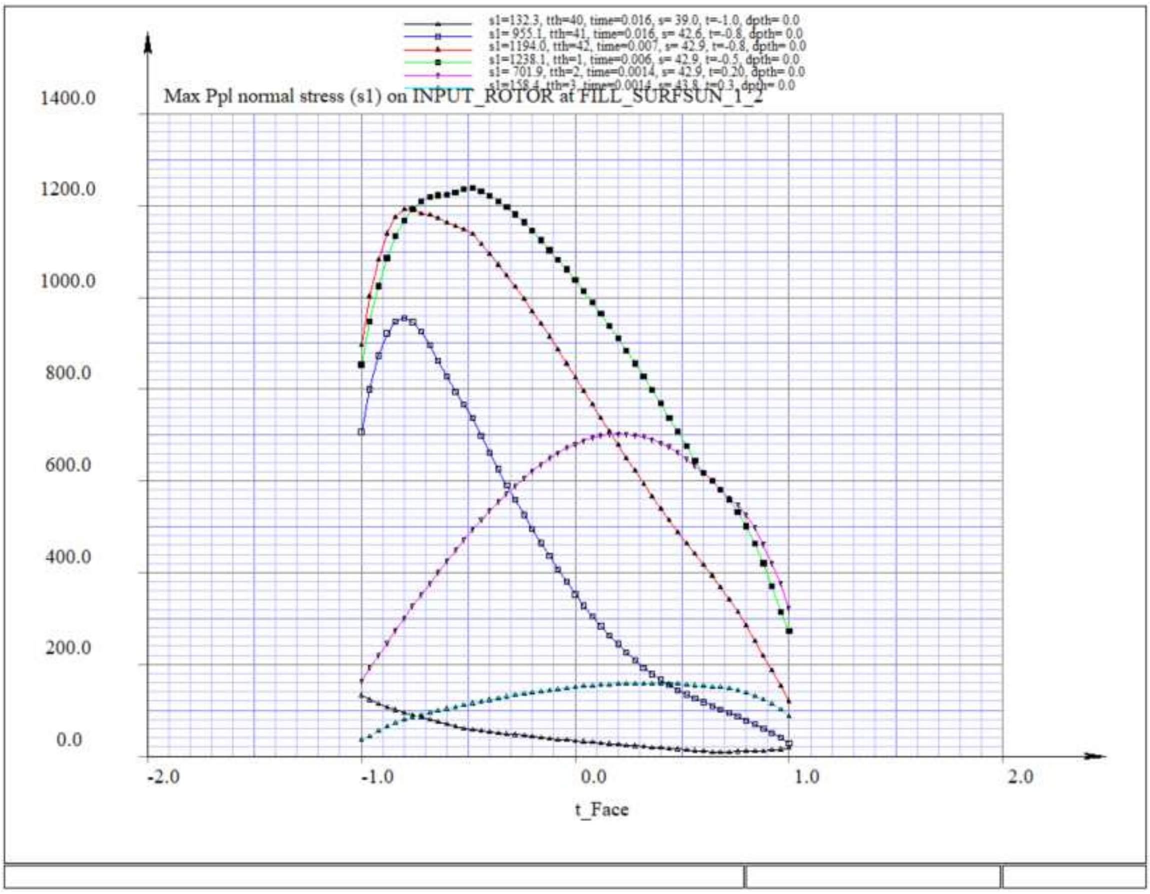

Figure 1.29 The graph of root stress vs. face, generated by the SEARCHSTRESS menu.

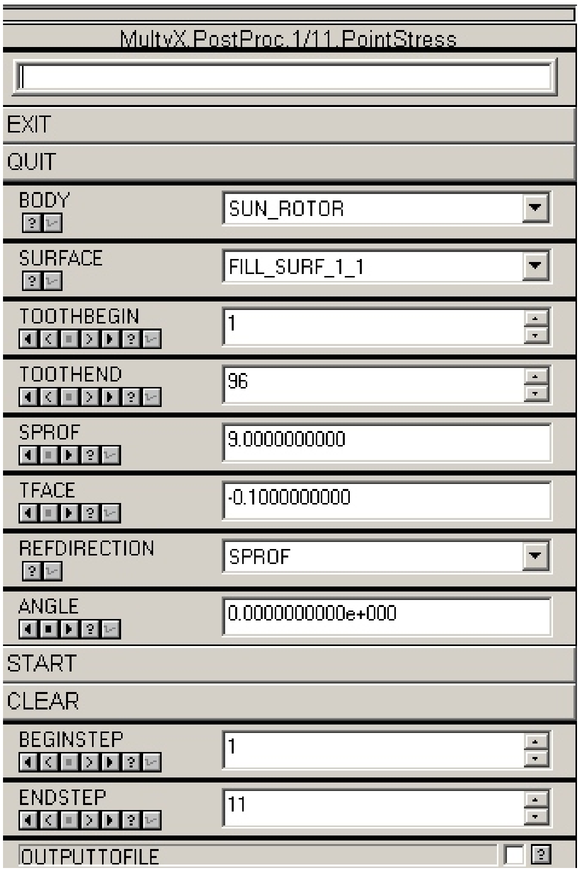

Figure 1.30 The POINTSTRESS menu.

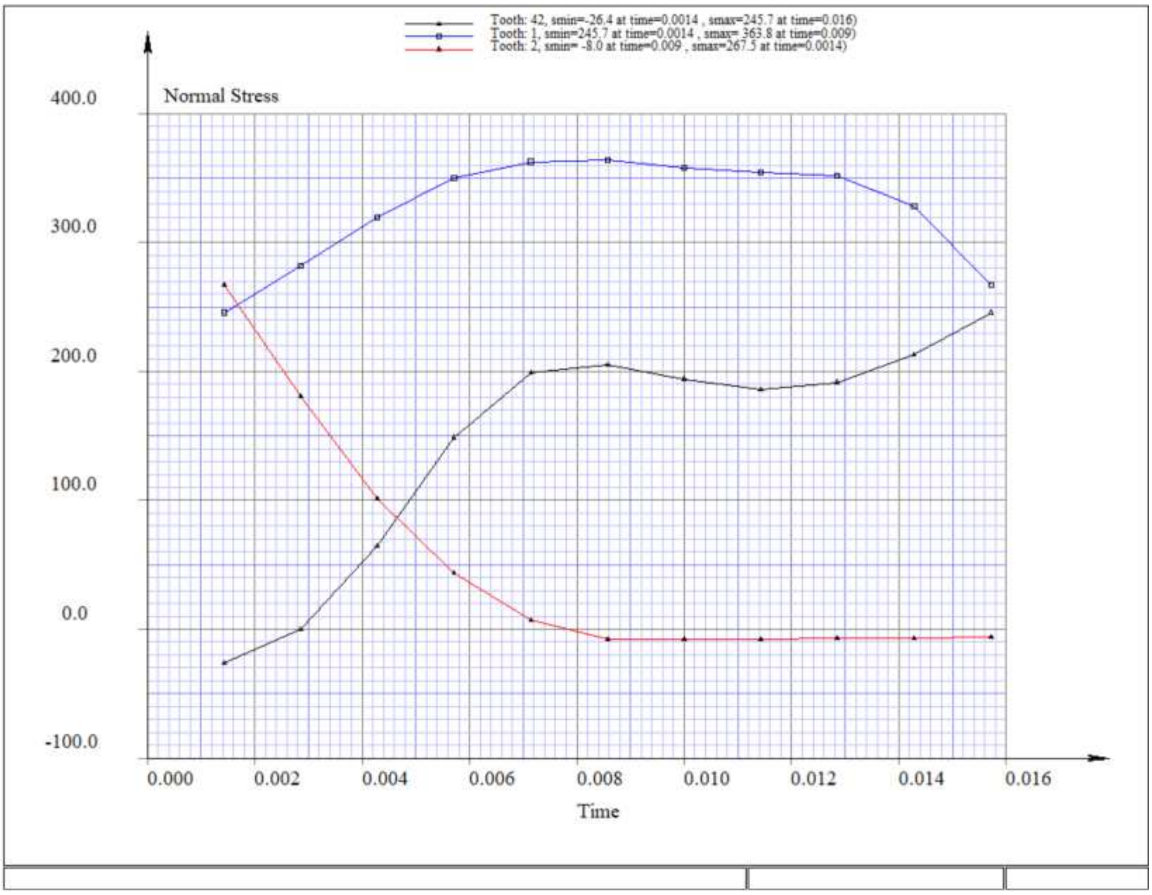

Figure 1.31 The graph of root stress vs. face, generated by the POINTSTRESS menu.

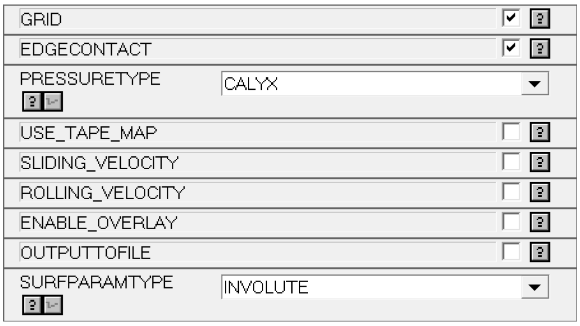

Figure 1.32 PATTERN menu with involute angle grid overlay (SURFPARAMTYPE=INVOLUTE).

1.15 The PATTERN command

The PATTERN command of the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.37. This menu is used to draw color or contour distribution of contact pressure, flash temperature, specific film thickness on the tooth surface and surface wear.

The tooth surface is selected by choosing the gear pair in the SURFACEPAIR box and a member from the MEMBER drop down list. A range of teeth in contact is selected through the TOOTHBEGIN and TOOTHEND items. The range of time steps is specified by the BEGINSTEP and ENDSTEP items.

The parameter to be plotted is chosen in the PATTERNCOMPONET menu item. The distribution can be displayed in color if the COLORS box is checked, or with contour lines if the CONTOURS box is checked. At least one of them must be turned ON to draw the pattern.

The pattern drawing is not three-dimensional. It is a projection of the contact surface in the $r - z$ coordinate plane. A line is drawn on the plot at the root-face transition radius.

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

If the FLIP box is checked, the orientation of the Z axis on the plot is flipped pointing towards left of the screen.By default, the Z axis points towards the right end of the screen.

The GRID option enables the user to overlay the *Calyx* parametric tooth surface coordinates (S&T), involute coordinates (angles), or cyclindrical coordinates (radius) on the pattern distribution. When the GRID box is selected, the coordinate type for the grid can be selected from the SURFPARAMTYPE drop-down menu. Figures 1.32 through 1.35 show the menus and patterns for the involute and cylindrical grid overlays.

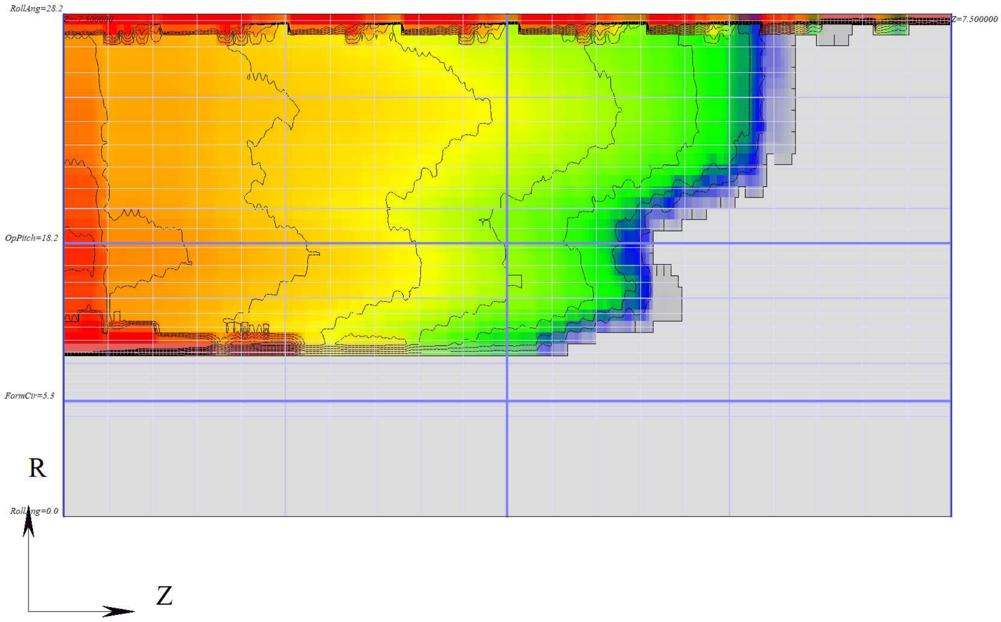

Figure 1.33 Contact pattern with involute angle grid overlay.

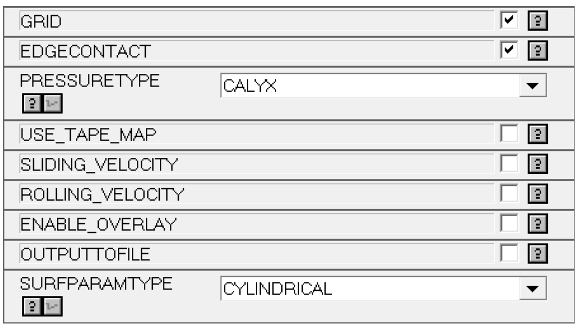

Figure 1.34 PATTERN menu with cylindrical coordinate grid overlay (SURFPARAMTYPE=CYLINDRICAL).

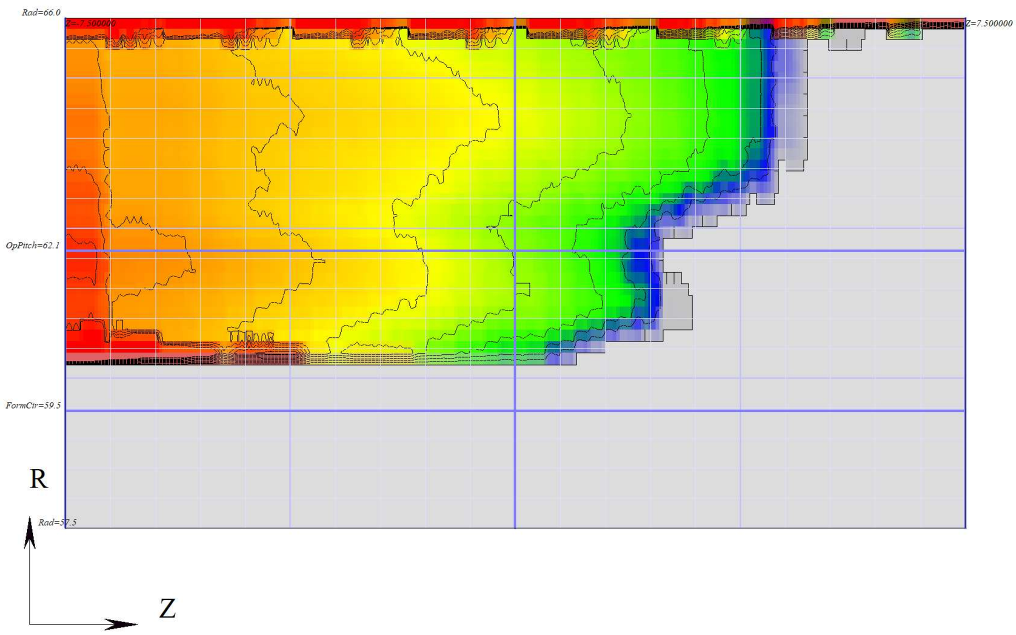

Figure 1.35 Contact pattern with cylindrical coordinate grid overlay.

To draw sliding velocity and rolling velocity on the pattern plot, the corresponding SLIDING VELOCITY and ROLLING VELOCITY checkbox must be turned ON. This draws a red arrow in the direction of the vector.

The ENABLE_OVERLAY checkbox allows the user to import an outline of the tooth to overlay onto the contact pattern plot. This allows the user to visualize the pattern on a gear tooth with any special features (chamfers, etc.) since the default drawing is on top of a rectangular tooth surface. The overlay file format is such that each line in the file contains the (r,z) coordinates of a single point on the curve. The curve is closed automatically by joining the first and last points. Multiple curves can be created by leaving blank lines in between each curve.

MODELUNITS will be the system of units used in setting up the model. This is needed to convert the stresses and gear material properties from model units to ISO standard units to calculate flash temperature and specific film thickness.

1.15.1 Contact Pattern

To draw the contact pattern,PATTERNCOMPONENT must be set to CONTACTPRESSURE and the PRESSURESTYPE is set to either CALYX or HERTZ. CALYX uses the contact grid based pressure values calculated in *Multyx* and HERTZ uses the Hertz formula along with the load intensity and relative curvatures.A sample contact pattern is shown in Figure 1.38.

1.15.1.1 Edge Contact Considerations *Transmission3D* uses a linear elastic deformation model, which has the consequence of pressure singularities (infinite pressure) where a tooth makes edge contact. In a numerical program like *Transmission3D*, this produces a vary large pressure value that increases with resolution and fails to converge locally.

The Hertz formula is valid for infinitely long contacting cylinders with constant curvature. Although it is invalid for gears where curvatures is not constant, it is still useful for comparison so it is provided. In real world scenarios, there are no singularities. Near the edge, linear elasticity breaks down and most likely enters the elastic-plastic regime, the details of which have not been sufficiently studied.

The degree to which edge contact matters is largely dependent upon the design criteria. In aerospace applications, where safety considerations are of upmost importance, edge contact is not allowed. In automotive applications, noise is often an important consideration, and gear sets where the contact pattern does not touch the edge produce more noise due to a larger transmission error. Most automotive engineers will ignore edge contact and filter out the higher values when extracting pressure numbers.

The EDGECONTACT box can be used to enable/disable edge contact in both the PATTERN and CONTACT postprocessing menus.

1.15.2 Flash Temperature

The flash temperature is calculated based on the ISO standard, *ISO/TR 15144-1,Calculation of micropitting load capacity of cylindrical spur and helical gears* [19].To get a distribution of flash temperature at the contact surface, the PATTERNCOMPONENT is set to FLASHTEMP and set coefficient of friction MU, specific heat of conductivity ($LAMBDA\_SI$) and specific heat capacity(CP_SI)for the gears as shown in Figure 1.37.Since the ISO standard is based on SI units, gear thermal properties, specific heat capacity and conductivity must be given in SI units of J/kgK and W/mK respectively.

If SAMEMATERIAL flag is checked on, both the gears are modeled with same thermal properties. When turned OFF, independent material properties of two gear should be given.

The equation to calculate the flash temperature is stated below,

$$\theta_f = \frac{\sqrt{\pi}}{2} \frac{10^6.\mu.H_s.|V_s|}{B_{M1}\sqrt{V_{r1}} + B_{M2}\sqrt{V_{r2}}} \sqrt{8\kappa \frac{H_s}{1000E_r}}$$

where,

$$B_{M1} = \sqrt{\rho_{M1}.\lambda_{M1}.c_{M1}}$$
$$B_{M2} = \sqrt{\rho_{M2}.\lambda_{M2}.c_{M2}}$$
$$E_r = 2\left(\frac{1-{\nu_1}^2}{E_1} + \frac{1-{\nu_2}^2}{E_2}\right)^{-1}$$

θ_f	-	Flash Temperature, Celsius
μ	-	Coefficient of friction
B_{M1}, B_{M2}	-	Thermal Coefficient of member 1 and member 2
$\lambda_{M1}, \lambda_{M2}$	-	Specific heat conductivity of member 1 and member 2, (Default: 45 W/mK)
c_{M1}, c_{M2}	-	Specific heat capacity of member 1 and member 2,(Default: 440 J/kgK)
ρ_{M1}, ρ_{M2}	-	Density of member 1 and member 2, kg/m^3
E_1, E_2	-	Young's Modulus of member 1 and member 2,N/mm^2
ν_1, ν_2	-	Poisson's Ratio of member 1 and member 2
E_r	-	Reduced Modulus of Elasticity, N/mm^2
H_s	-	Contact pressure, N/mm^2
V_s	-	Sliding velocity, m/s
V_{r1}, V_{r2}	-	Rolling velocity of member 1 and member 2, m/s
κ	-	Normal radius of relative curvature, mm

1.15.2.1 Coefficient of Friction The ISO standard defines the equation for mean coefficient of friction (μ_m) over the entire tooth surface. In *Calyx*, this equation is modified to compute the local coefficient of friction using velocity, curvature, and load intensity at the local discretized points when the $AUTOCOMPUTE\_MU$ flag is turned ON. This local, instantaneous coefficient of friction is computed using the following equation. The factors compensating for the nominal load and dynamic factor is assumed to be 1.0 in the equation. With $AUTOCOMPUTE\_MU$ flag on,USE_OIL_TEMP check box is shown.Turning it ON will allow user to specify independent value for oil inlet temperature. If turned off, then the oil inlet and bulk temperatures are assumed to be the same.

$$\mu = 0.045\left(\frac{K_A.K_V.K_{H\alpha}.K_{H\beta}.K_{H\gamma}.L_I}{V_r.\kappa}\right)^{0.2}\left(10^3.\eta_{\theta M}\right)^{-0.05} X_R X_L$$

where,

$$X_R = 2.2\left(\frac{R_a}{\kappa}\right)^{0.025}$$
$$X_L = 1.0$$
$$K_A, K_V, K_{H\alpha}, K_{H\beta}, K_{H\gamma} = 1.0$$

X_R	-	Roughness Factor
L_I	-	Load Intensity, N/mm
V_r	-	Sum of rolling velocities $(V_{r1} + V_{r2})$, m/s
κ	-	Normal radius of relative curvature, mm
θ_M	-	Oil inlet or bulk temperature, Celsius
$\eta_{\theta M}$	-	Dynamic viscosity at oil inlet temperature or bulk temperature , Ns/m^2
K_A	-	Application factor
K_V	-	Dynamic factor
$K_{H\alpha}$	-	Transverse load factor
$K_{H\beta}$	-	Face load factor
$K_{H\gamma}$	-	Helical load factor
X_L	-	Lubrication factor

1.15.3 Film Thickness

The FILMTHICKNESS pattern component calculates the specific lubricant film thickness at the contact surface. This is also calculated based on the formulation from ISO standard 15144-1 [19]. The equations used to calculate the film thickness are listed below, for detailed explanation of all parameters please refer to the standard. The additional inputs needed for lubricant film thickness calculations are

1. $ALPHA\_SI$,Pressure Viscosity Coefficient of the lubricant at 38 C, m^2/N
2. ETA_40_METRIC,Kinematic Viscosity of the lubricant at 40 C, mm^2/s
3. ETA_100_METRIC,Kinematic Viscosity of the lubricant at 100 C, mm^2/s
4. $DENSITY_15_SI$Density of the lubricant at 15 C, kg/m^3
5. $BULKTEMP$,Bulk Temperature, Celsius
6. RA_SI,Effective arithmetic mean surface roughness of the gears, μm

$$h_s = \frac{h_y}{R_a}$$
$$h_y = 1600.\kappa.{G_M}^{0.6}.{U_Y}^{0.7}.{W_Y}^{-0.13}.{S_Y}^{0.22}$$

where,

h_s	-	Local Specific Film Thickness
h_y	-	Local Film thickness, μm
R_a	-	Effective arithmetic mean roughness value, μm
G_M	-	Material parameter
U_Y	-	Local Velocity parameter
W_Y	-	Local Load parameter
S_Y	-	Local Sliding parameter

1.15.3.1 Material parameter

$$G_M = 10^6.\alpha_{\theta M}.E_r$$
$$\alpha_{\theta m} = \alpha_{38} * \left[1 + 516\left(\frac{1}{\theta_M + 273} - \frac{1}{311}\right)\right]$$

where,

α_{38} - Pressure viscosity coefficient of the lubricant at 38 C, m^2/N

θ_M - Bulk temperature, Celsius

Local velocity parameter

$$U_Y = \eta_{\theta M} \frac{V_r}{2000.E_r.\kappa}$$
$$\eta_{\theta M} = 10^{-6}.\nu_{\theta M}.\rho_{\theta M}$$

where,

$\eta_{\theta M}$ - Dynamic viscosity at bulk temperature, Ns/m^2

V_r - Sum of rolling velocities $(V_{r1} + V_{r2})$, m/s

$\nu_{\theta M}$ - Kinematic viscosity at bulk temperature, mm^2/s

$\rho_{\theta M}$ - Density of lubricant at bulk temperature, kg/m^3

$$\nu_{\theta M} = 10^{10^{A.log(\theta_M+273)+B}} - 0.7$$

where,

$$A = \frac{log[log(\nu_{40} + 0.7)/log(\nu_{100} + 0.7)]}{log(313/373)}$$
$$B = log[log(\nu_{40} + 0.7)] - A.log313$$

ν_{40} - Kinematic viscosity at 40 C, mm^2/s

ν_{100} - Kinematic viscosity at 100 C, mm^2/s

$$\rho_{\theta M} = \rho_{15}.\left[1 - 0.7.\frac{(\theta_M + 273) - 289}{\rho_{15}}\right]$$

ρ_{15} - Density of Lubricant at 15 C, kg/m^3

1.15.3.2 Local load parameter

$$W_Y = \frac{2.\pi.{H_s}^2}{{E_r}^2}$$

where,

H_s - Local Contact Stress, N/mm^2

E_r - Reduced modulus of elasticity, N/mm^2

1.15.3.3 Local sliding parameter

$$S_G = \frac{\alpha_{\theta B}.\eta_{\theta B}}{\alpha_{\theta M}.\eta_{\theta M}}$$
$$\theta_B = \theta_M + \theta_f$$

where,

$\alpha_{\theta B}$ - Pressure viscosity coefficient of the lubricant at contact temperature, m^2/N

$\alpha_{\theta M}$ - Pressure viscosity coefficient of the lubricant at bulk temperature, m^2/N

$\eta_{\theta B}$ - Dynamic viscosity of the lubricant at contact temperature, Ns/m^2

$\eta_{\theta M}$ - Dynamic viscosity of the lubricant at bulk temperature, Ns/m^2

θ_B - Contact temperature, Celsius

1.15.3.4 Safety factor against micropitting The micropitting safety factor according to the ISO standard [19] is given by the equation

$$S_{contact} = \frac{h_{s,min}}{h_{sp}}$$

where,

$h_{s,min} = min(h_s)$	-	Minimum specific lubricant film thickness in the contact area
h_{sp}	-	Permissible specific lubricant film thickness

1.15.4 Wear

The surface wear is another important parameter for studying gear pitting failures. The wear depth is calculated based on Archard's wear equation as given below

$$\frac{dw}{ds} = kP(x,t) \tag{1.1}$$

Substituting $ds = v_s(t)dt$

$$w = k \int P(x,t)v_s(t)dt \tag{1.2}$$

Assuming sliding velocity and contact pressure distribution remains constant at the given location,

$$w = k \int P(t)v_s(t)dt$$
$$w = kv_s \int \frac{P(t)}{\frac{dx}{dt}}dx$$
$$w = k\frac{v_s}{v_r} \int P(x)dx$$

Replacing $\frac{dx}{dt}$ as rolling velocity v_r

$$w = k\frac{v_s}{v_r} \int P(x)dx \tag{1.3}$$

For a parabolic pressure distribution (Figure 1.39),

$$w = \frac{4}{3}k\frac{v_s}{v_r}P.l$$

where,

w	-	Wear per unit cycle, m
k	-	Wear Coefficient, m^2/N (Default: 9.65e-19 m^2/N)
P	-	Contact stress, N/m^2
l	-	Hertzian Semiwidth, m
v_s	-	Sliding velocity, m/s
v_r	-	Rolling velocity, m/s

To calculate the wear , set PATTERNCOMPONENT to WEAR and provide the number of cycles of gear and wear coefficient. The wear coefficient is default to 9.65e-19 based on study done in reference [20]

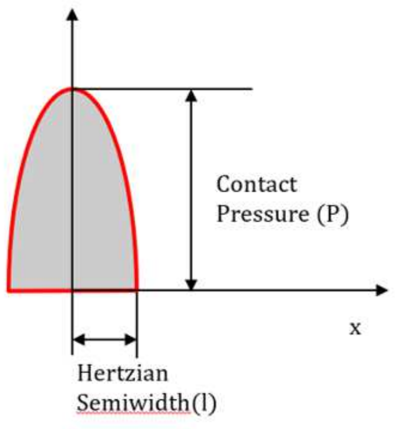

Figure 1.36 Parabolic pressure distribution.

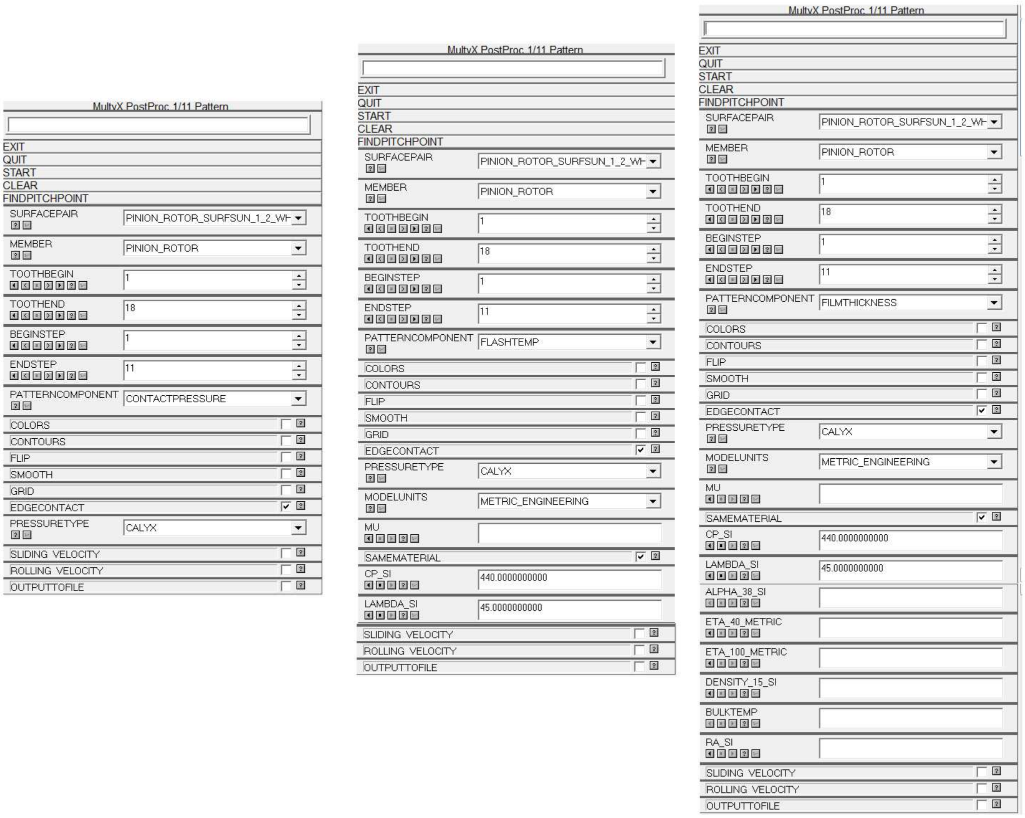

Figure 1.37 The PATTERN menu.

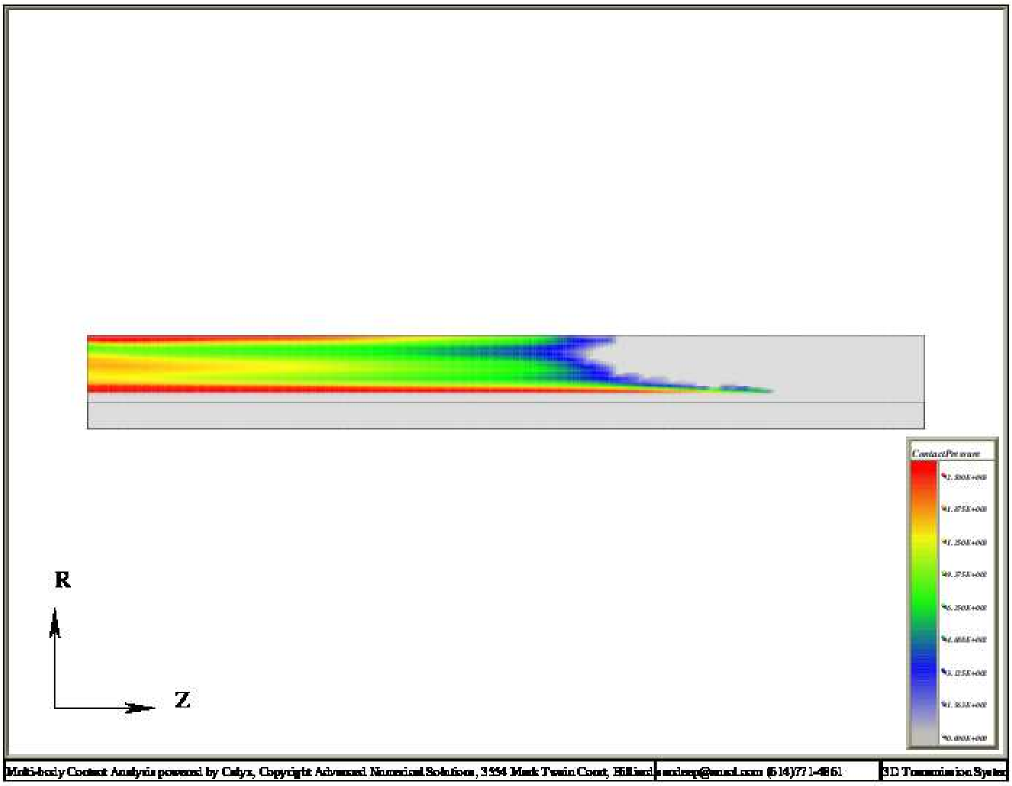

Figure 1.38 The contact pattern generated by the PATTERN menu.

1.15.5 Energy Loss Output and Power Loss Calculation

The flash temperature calculation also outputs energy loss as a result of sliding friction. This calculation uses the Dowson Higginson model for film thickness, and estimates the local coefficient of friction, flash temperature, specific film thickness, and energy dissipated. This method is the recommended method for helical gears, and works equally well for spiral and straight bevel gears, but it has not been validated for hypoid gears. The calculation does not consider energy loss due to losses other than sliding friction (i.e. churning, windage, spin, etc.).

To obtain the energy loss, first select PATTERNCOMPONENT = SPECIFICFILMTHICKNESS in the PATTERN postprocessing menu. Next, the lubricant properties, oil inlet temperature, bulk temperature, and steel bulk properties described above must be specified. Since film thickness is speed dependent, the SPEEDFACTOR input allows the user to modify the input speed without running another analysis. Selecting START will run the postprocessing calculations and the energy loss per tooth engagement will be output in model units to the information window.

If the model units are in N and mm, then the energy loss is output in $N - mm$. Dividing by 1000 will produce the energy loss in Joules. To convert Joules to power in Watts, the energy loss must be multiplied by the mesh frequency in Hz, or $1/(meshcycletime)$

1.15.5.1 Energy Flux The frictional energy loss per unit area at a tooth engagement is defined by the following equation:

$$J_l = \int (\mu P(x,t))ds$$

Substituting, $ds = v_s(t)dt$

$$J_l = \int (\mu P(x,t)) * v_s(t)dt$$

Assuming contact pressure and sliding velocity is independent with time at a given location

$$J_l = \mu * v_s \int \frac{P(x)}{dx/dt} dx$$

Replacing $\frac{ds}{dt}$ as rolling velocity v_r

$$J_l = \mu \frac{v_s}{v_r} \int P(x)dx$$

For a parabolic pressure distribution (Figure 1.39), the closed form solution of the integral is

$$J_l = \frac{4}{3} \mu \frac{v_s}{v_r} P * l$$

where,

J_l - Energy flux (Energy loss per unit area)

μ - Coefficient of friction

v_r - Rolling velocity

v_s - Sliding velocity

P - Contact pressure

l - Hertzian semi-width

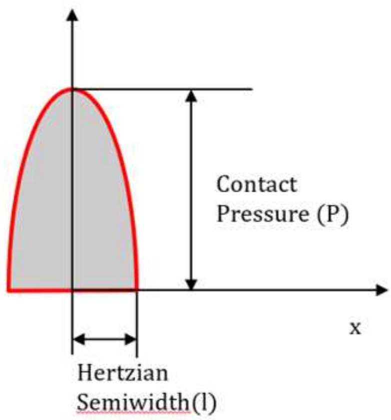

Figure 1.39 Parabolic pressure distribution.

1.16 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 1.3) generates an equilibrim 'audit' of all the forces and moments acting on each body. Figure 1.40 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 if time steps is specified in the BEGINSTEP and ENDSTEP boxes.

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

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

```
Time=-0.4
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.

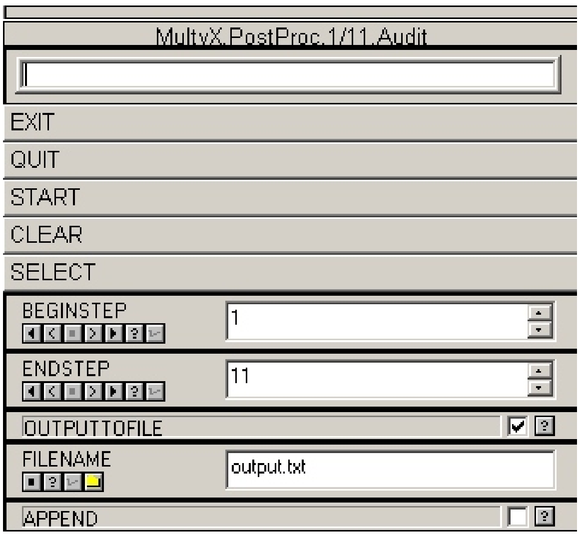

Figure 1.40 The AUDIT menu.

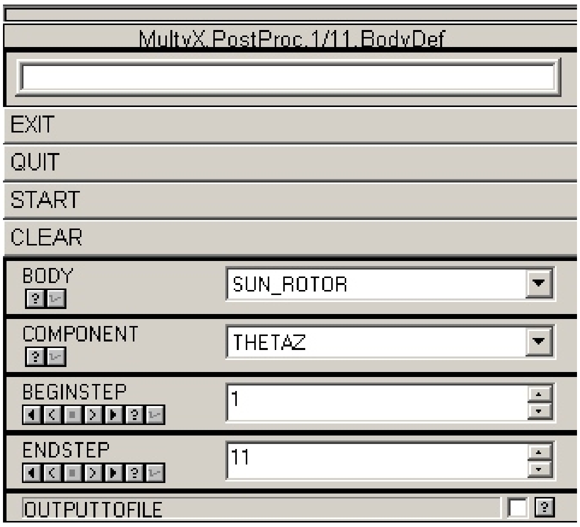

Figure 1.41 The BODYDEFLECTION menu.

1.17 The BODYDEFLECTION command

The BODYDEFLECTION command of the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.41. This menu is used to generate a graph (Figure **??**) 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 $Radians$.

1.17.1 Obtaining Transmission Error with the BODYDEFLECTION command

The transmission error (TE) of a two gear model can be calculated and plotted using the BODYDEFLECTION menu. The high speed member (ROTORTYPE=INPUT) is fixed, so generating a plot for the THETAZ component of the low speed member (ROTORTYPE=OUTPUT) produces the transmission error, as shown in Figure 1.42. The TE peak to peak value is also displayed in the information log window. The transient andor frequency domain data can be output to a file by selecting the OUTPUTTOFILE option along with the OUTPUT_TRANSIENT andor OUTPUT_HARMONICS option(s).

1.18 The BODYREACTION command

The BODYREACTION command of the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.43. This menu is used to generate a graph (Figure 1.44) 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.

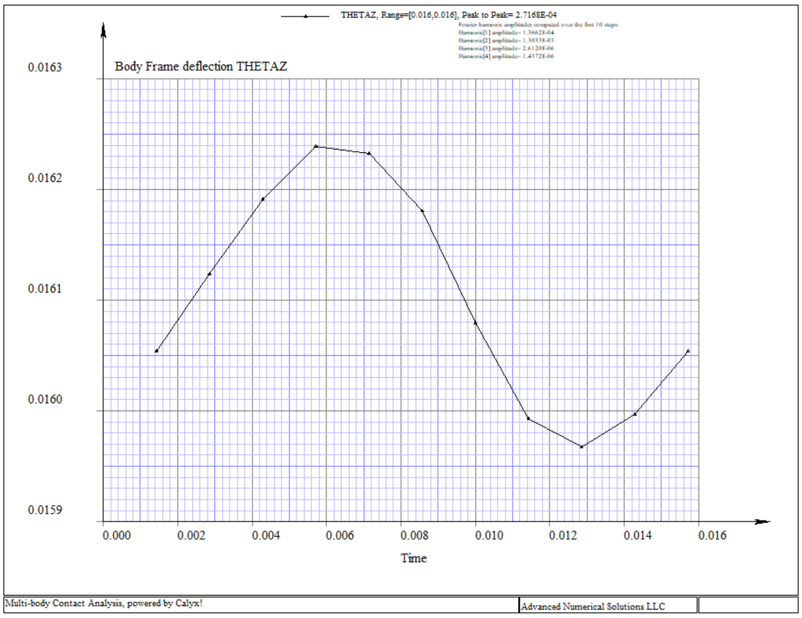

Figure 1.42 The transmission error plot using the BODYDEFLECTION menu.

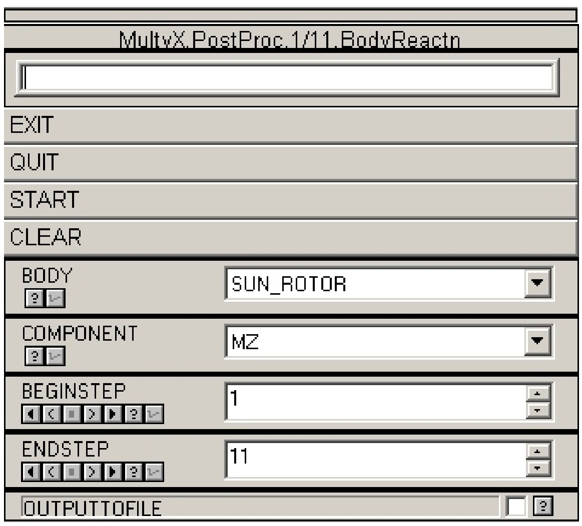

Figure 1.43 The BODYREACTION menu.

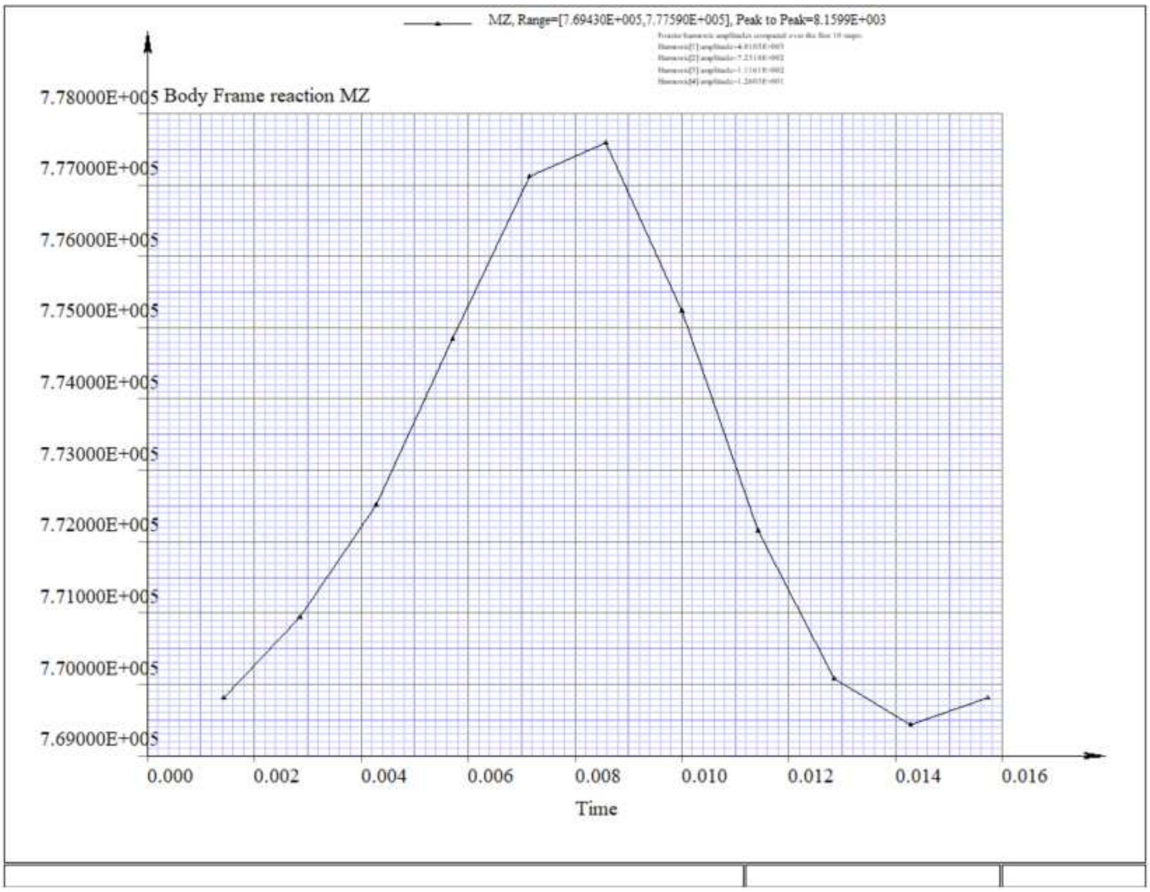

Figure 1.44 The graph generated by the BODYREACTION menu.

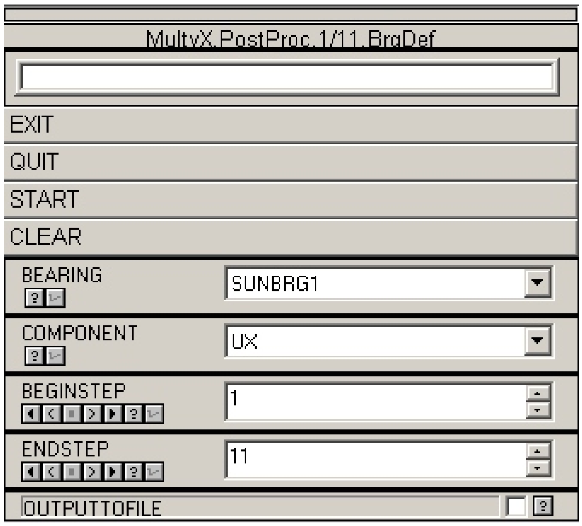

Figure 1.45 The BRGDEFORMN menu.

1.19 The BRGDEFORMN command

The BRGDEFORMN command of the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.45. This menu is used to generate a graph (Figure 1.46) 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 3 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 *Multyx*, 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 $Radians$.

1.20 The BRGREACTION command

The BRGREACTION command of the post-processing menu (Figure 1.3) leads to the menu shown in Figure 1.47. This menu is used to generate a graph (Figure **??**) 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. The forces are measured relative to the reference race in its reference frame. For example, if race 1 is the reference race, the bearing reaction force is the force exerted by race 2 on race 1 measured in the race 1 reference frame.

1.21 The BRGPATTERN command

The BRGPATTERN command of the post-processing menu (Figure 1.3) leads to the menu shown in Figure **??**. This menu is used to generate a contact pattern (Figure 1.50) of the roller length (Z) as a function of angular position T. The contact pattern can be generated for the contact between the roller contact with the INNER or OUTER race and can be generated for any number of time steps. There are three PATTERN-COMPONENTS that can be generated: CONTACTPRESSURE, SUBSURFACESHEAR, and SUBSURFACEVONMISES. The COLORS option generates the pattern in color. CONTOURS draws the pattern with contour lines at a specified DELTAPRESS value. For both the COLORS and CONTOURS options, MINPRESS and MAXPRESS values are also required. The FLIP option flips the orientation of the z-axis in the pattern, and GRID turns on a mesh grid which is overlayed on top of the pattern. PRESSURETYPE can be

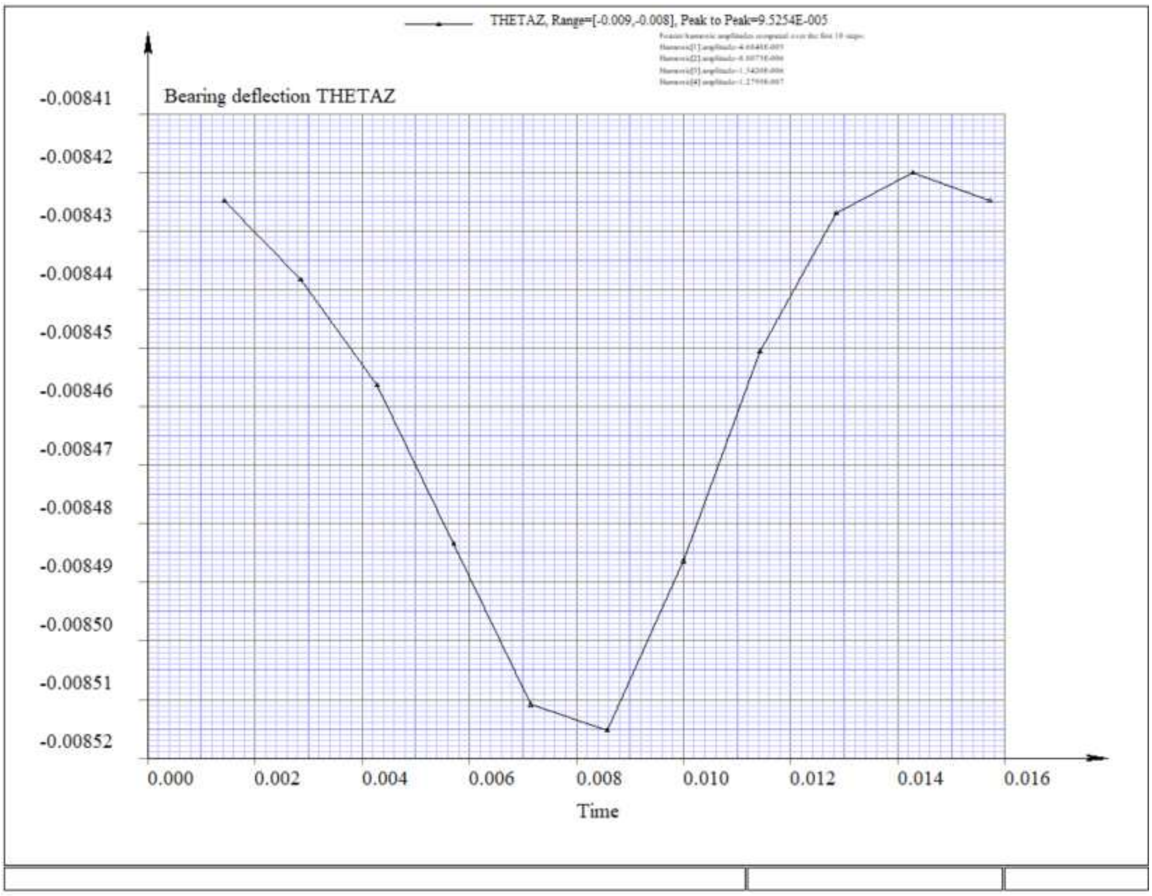

Figure 1.46 The graph generated by the BRGDEFORMN menu.

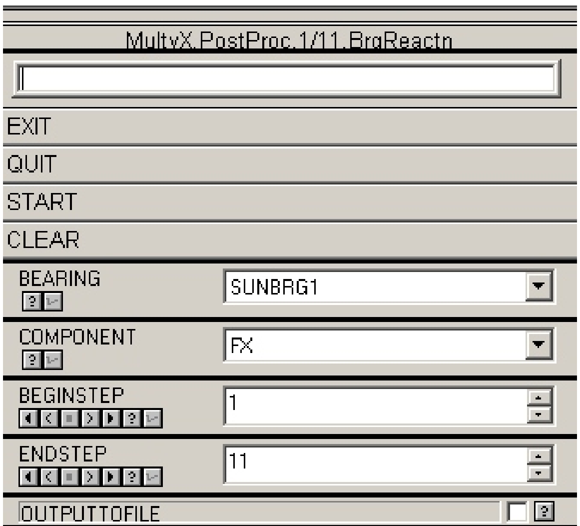

Figure 1.47 The BRGREACTION menu.

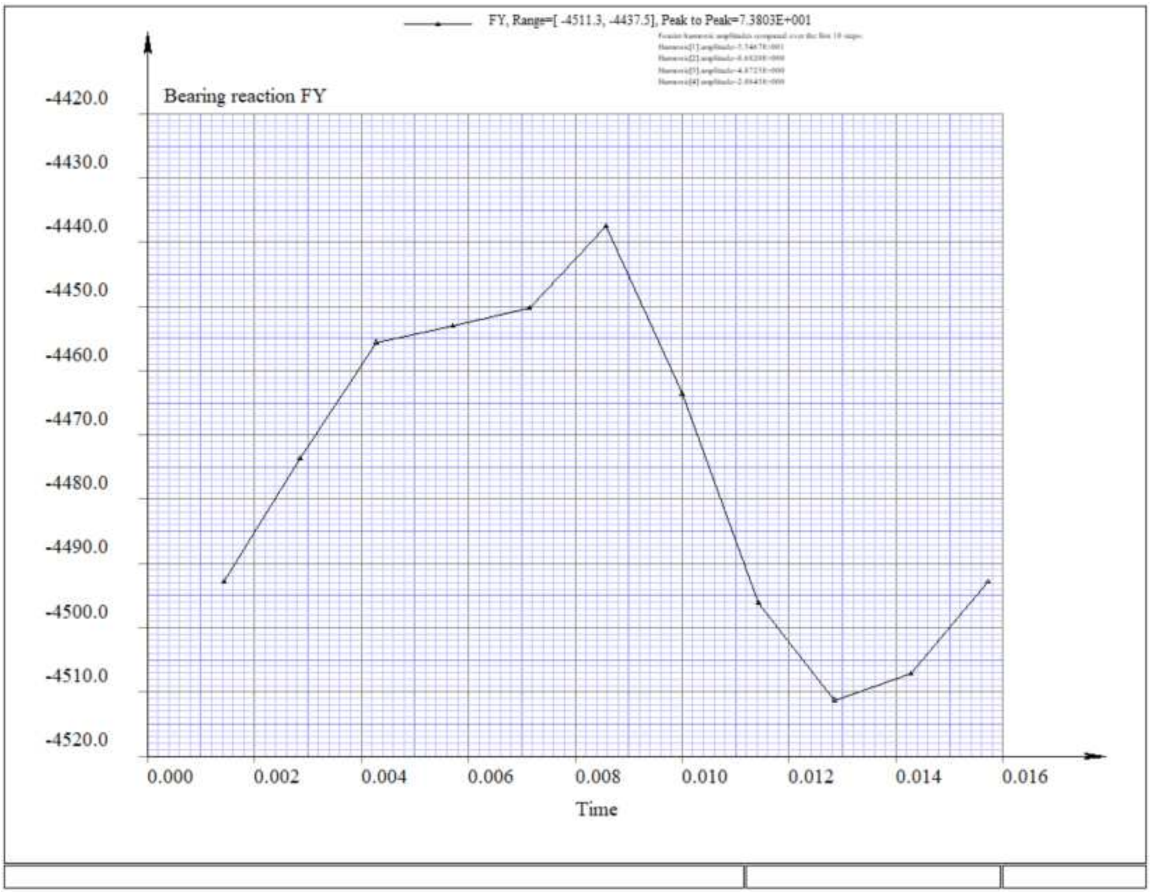

Figure 1.48 The graph generated by the BRGREACTION menu.

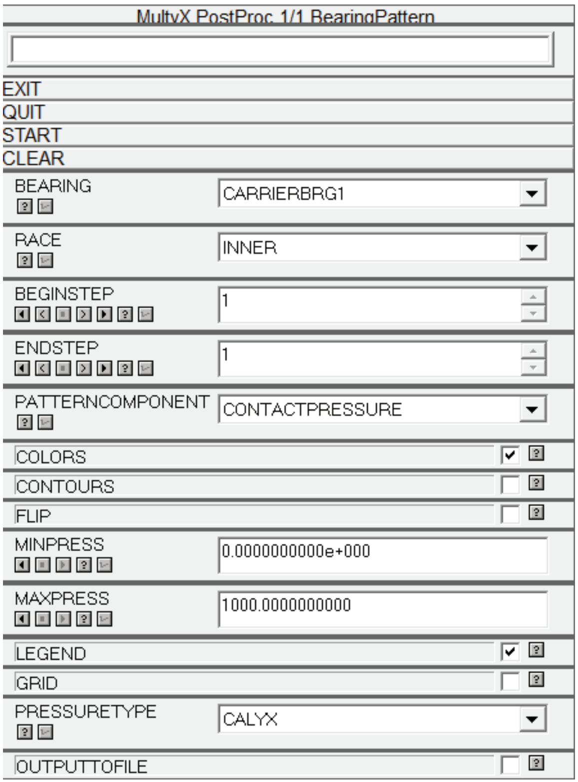

Figure 1.49 The BRGPATTERN menu.

set to CALYX or HERTZ for the contact pressure and sub surface shear components. The OUTPUTTOFILE switch allows the pattern data to be output to a text file specified in the FILENAME input field.

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 *Multyx*, 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.

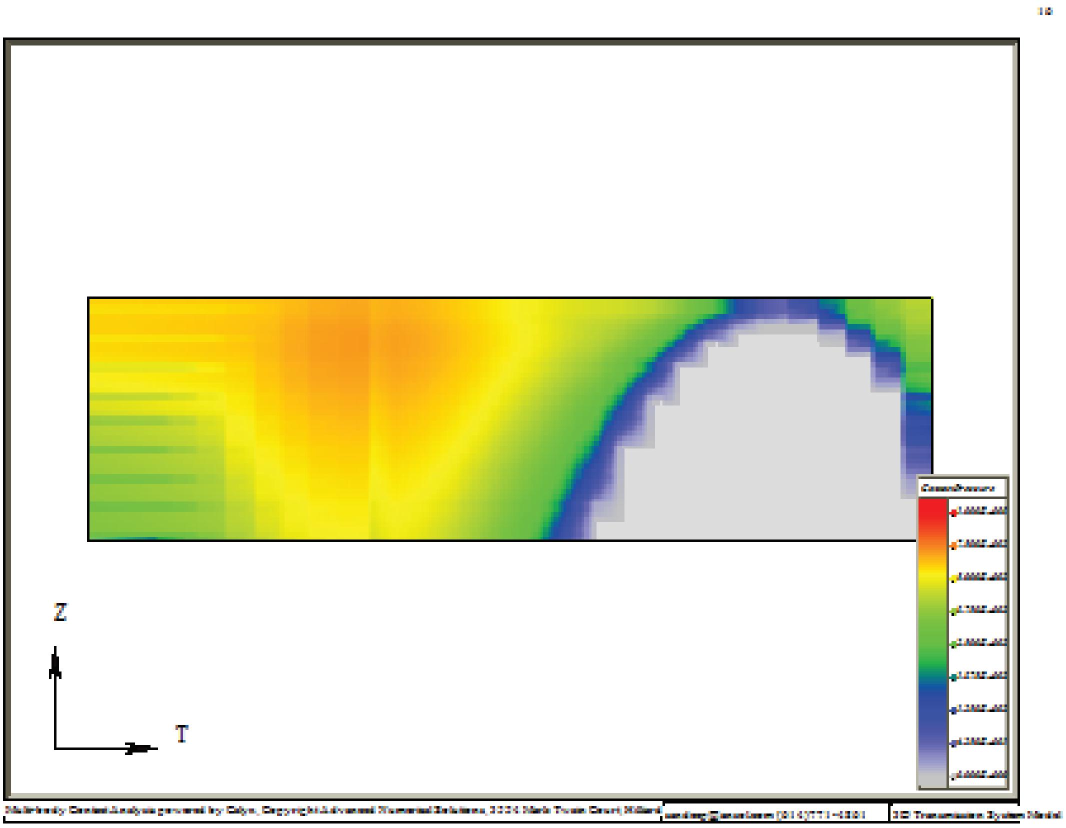

Figure 1.50 The bearing contact pattern.

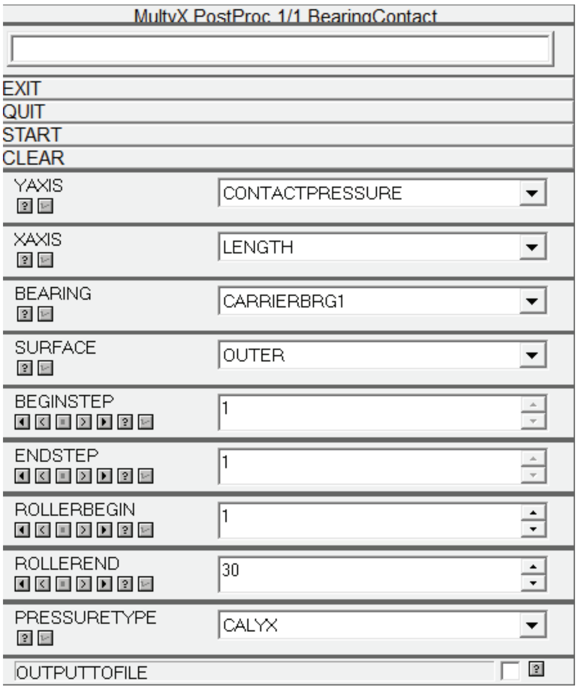

Figure 1.51 The BRGCONTACT menu.

1.22 The BRGCONTACT command

The BRGCONTACT postprocessing command leads to the menu shown in Figure 1.51. This menu can be used to generate a plot of contact pressure, load intensity, sub surface shear stress, or Von Mises sub surface stress vs either length or roller number. For the ’vs length’ plots, the values are taken for each contact grid in the length direction, and the maximum values at each length location are plotted for each roller as shown in Figure 1.52. For the ’vs roller’ plots, the maximum value on each roller is taken and plotted against the roller identification number as shown in Figure 1.53. The roller load plot can only be plotted vs roller ID since the load is a summation of the individual maximum grid loads in the length direction. The roller load vs roller plot is shown in Figure 1.54.

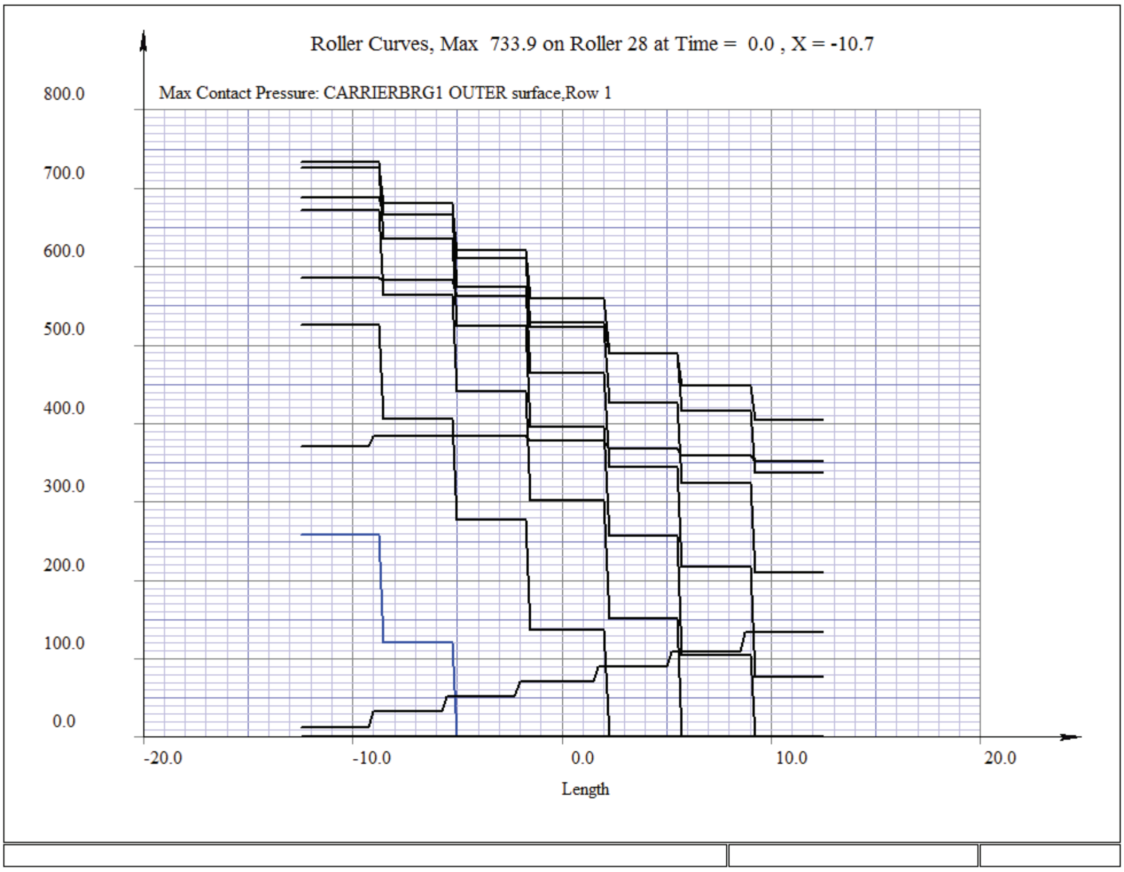

Figure 1.52 Contact pressure vs. length plot.

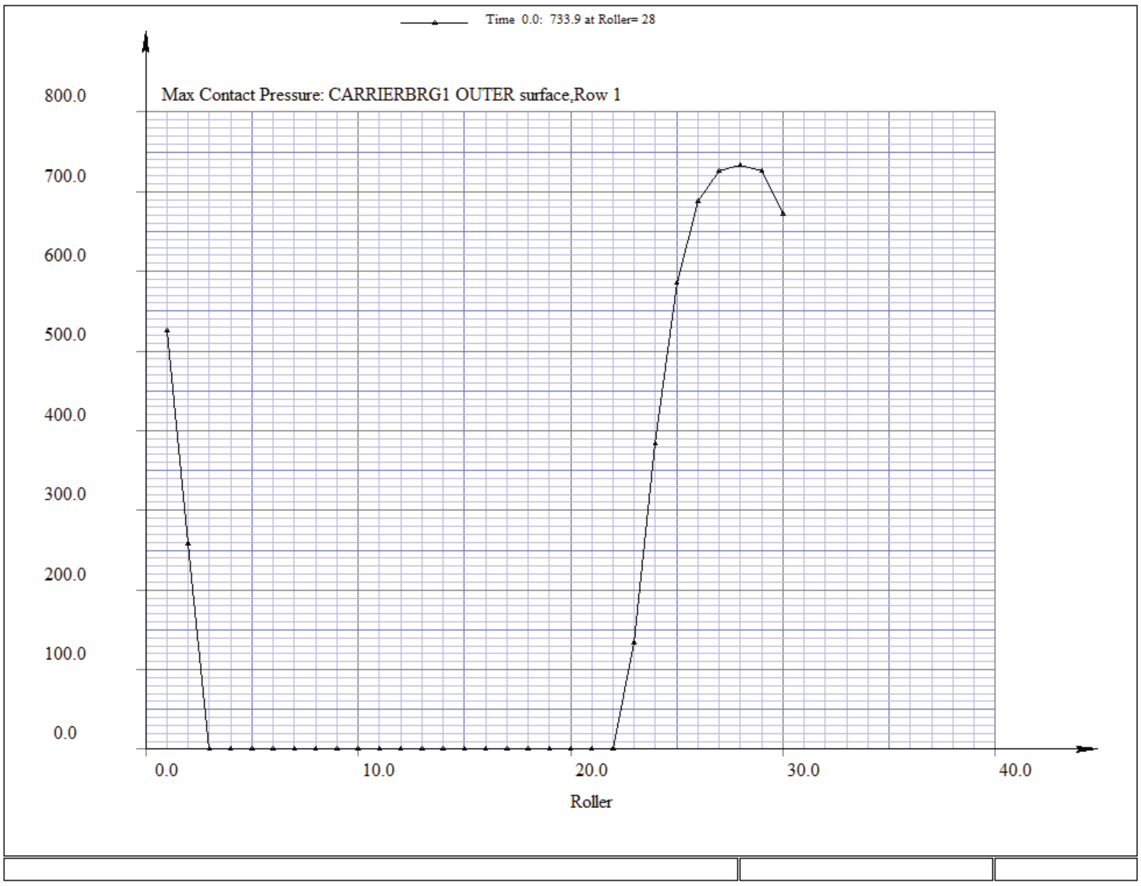

Figure 1.53 Contact pressure vs. roller plot.

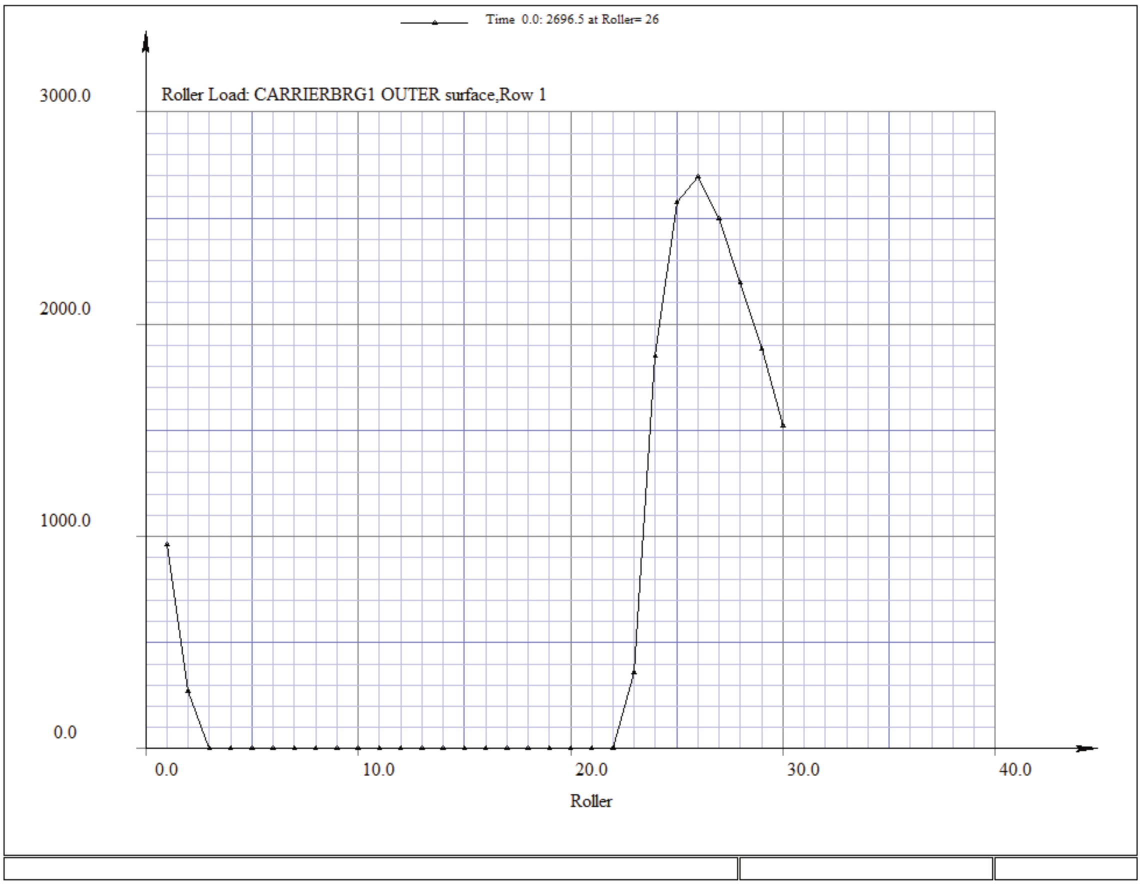

Figure 1.54 Roller load plot.

Table 1.4 The BRGCONTACT menu inputs.

Item	Description
YAXIS	Switch, The Y-axis variable to be plotted. Options are: CONTACTPRESSURE, LOADINTENSITY, SUBSURFACESHEAR, SUBSURFACEVONMISES, and ROLLERLOAD.
XAXIS	Switch, The X-axis variable to be plotted. Options are LENGTH and ROLLER. *Note: LENGTH is not an option when ROLLERLOAD is selected as the Y-axis variable, since ROLLERLOAD plots the total load on each roller.
BEARING	Switch, The name of the bearing for which the plot is desired.
SURFACE	Switch, The roller-race surface to sample data from. Options are INNER or OUTER.
BEGINSTEP	Integer, The time step to begin sampling data.
ENDSTEP	Integer, The time step to end sampling data.
ROLLERBEGIN	Integer, The roller number to begin sampling data.
ROLLEREND	Integer, The roller number to end sampling data.
OUTPUTTOFILE	Boolean, Enables the ability to write the data to a text file.

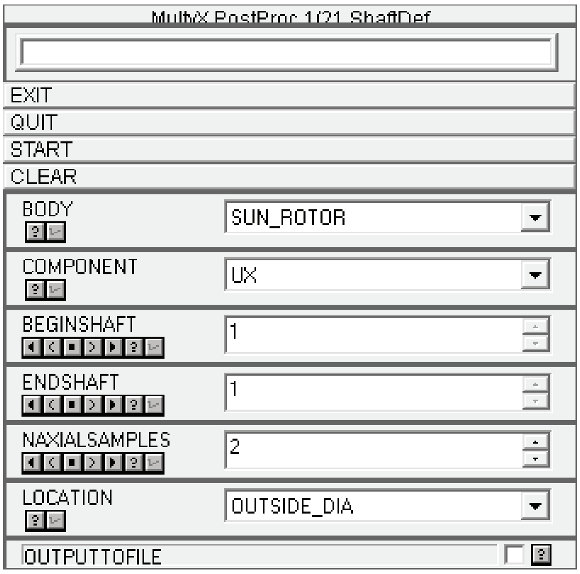

Figure 1.55 Shaft Deformation Menu.

Table 1.5 The SHAFTDEFORMN menu inputs.

Item	Description
BODY	Switch, Selects the body.
COMPONENT	Switch, Selects the component of the shaft deformation.
BEGINSHAFT	Switch, The first shaft of interest.
ENDSHAFT	Switch, The last shaft of interest.
NAXIALSAMPLES	Integer, The number of samples over each finite element in the axial direction.
LOCATION	Switch, The surface location where the sample points are to be located.
OUTPUTTOFILE	Boolean, Enables user to output deformation data to a text file.

1.23 The SHAFTDEFORMN command

The shaft deformation postprocessing menu (Figure 1.55) allows the user to obtain the global x, y, or z components of the shaft deflection as a function of axial position of the shaft along the rotor axis. The deflection values are calculated by sampling a number of points on the desired surface, chosen with the LOCATION input, and averaging the deflection vector component at each axial location. NAXIALSAMPLES defines the number of samples on each finite element in the axial direction. Deformation data is output to a data file by selecting the OUTPUTTOFILE box. Table 1.5 shows the description of each of the menu items.

1.24 The FATIGUE command

Bending fatigue occurs in the fillet region of a gear tooth, and is distinct from the contact fatigue phenomenon observed in the contacting zone. The peak tensile bending stress values occur on the fillet of the loaded side of the gear tooth, while the peak compressive stress occurs on the unloaded fillet

If we search for the maximum s_1 in the profile direction, and over all time instances for individual face cross sections of individual teeth, it is possible to generate a graph of s_1 vs face position, where each curve represents an individual tooth. Each data point on the curve represents the maximum over all time instances and profile positions. Similarly, the instantaneous distribution of minimum principal normal stress s_3 can be plotted vs face position.

The peak values of s_1 and s_3 do not occur at the same place on the fillet. Hence the peak s_1 and s_3 cannot be used simultaneously for calculating the fatigue life. Instead the local values of s_1 and s_3 at every point on the fillet must be used to calculate a local life. The life of the gear or pinion will be the life at the point on the fillet with the shortest life.

To calculate local bending life at any point on the fillet, we look at the time-history of stress at that fillet point. If we run a model for exactly one mesh-cycle, each tooth advances by exactly one tooth pitch over the analysis time range. Since all teeth on a particular gear are identical, we can replicate the entire stress history of a single tooth as it goes all the way around the by splicing together predictions on all individual teeth. This allows us to compute the maximum (over time) of the maximum principal normal stress s_1 and the minimum (over time) of the minimum principal normal stress s_3 at each point on the fillet. Then we compute the local alternating stress s_{alt} and mean stress s_{mean} values:

$$s_{mean} = \frac{\underset{t}{max}(s_1) + \underset{t}{min}(s_3)}{2}$$

$$s_{alt} = \frac{\underset{t}{max}(s_1) - \underset{t}{min}(s_3)}{2}$$

A specimen under purely alternating uni-axial stress amplitude s_{eq} would be equivalent to the state of stress at this fillet point (s_{mean}, s_{alt}) if

$$s_{eq} = \begin{cases} s_{alt} & \text{when } s_{mean} \leq 0 \\ \dfrac{s_{alt}}{1 - s_{mean}/S_{ult}} & \text{otherwise} \end{cases} \tag{1.4}$$

A Haigh diagram (Figure 1.56 for example) is an X-Y plot in which the X axis represents the mean stress s_{mean} and the Y axis represents the alternating stress s_{alt}. The values of (s_{mean}, s_{alt}) at individual fillet points appear as discrete points on the Haigh diagram. For the points lying on the right of the vertical axis, s_{eq} is the intersection point of the vertical axis with a line through (s_{mean}, s_{alt}) and $(S_{ult}, 0)$. S_{ult} is the ultimate tensile strength (See Table 1.6). For points that lie on the left of the vertical axis, s_{eq} is the same as s_{alt}.

Table 1.6 Strength Parameters used in the fatigue calculation.

Ultimate Strength S_{ult}	1585 MPa
Yield Strength S_{yield}	1515 MPa
Endurance Limit S_{end}	700 MPa

The two red lines indicate the points on the fillet that have the highest value of s_{eq}, and the highest value for s_{alt}. The point with the highest s_{eq} is considered the critical point for bending fatigue failure.

The blue line which connects $(S_{ult}, 0)$ to $(0, S_{end})$ demarcates the boundary between points with infinite life, and points with finite life. Any point (s_{mean}, s_{alt}) that lies above the blue line would generate a value for s_{eq} higher than the endurance limit S_{end}, and would fail under fatigue after a finite number of cycles. Any point that lies below the blue line would have infinite life.

2

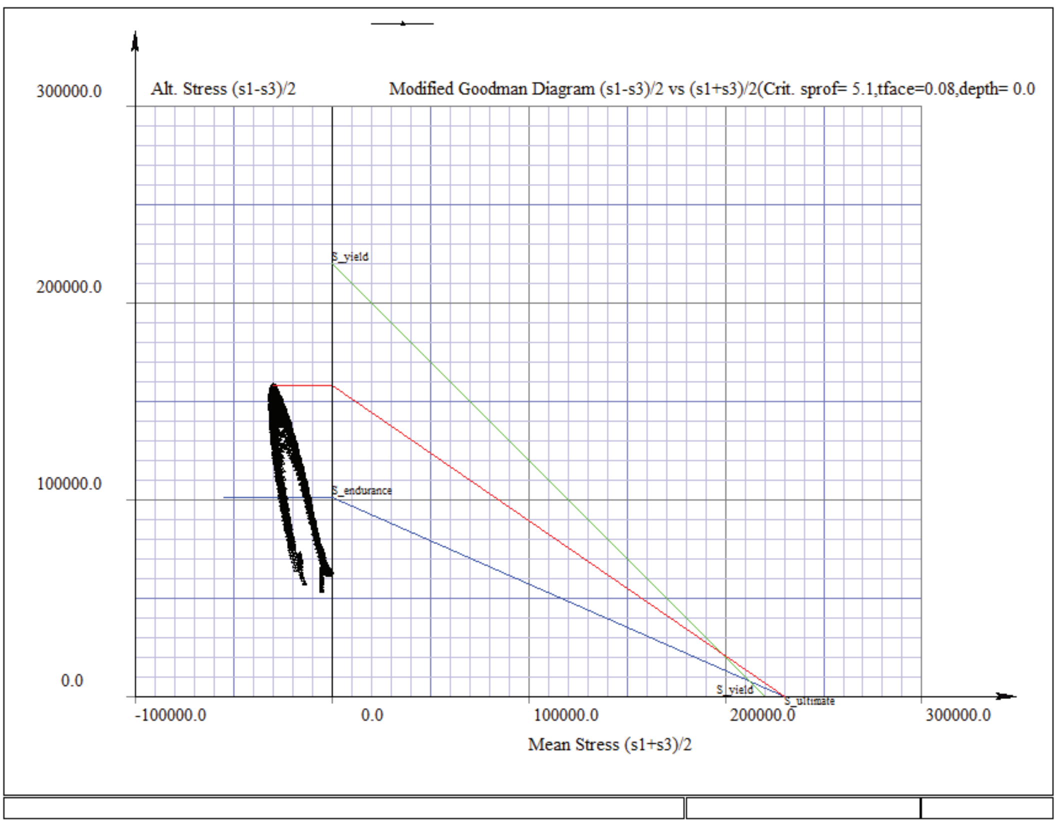

Figure 1.56 A Haigh Diagram.

The green line on the Haigh diagram joins $(S_{yield}, 0)$ with $(0, S_{yield})$, and demarcates the separation between points that undergo tensile yielding ($\max_t(s_1) > S_{yield}$) in the first load cycle, and those that will not. S_{yield} is the tensile yield strength.

The local life N_{life} is related to the local s_{eq} through an $S-N$ curve. Various forms of $S-N$ curve are available, and should be chosen based on the application. For demonstration purposes we use a very simple $S-N$ curve commonly used for steel, based on a text-book stress-life failure theory [21]. This theory assumes that the life of steel is infinite when $s_{eq} < S_{end}$, life at at $s_{eq} = S_{end}$ is $N_{life} = 10^6$ load cycles, that the life at $s_{eq} = S_{1000} \cong 0.9S_{ult}$ is $N_{life} = 10^3$ load cycles, and that in between these two points, the $S-N$ curve is a straight line when the life axis is in log scale, as shown in Figure 1.57.

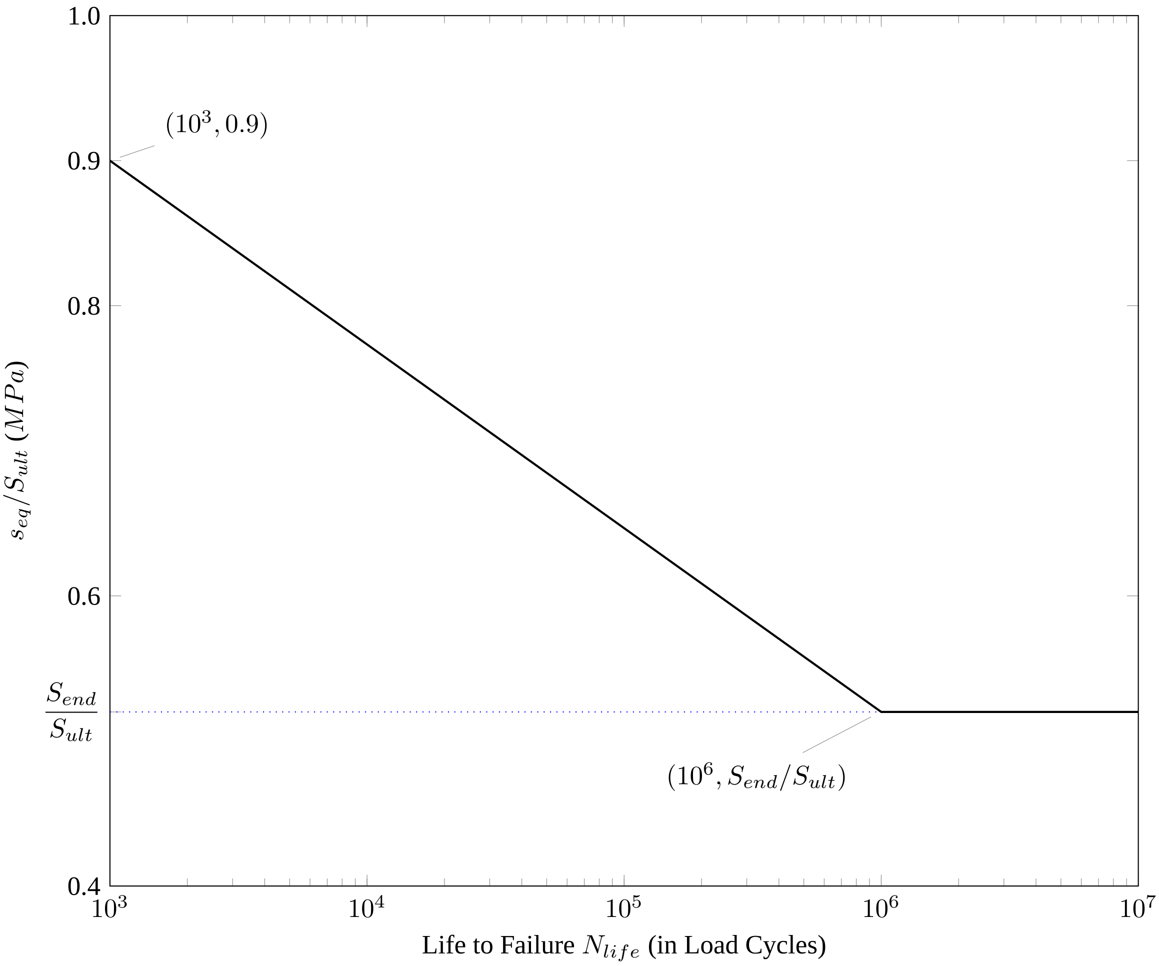

Figure 1.57 An S-N curve commonly used for steel [21].

This $S-N$ curve can be represented by

$$N_{life} = \begin{cases} \infty & \text{when } s_{eq} < S_{end} \\ 10^{-C/b} s_{eq}^{1/b} & \text{when } s_{eq} > S_{end}) \end{cases} \tag{1.5}$$

or

$$s_{eq} = 10^c N_{life}^b \text{when } 10^3 < N_{life} < 10^6 \tag{1.6}$$

where the constants b and C are calculated to generate a straight line on the graph:

$$b = -\frac{1}{3} log_{10} \frac{S_{1000}}{s_{end}}$$

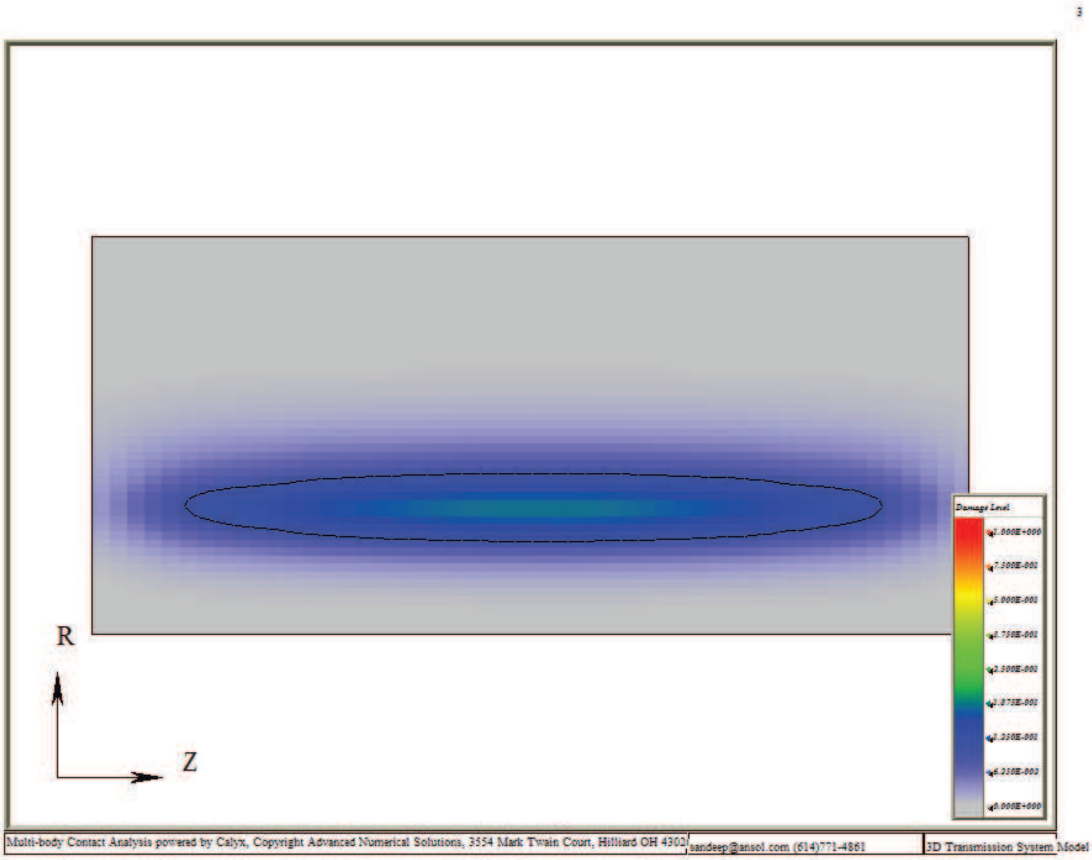

Figure 1.58 Fatigue damage contour plot.

$$C = log_{10} \frac{(S_{1000})^2}{s_{end}}$$

The local damage fraction D at each point on the fillet after N load cycles is defined as the fraction:

$$D = N/N_{life} \tag{1.7}$$

where N_{life} is the predicted local life at that fillet point. Figure 1.58 shows an example of damage contour plot on the fillet region of a gear tooth. In this case, since $D < 1.0$ at all points on the surface, failure is not predicted to occur when subjected to only this loading cycle.

These damage distribution maps are easily used to compute cumulative damage when the pinion is subjected to varying load conditions. We would simply run a separate analysis for each loading condition i, and obtain damage distribution plots for D_i using the process described above. Then, using Miner's rule, we simply add the damage distributions to get the cumulative damage distribution:

$$D = \sum_i D_i \tag{1.8}$$

1.24.1 Max Damage Criterion

The FATIGUE menu for the MAX_DAMAGE CRITERION is show in Figure 1.59. For the max damage criterion we sample the fillet stress in the direction normal to the tooth cross section at each critical s,t location for each tooth in the range BEGINTOOTH to ENDTOOTH. The values are sampled over the time step range

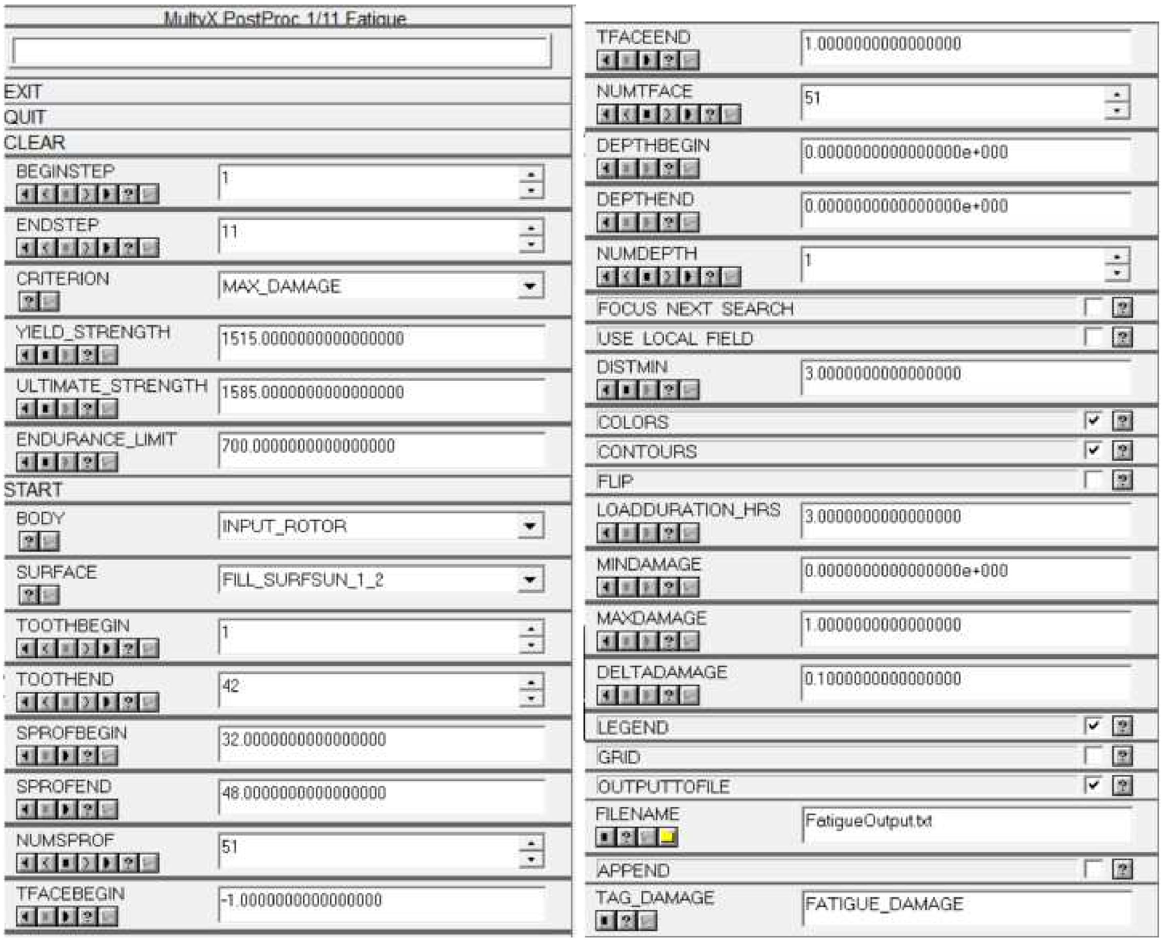

Figure 1.59 The FATIGUE menu with MAX_DAMAGE criteron.

BEGINSTEP to ENDSTEP and then stitched together by time shifting the samples for each successive tooth by the mesh cycle time. If the time step range is equivalent to the mesh cycle time, then the extended time period is:

$$T_{ext} = (ENDTOOTH - BEGINTOOTH + 1) * t_{MeshCycle}.$$

This stress signal S_{uu}, shown in Figure 1.60, is then run through a rainflow counter to count the reversals in the signal. To ensure proper rainflow counting, check that the DISTMIN parameter is set to roughly 1/4 of the tooth height. If not, points near the contact zone may be sampled resulting in large spikes the the signal that will affect the rainflow counting.

Figure 1.61 shows what the rainflow chart might look like. For each reversal, the fraction of cycles, S_{mean}, and S_{alt} are calculated from the signal and used to compute S_eq using Equation 1.9. The fatigue life for each reversal can then be obtained from the S-N curve defined by the user inputs for YIELD_STRENGTH, ULTIMATE_STRENGTH, and ENDURANCE_LIMIT.

$$s_{eq} = \begin{cases} s_{alt} & \text{when } s_{mean} \leq 0 \\ \dfrac{s_{alt}}{1 - s_{mean}/S_{ult}} & \text{otherwise} \end{cases} \tag{1.9}$$

The damage for each reversal is then calculated as $d_i = \frac{c_i}{N_i}$, where c_i is the fraction of cycles for the reversal and N_i is the reversal fatigue life. The cumulative damage for the load case is then

$$D_{LoadCase} = \sum_{i=1}^{n_{reversals}} d_i * \frac{T_{LoadCase}}{T_{Ext}}$$

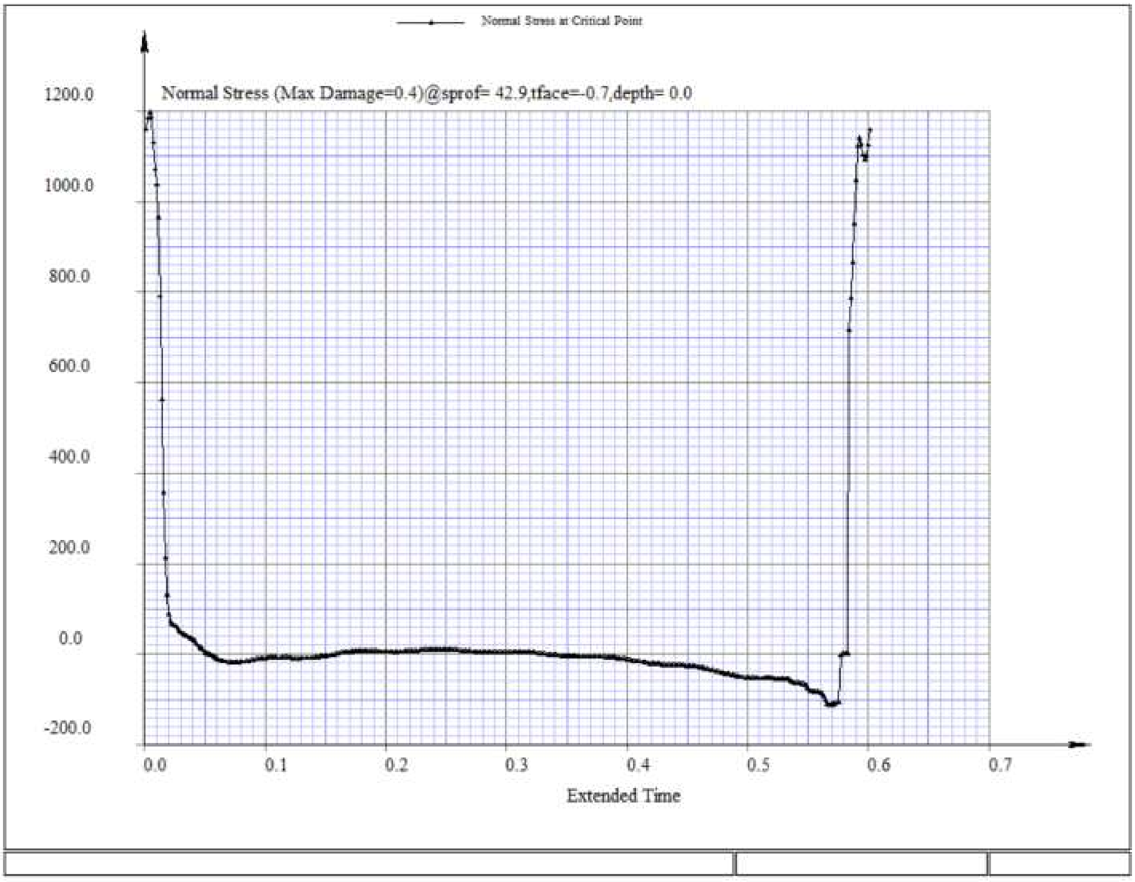

Figure 1.60 The normal stress graph over extended time.

Occurrence i	Fractions of Cycles c_i	Mean stress for reversal i $S_{i,mean}$	Alternating stress for reversal i $S_{i,alt}$
1	0.5	546	0.01
2	0.5	345	0.12
3	1.0	64	1.2
:	:	:	:
:	:	:	:
145	0.5	423.6	354.9

Figure 1.61 Rainflow counting data.

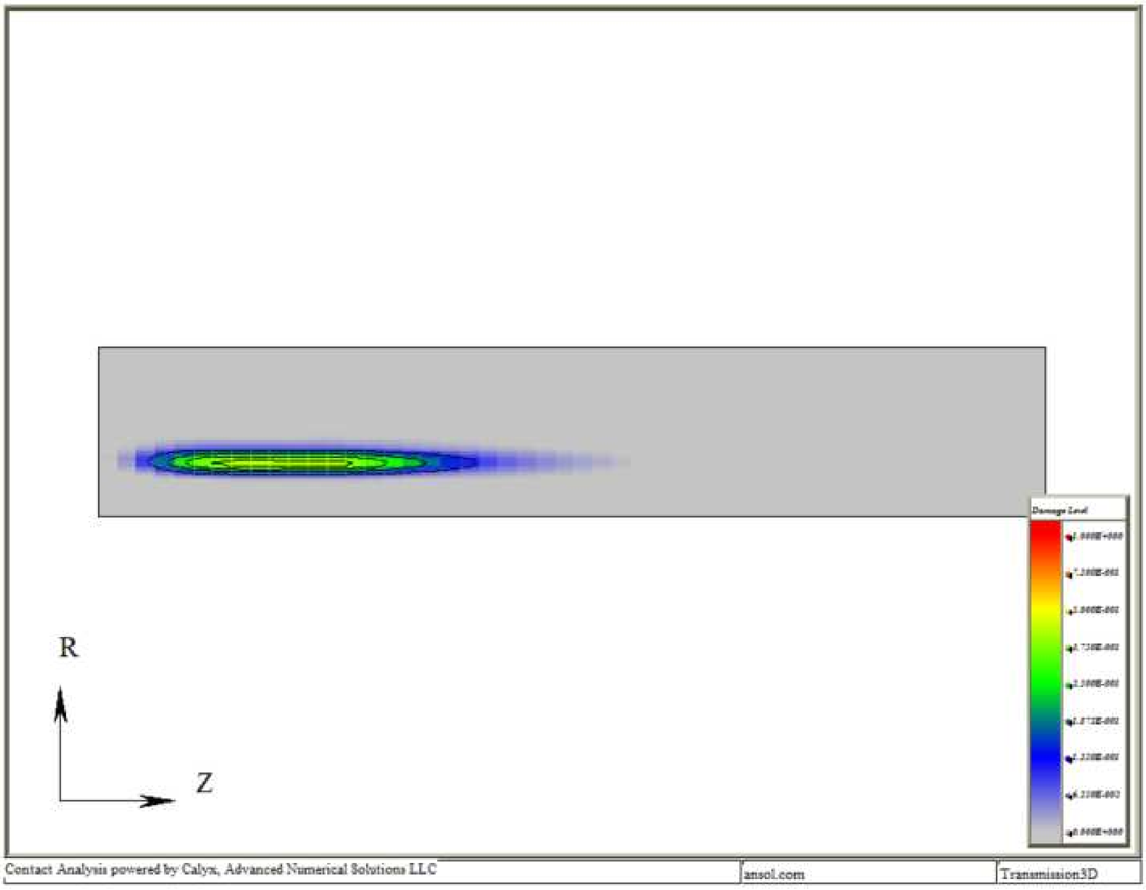

Figure 1.62 Fatigue damage contour plot.

where $T_{LoadCase}$ is equal to the user input LOADDURATION_HRS. A damage contour plot, similar to the one shown in Figure 1.62, is generated in the guide output display pane.

A results file can also be written by selecting the OUTPUTTOFILE option and entering an OUTPUTFILENAME. This file contains the S_{uu} vs T_{ext} data, rainflow data, and surface damage and life contour maps.

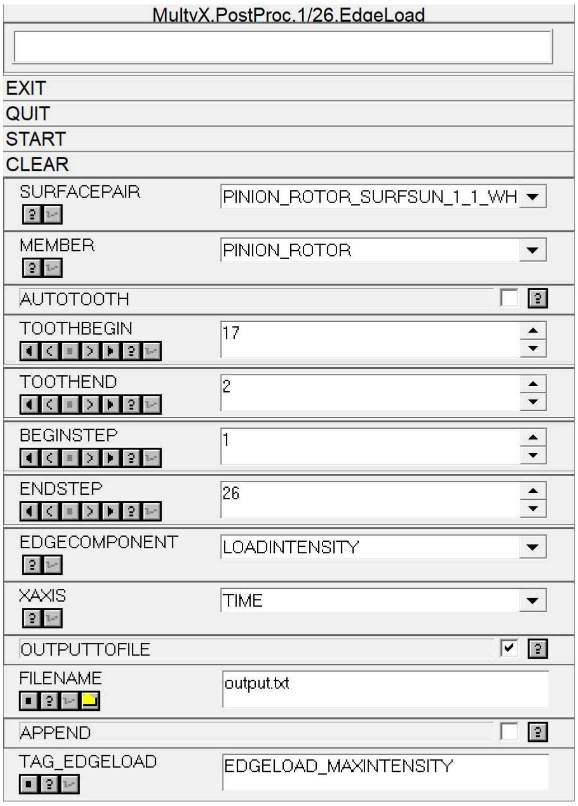

Figure 1.63 The EGDELOAD post-processing menu.

1.25 The EDGELOAD Command

The EDGELOAD post-processing menu produces the load or load intensity over time for any contact deemed edge contact. Within the EDGELOAD menu, shown in Figure 1.63, the user selects the SURFACEPAIR and MEMBER of interest, where the member is the gear containing the tooth edge of interest. Selecting AUTOTOOTH will automatically determine the teeth in contact, otherwise the tooth range is specified by the BEGINTOOTH and ENDTOOTH fields. The time history data is generated for the time range determined by the BEGINSTEP and ENDSTEP inputs. The EDGECOMPONENT drop-down selects LOAD or LOAD-INTENSITY component, and the XAXIS drop-down selects TIME or TFACE as the horizontal axis for the plotdata generated. Turning on the OUTPUTTOFILE box and specifying an OUTPUTFILENAME writes the data to a text file. APPEND allows the file to be appended with each execution of the EDGELOAD menu so that data for multiple pairsmembers can be written to the same file.

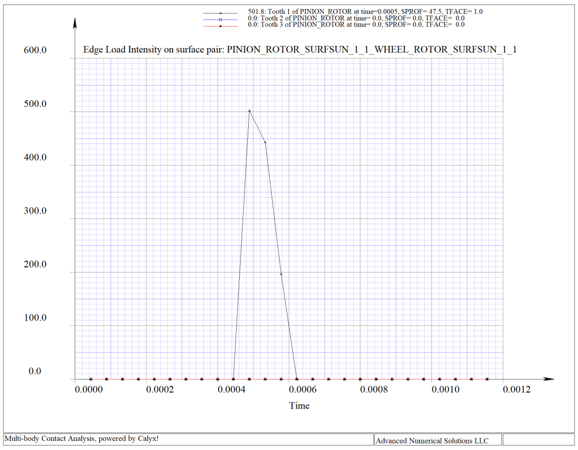

Figure 1.64 Edge load intensity time history.

1.26 The Helical Misaligment Output File

Successful completion of a model analysis results in the creation of the `HELICALMISALIGNMENT.DAT` file within the calyxtmp/ subdirectory of the model working directory. This file contains the misalignment data for each gear pair in the model at each analysis time instance. The format of the file is:

$$Time \quad \Theta_1 \quad ls_1 \quad \Theta_2 \quad ls_2 \quad \Theta_{31} \quad ls_{31} \quad \Theta_{32} \quad ls_{32} \quad \Theta_{33} \quad ls_{33} \quad ...$$

where,

Time	-	Anaysis step time in seconds
Θ_1	-	Misalignment of pair 1
ls_1	-	Lead Slope of pair 1
Θ_2	-	Misalignment of pair 2
ls_2	-	Lead Slope of pair 2
Θ_{31}	-	Misalignment of pair 3, group 1 (This occurs for SUN-PINION,RING-PINION,PINION-PINION pairs)
ls_{31}	-	Lead Slope of pair 3, group 1
Θ_{32}	-	Misalignment of pair 3, group 2
ls_{32}	-	LeadSlope of pair 3, group 2

$\vdots$

The sign convention for the misalignment and lead slope values is positive shifts contact towards the zeta=+1 side of the tooth, and a negative value shifts contact toward the zeta=-1 side. The lead slope values are given per unit facewidth.

1.26.1 Misalignment Application as Lead Slope Correction

The misalignment can be applied as a lead slope correction to one of the gears of the gear pair of interest by multiplying the the misalignment per unit facewidth given in the HELICALMISALIGNMENT.DAT file by the facewidth of the gear. Note the sign of the misalignment provided and cautiously apply this correction such that it will shift contact in the opposite direction.

1.26.2 Misalignment Application as Rotor Misalignment

The misalignment may also be applied to one of the two gears as a rotor misalignment. Note that application of the misalignment using this method will apply the misalignment to all existing rotor components, which may not be desired. Application of the misalignment using this method requires the following transformations from the line normal to the line of action to the x and y axis of the rotor (assuming z is the axis of rotation).

$$\Theta_y = \Theta_{File} * cos(\phi)$$

$$\Theta_x = \Theta_{File} * sin(\phi)$$

1.27 The Backlash Output File

Transmission3D calculates and outputs the backlash of a gear tooth pair during an analysis if the BACK-CONTACT box is selected in the PAIRS menu. Figure 1.66 shows the input parameters when this box is selected. The backlash results data is written to a file named `BACKLASH.DAT` located in the `calyxtmp` subdirectory of the model directory. The first column of the file shows the time data, while the 2nd and 3rd columns show the angular (radians) and linear backlash, respectively.

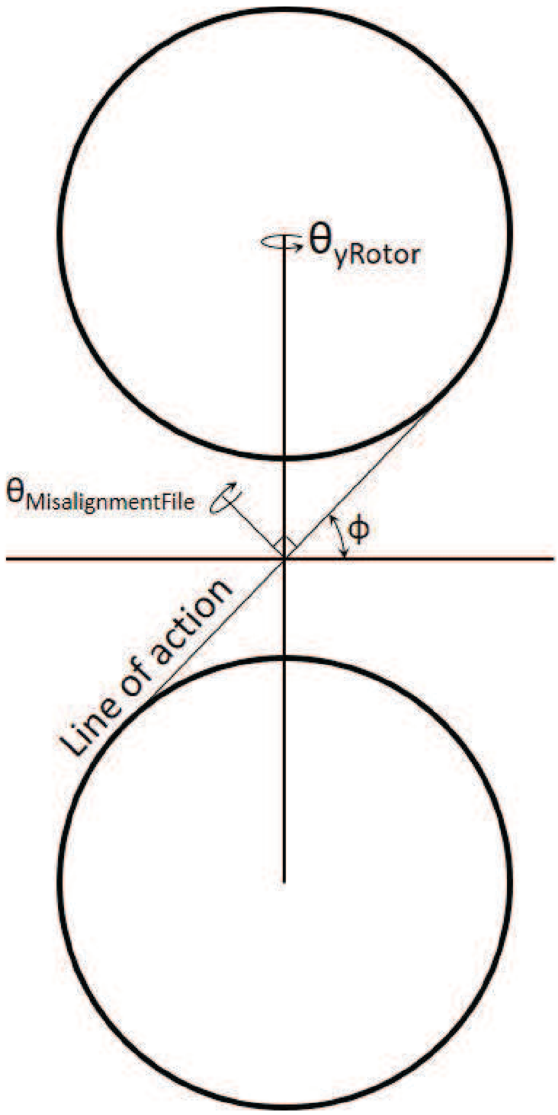

Figure 1.65 Rotor misalignment schematic.

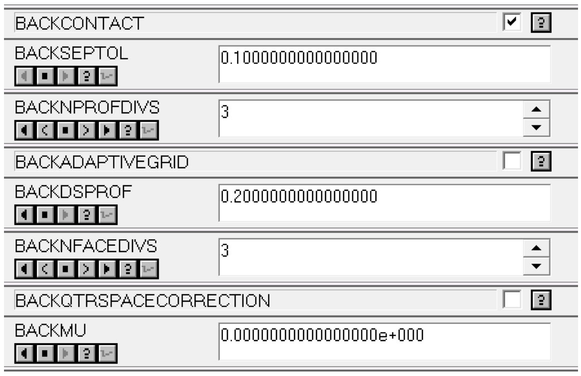

Figure 1.66 The BACKCONTACT input parameters from the T3D PAIRS menu.

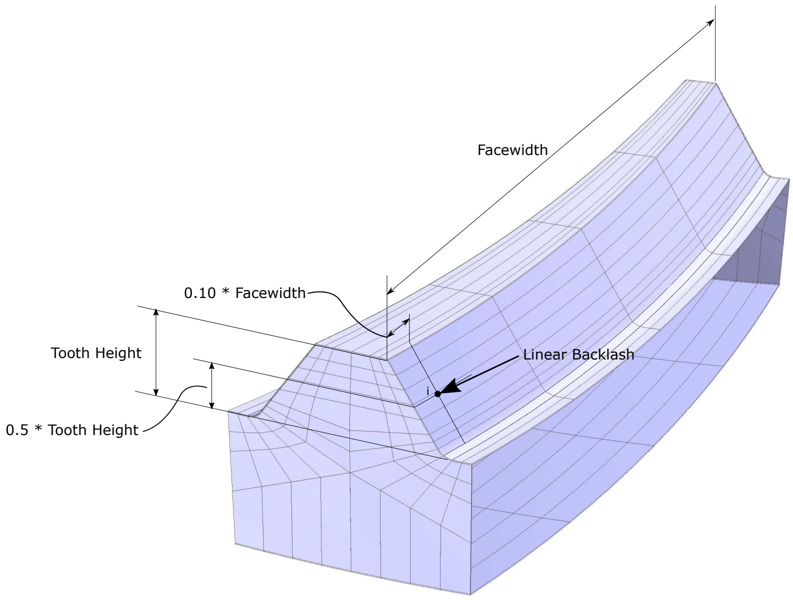

Figure 1.67 Hypoid gear backlash point of measurement.

Angular backlash is determined by holding the pinion at any particular time step and rotating the gear until contact is made on the back side of the tooth. The angular rotation, in radians, is the angular backlash. The angular backlash is different for each time step becuase the pinion position changes at each step.

Linear backlash must be measured following a process that is recommended by Gleason. Figure 1.67 shows the point, i, at which the backlash is measured using a dial indicator normal to the tooth surface. The point is located at a distance of 10% of the face width from the heel end of the gear on the convex side of the tooth. It is at the midpoint of the gear tooth in the profile direction. If the pinion is fixed and the gear is rotated until the back side makes contact, linear backlash is the dial indicator measurement at point i.

1.28 The STEPCREATE and STEPCONVERT Programs

The STEPCREATE and STEPCONVERT programs are utility programs that can convert gears between CAD and *Transmission3D*. The STEPCREATE program creates a 3D CAD gear solid from a gear in a

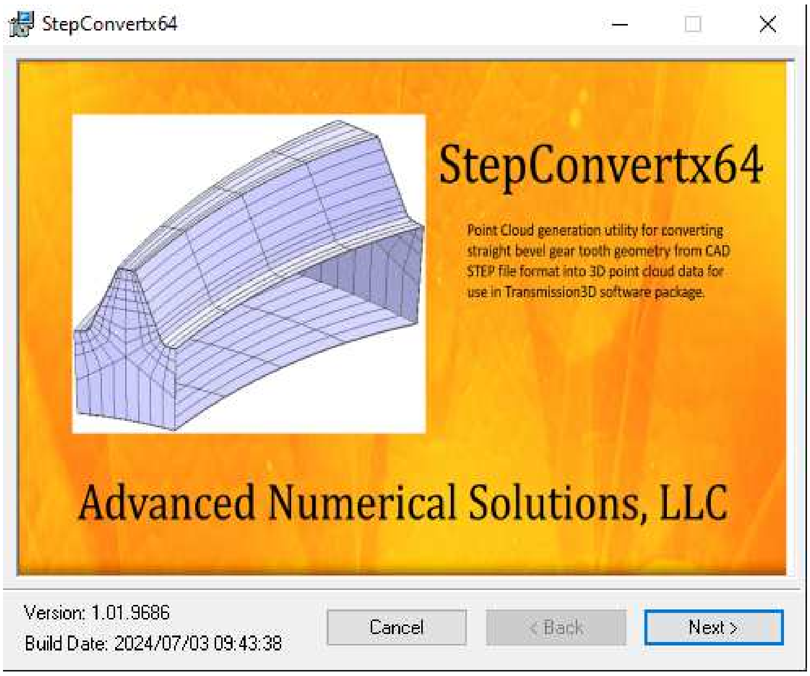

Figure 1.68 STEPCREATE/STEPCONVERT Installation.

Transmission3D model. The STEPCONVERT program converts a CAD gear tooth slot surface into a 3D point cloud data structure that can be imported into *Transmission3D* to model the gear.

The two programs are installed together with a single installer. The installer can be downloaded from the *Ansol* tech-support website (`techsupoort.ansol.com`). To install the software, simply open the MSI file and follow the installation instructions from the main install menu shown in Figure 1.68. Note the installation path chosen during this process as this will be used to execute the program from the command prompt.

1.28.1 The STEPCREATE Program

The StepCreate program converts a *Transmission3D* sun, helical pinion, ring, bevel, bevel pinion, or hypoid into CAD format. To run the program, open a command prompt, change to the model directory to the directory of the desired *Transmission3D* model, and type the installation path of the StepCreate.exe. The model must first be generated in *Transmission3D* in order to populate the system.cfg file. The user inputs are shown in Figure 1.69. Simply enter the desired rotor, carrier (if applicable), and gear id, and the program will automatically perform the conversion. Some more intricate gear geometries may require using advanced settings. These settings can adjust the number of face/profile points, or the offset tolerance, which is the amount the tooth slot surface is extended at all edges to intersect the gear blank solid. Figure 1.70 shows the output log after a successful conversion. The exported tooth slot in *Hypermesh* is shown in Figure 1.71.

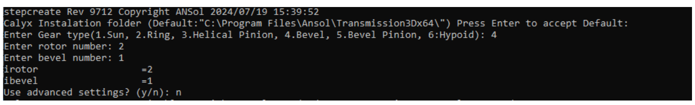

Figure 1.69 The STEPCREATE program command prompt inputs.

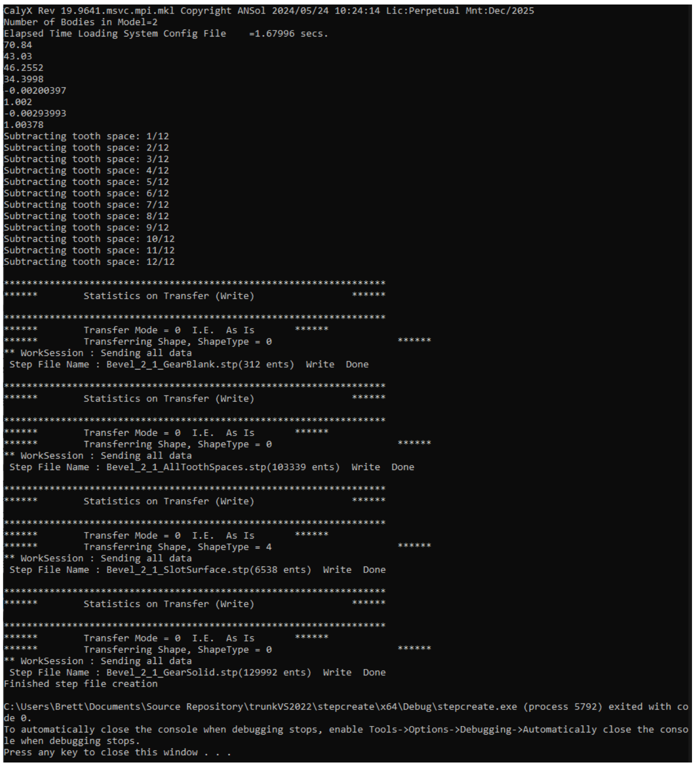

Figure 1.70 The STEPCREATE program command prompt output log.

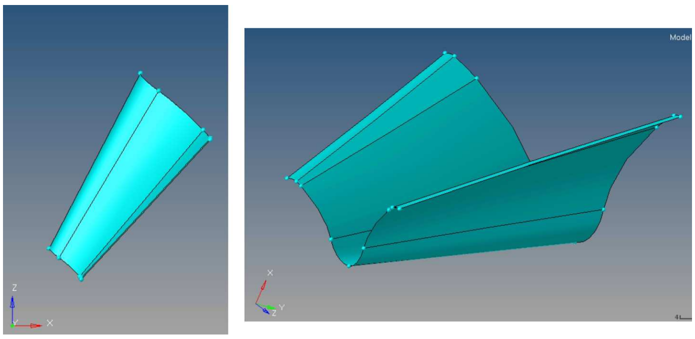

Figure 1.71 The exported tooth slot.

1.28.2 The STEPCONVERT Program

The STEPCONVERT program converts a CAD file of a tooth slot surface in STEP(`.stp`) or IGES(`.igs`) format into a pointcloud text file that can be used in *Transmission3D* to generate a bevel, bevel pinion, or hypoid gear. The required axis orientation of the file to be converted is shown in Figure 1.72. To run the program, open up a command prompt and switch to the directory containing the CAD file and execute the StepConvert.exe program by typing its full installation path. The program will ask for a number of inputs related to the gear tooth geometry. The output file label is used to label the point cloud files generated by the program. After entering the required inputs the program will executed and display an output log, if successful. Two files are generated in the working folder: a text file containing the point cloud coordinate data, and a multyx script file that can be executed within *Transmission3D* to automatically fill the POINTCLOUD menu. The StepConvert program inputs and output log, as well as the files generated are shown in Figure 1.73. To execute the script, simply click the EXECUTESCRIPT button once inside the POINTCLOUD menu, shown in Figure 1.74, and select the *Multyx* script generated by the StepConvert program. The last row of data must be removed, since it contains only a partial row of data, so in this example, we change the 25 slices to 24 in the POINTCLOUD menu after the script was executed. The generated point cloud and its axis orientation are shown in Figures 1.75 and 1.76, respectively.

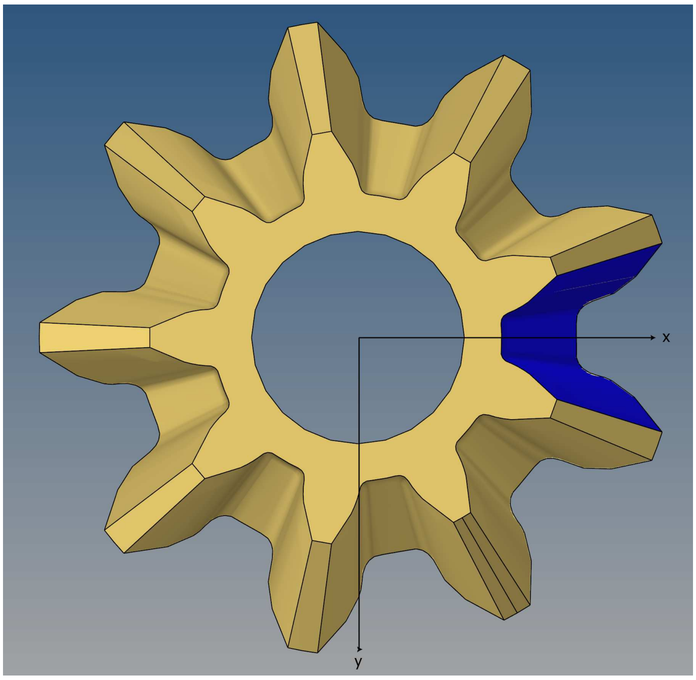

Figure 1.72 The required axis orientation.

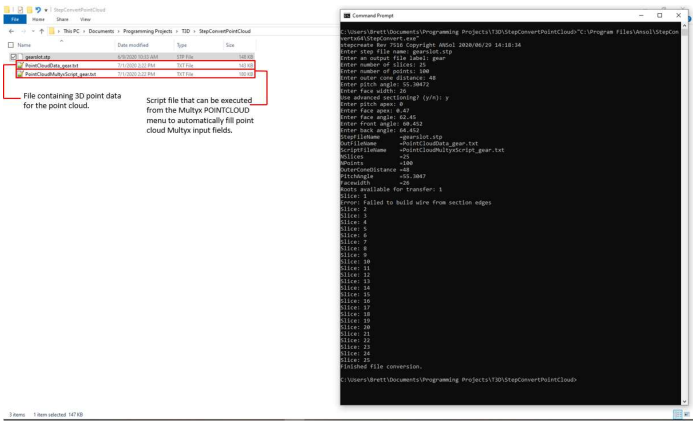

Figure 1.73 The step convert program execution and output.

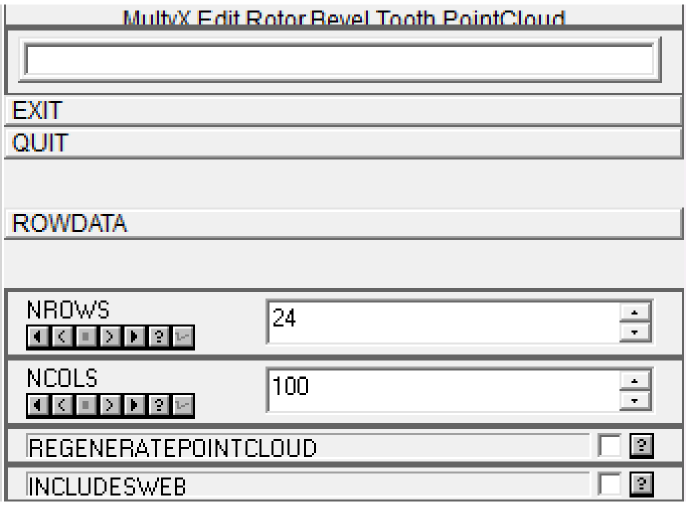

Figure 1.74 The bevel pinion POINTCLOUD menu.

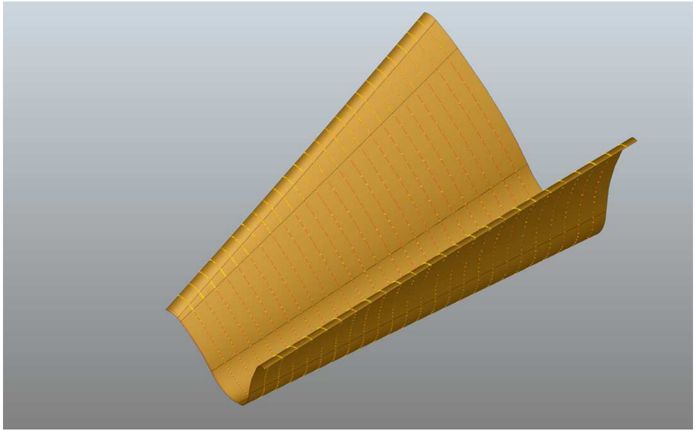

Figure 1.75 The generated point cloud.

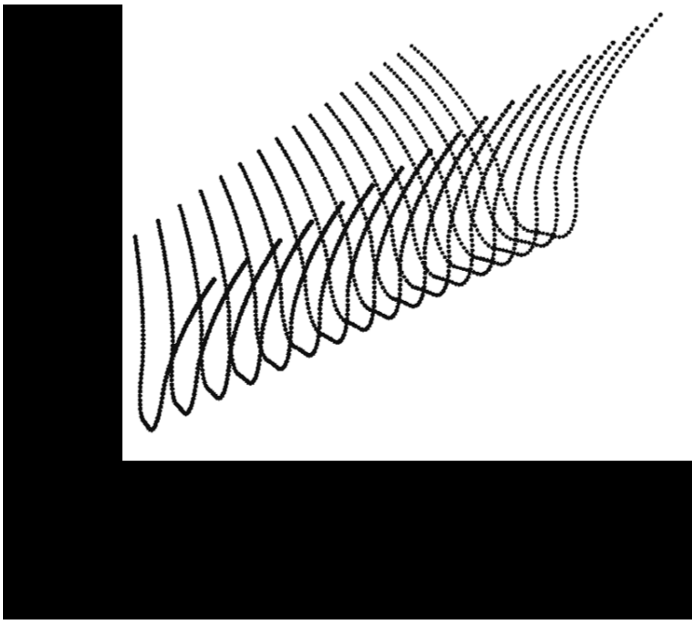

Figure 1.76 The point cloud orientation.

CHAPTER 2

PRE AND POST PROCESSING USING IGLASSVIEWER

IglassViewer is a powerful tool for pre and postprocessing gear models and results. Several features have been added to the *Multyx* program 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 animate the models 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.

2.1 Generating an Iglass file for preprocessing

The GENIGLASSFILE command in Figure 1.1 will lead to a menu shown in Figure 2.1 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 2.1 the user can select the bodies to be displayed in the Iglass graphical window. Click on the START button in Figure 2.1 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 planetary system is shown in Figure 2.2. 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.

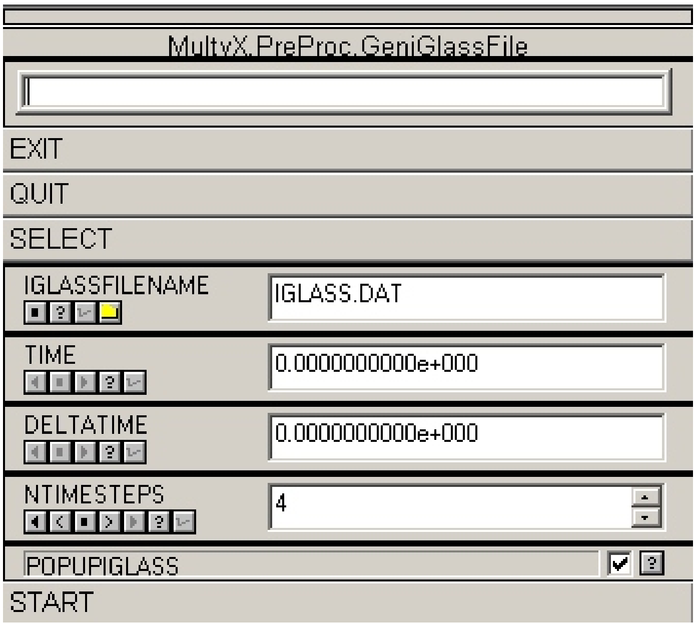

Figure 2.1 The generate Iglass file menu

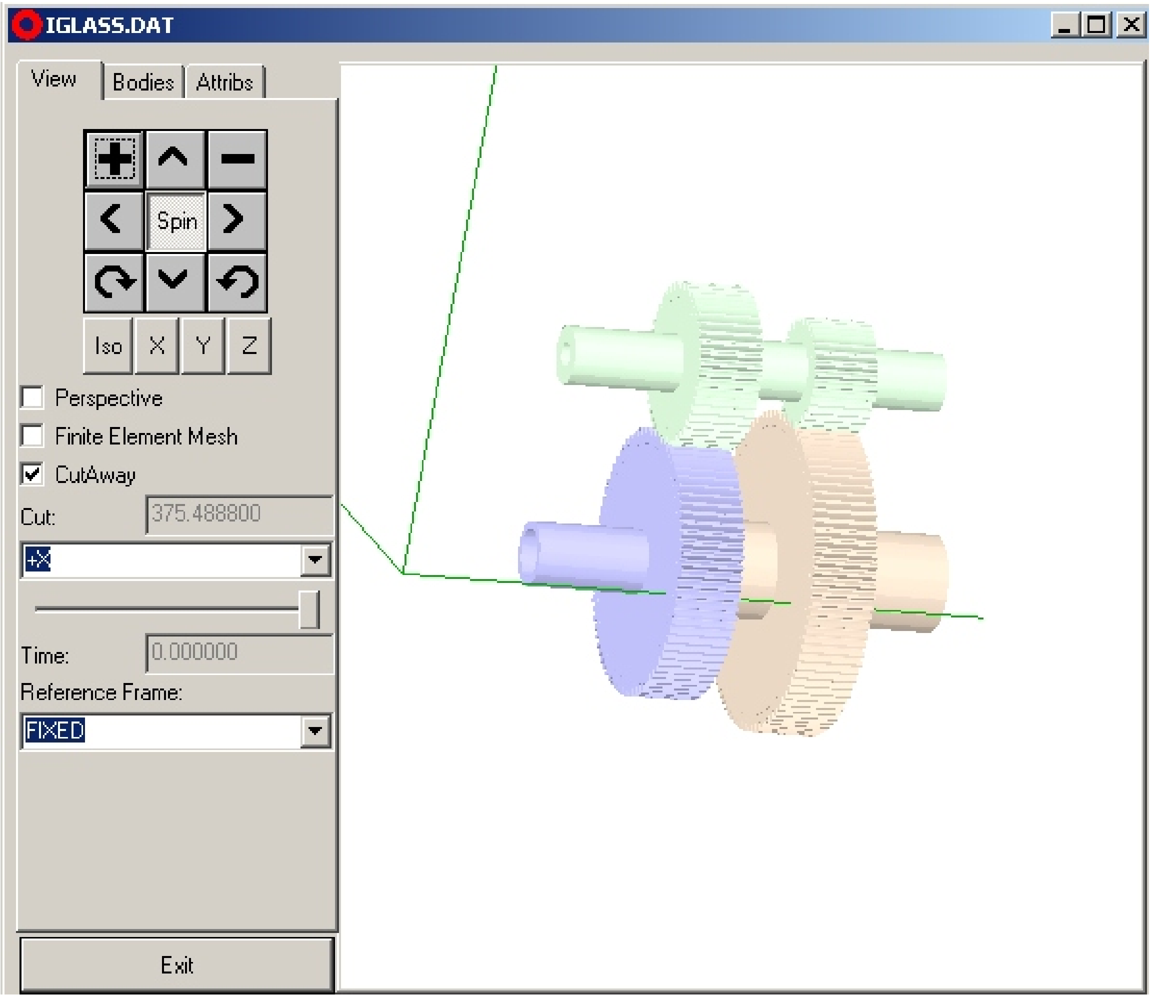

Figure 2.2 An example of an Iglass preprocessing window.

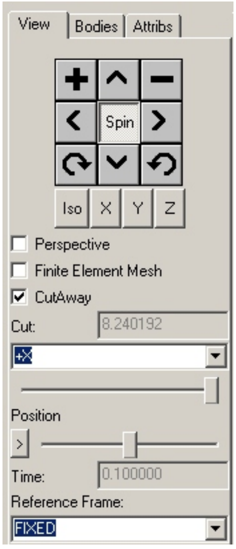

Figure 2.3 Iglass preprocessing view menu

2.2 View menu

The View menu is shown in Figure 2.3. Table 2.1 shows the common tasks performed by some of the buttons displayed in this menu.

Apart from all the features shown in Table 2.1 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.

2.2.1 Finite element mesh

The finite element mesh model can be visualised if the ‘Finite Element Mesh’ item is selected. Figure 2.4 shows the finite element mesh model of the gear bodies in iglass preprocessing.

2.2.2 Cutting plane

Using the cutting plane switch shown in Figure 2.5 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.

2.2.3 Selecting the time step

User can visualise the model at a particular timestep in iglass pre-processing using the ‘Position’ slider shown in Figure 2.6. 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 2.7.

2.2.4 Reference frames

The default reference frame is the FIXED reference frame. All the bodies appear to move when observed from the FIXED frame. The model will align itself to this reference frame when the iglass window pops up.

Table 2.1 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

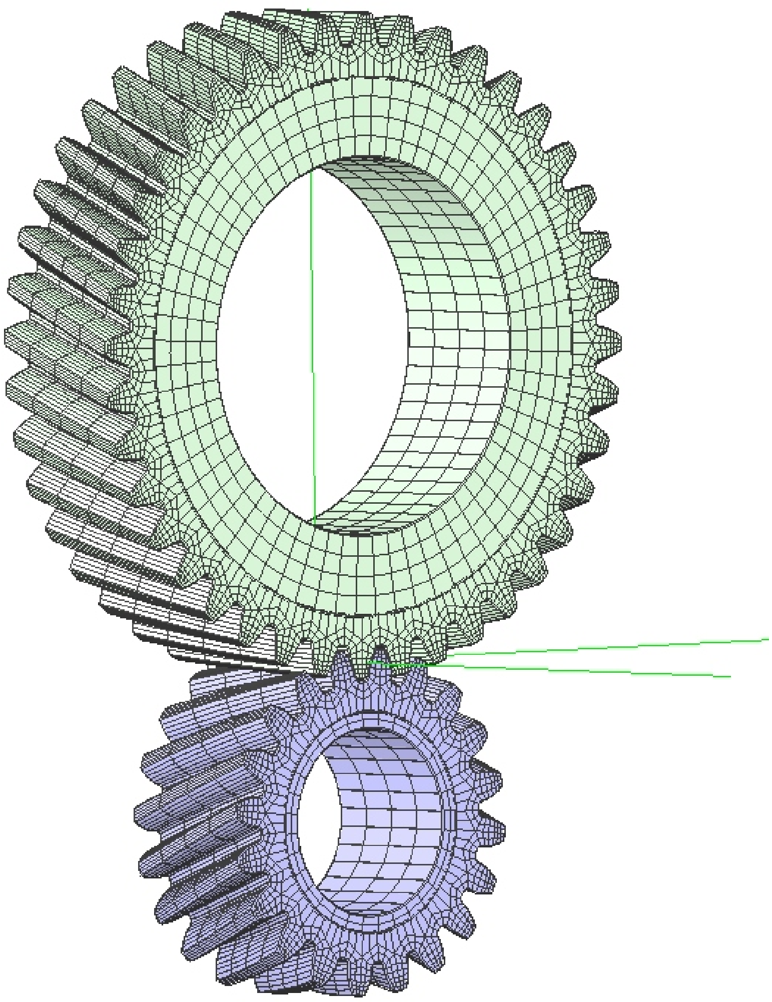

Figure 2.4 Finite element mesh model of the gear bodies

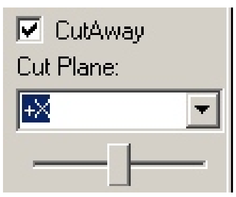

Figure 2.5 The cutting plane switch.

Figure 2.6 The position slider.

Figure 2.7 The time menu.

Figure 2.8 The reference frame switch.

The reference frame can be aligned to a body member using the reference frame switch shown in Figure 2.8. If you select the SUN gear as the reference frame the reference frame origin will coincide with the origin of the sun. The sun will appear stationary when observed from the SUN reference frame, and the other bodies orbit around it. If the PINION option is selected then the reference frame origin aligns itself to the origin of the pinion.

2.3 The Bodies menu

The 'Bodies' menu is shown in Figure 2.9. 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.

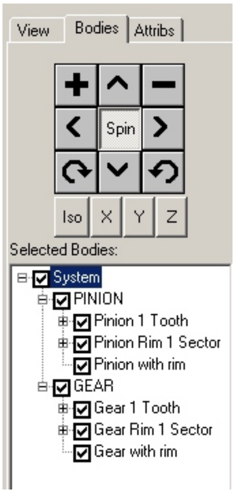

Figure 2.9 Iglass preprocessing Bodies menu

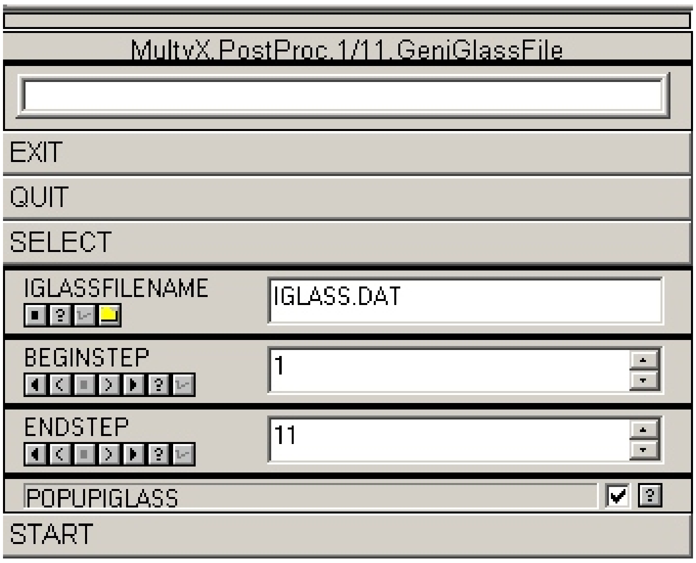

Figure 2.10 The generate iglass file menu for post processing.

2.4 Post processing using iglass

The GENIGLASSFILE command in Figure 1.3 leads to the generate iglass file menu shown in Figure 2.10 for post processing in iglass. BEGINSTEP and ENDSTEP menus shown in Figure 2.10 define the range for which you want to check for results. Note that these menus are independent of the GOTOPOSN menu shown in Figure 1.3.

An example of an iglass post processing window is shown in Figure 2.11.

2.5 Features specific to iglass post processing

The position switch shown in Figure 2.12 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 2.13 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 2.14 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 2.15. The magnification scale of the forces and the moments can be adjusted using the respective sliders.

The 'Attribs' menu is shown in Figure 2.16. The attribute menu shown in Figure 2.17 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.

The colors for minimum and maximum stress contours can be controlled using the palette mode menu shown in Figure 2.18. 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

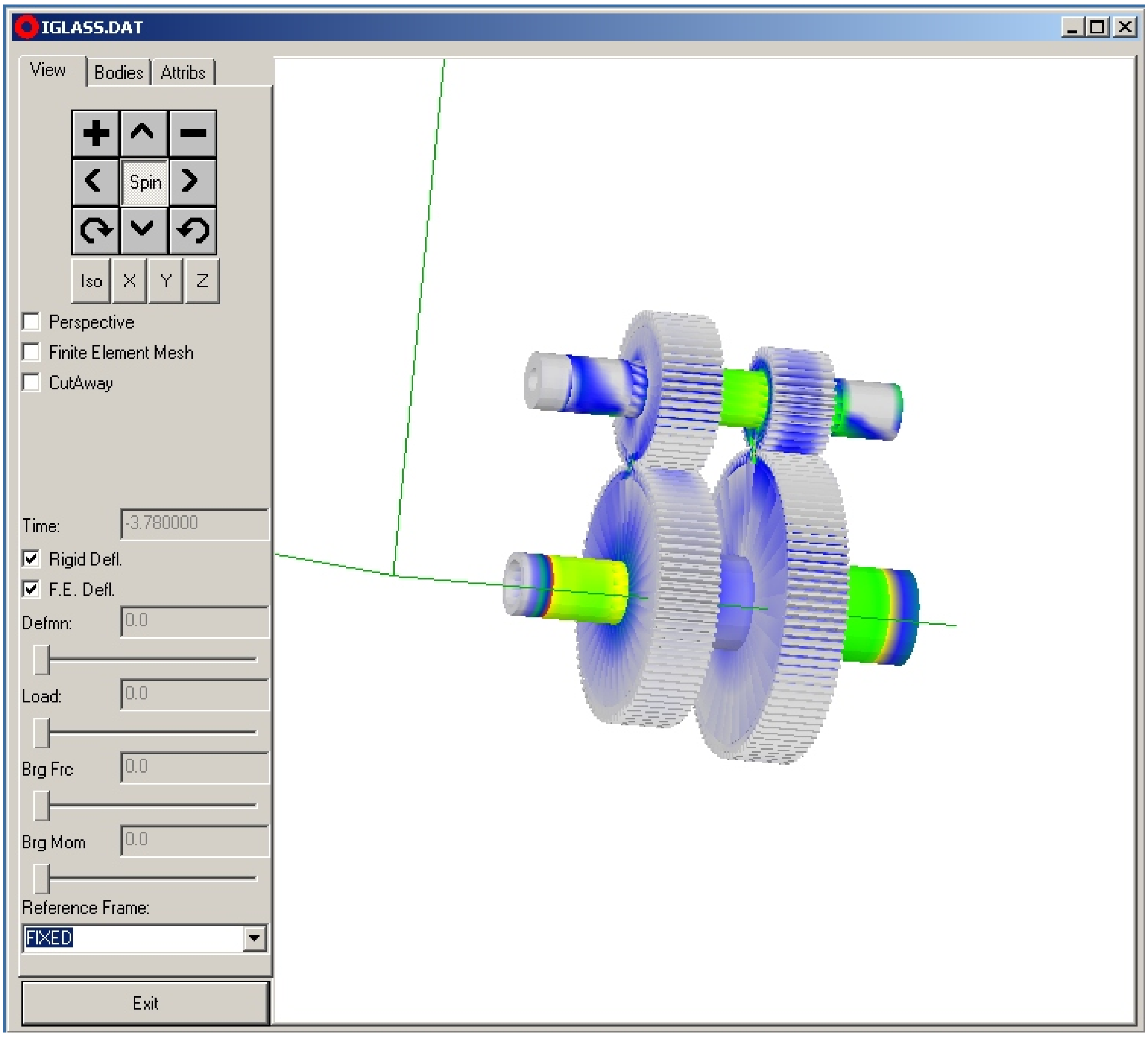

Figure 2.11 An example of an iglass post processing window.

Figure 2.12 The position slider.

Figure 2.13 The deformation slider.

Figure 2.14 The load slider.

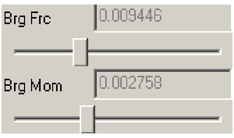

Figure 2.15 The bearing forces and moments sliders.

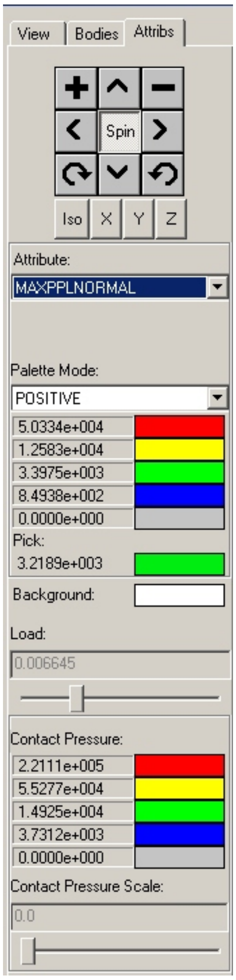

Figure 2.16 The iglass postprocessing attribute menu.

Figure 2.17 The attribute switch.

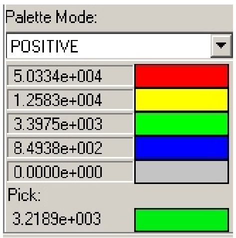

Figure 2.18 The palette switch.

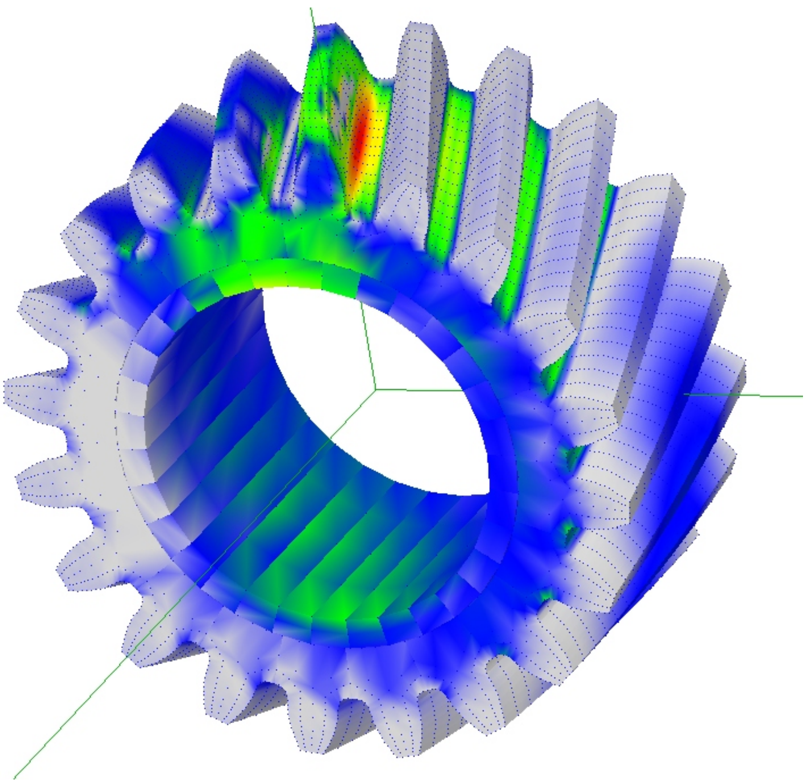

Figure 2.19 Picking the stress value at a nodal point of the finite element mesh

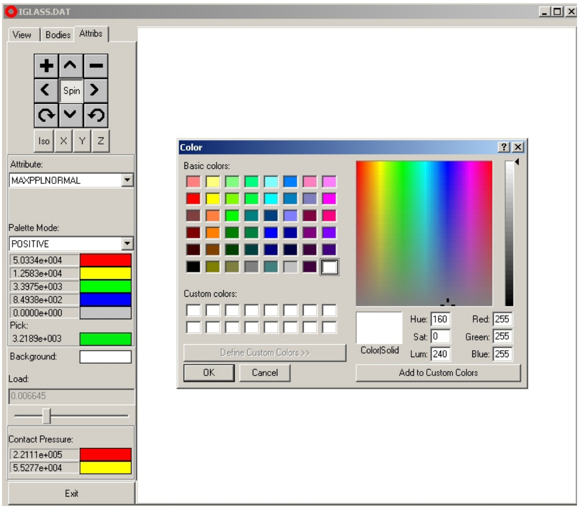

Figure 2.20 The background color popup window switch.

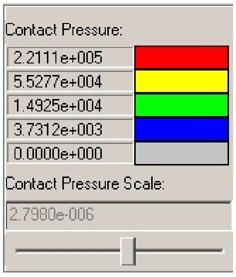

Figure 2.21 The Contact pattern menu.

BOTH type mode will align the scale from the maximum negative value (minimum stress) to a maximum positive value (maximum stress). In order to find the stress at a node, double click on the gear body. The finite element nodes are now visible as shown in figure 2.19. 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 2.20 using which you can change the background color of the iglass graphics window.

The Contact pattern menu shown in Figure 2.21 is used to view the contact pressure pattern on the contacting surfaces. Figure 2.22 shows an example of a contact pattern on the gear tooth.

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

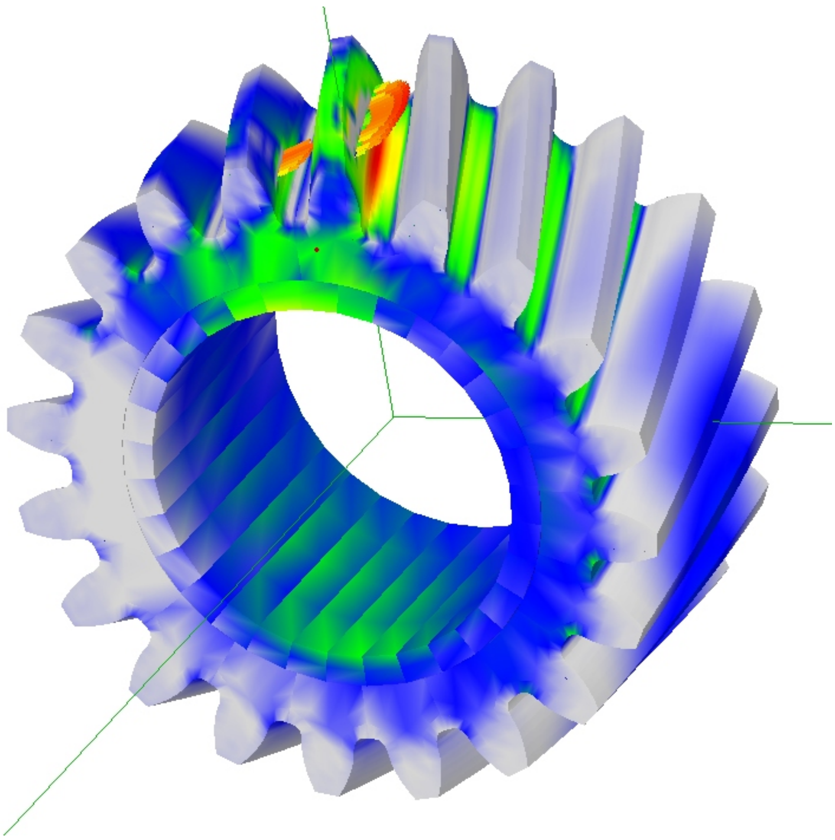

Figure 2.22 Example of a contact pattern on a gear tooth

REFERENCES

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

[18] *Analytical Mechanics of Gears*, Earle Buckingham, McGraw-Hill Book Company, Inc, 1949.

[19] ISO/TR 15144:2010: Calculation of Micropitting Load Capacity of Cylindrical Spur and Helical Gears - Part1: Introduction and Basic principles

[20] Bajpai, P., Kahraman, A., and Anderson, N. E. (2004), Surface Wear Prediction Methodology for Parallel-Axis Gear Pairs,Journal of Tribology 126, pp. 597-605

[21] *Fundamentals of Metal Fatigue Analysis*, Bannantine, Julie A., Comer, Jess J. and Handrock, James L. Prentice Hall, Inc., 1990.