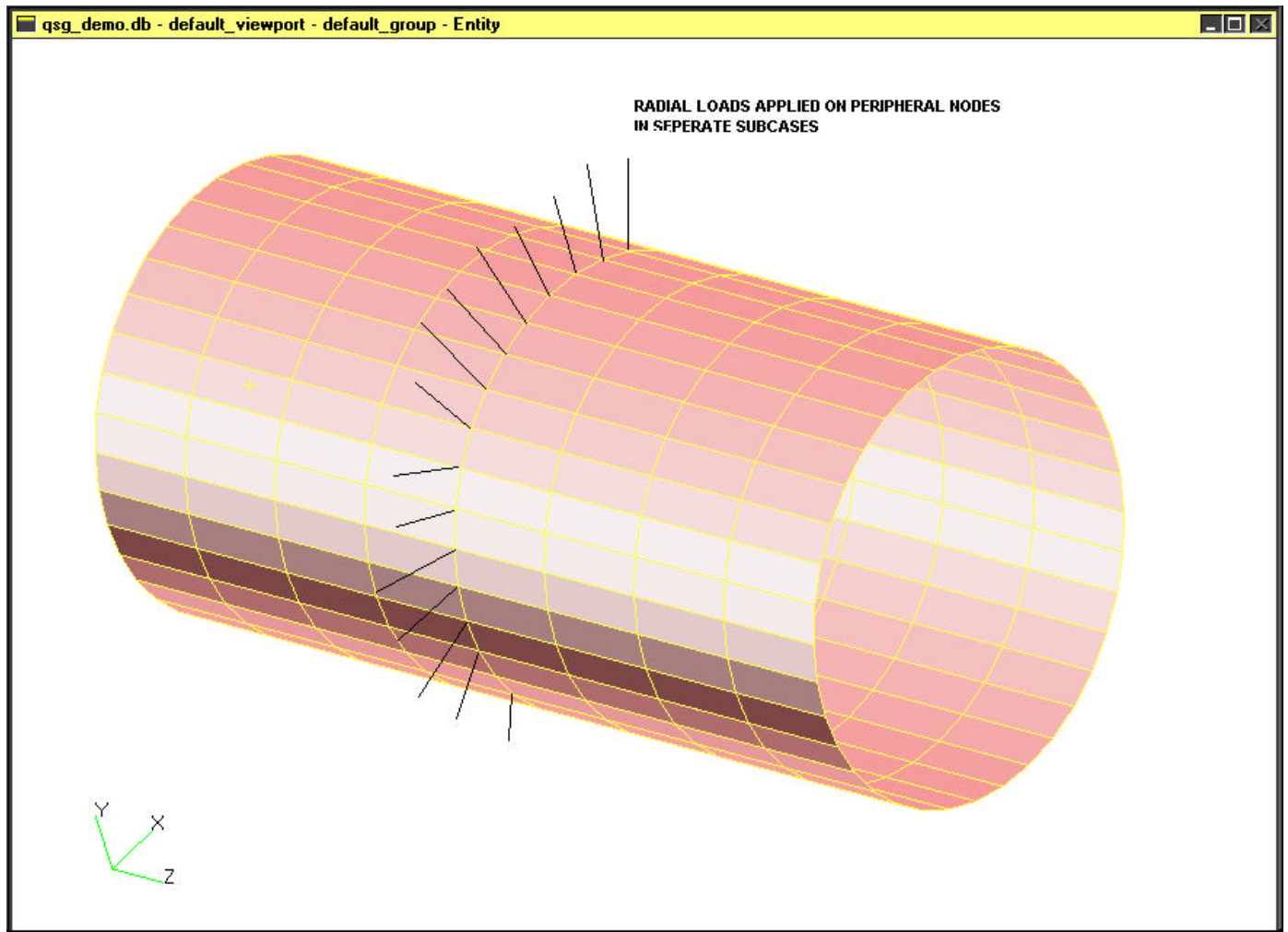


# Quick Start Guide

## Wheels Module – Analysis of Rotating Structures

### Problem Description

This simple example is used to demonstrate the Wheels module in MSC.Fatigue. To aid in quick familiarity with this module.op2 files for NT and Unix platforms are included for the model shown below.



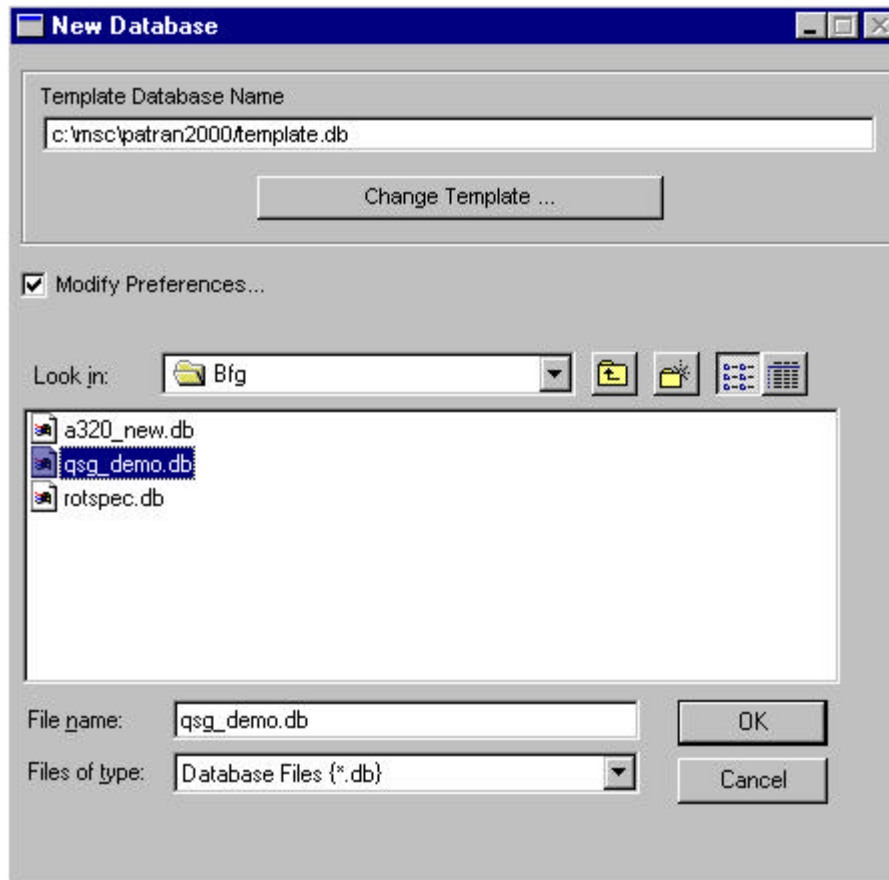
**Figure 1**

The model is an 80 in (L) by 20in (R) hollow cylindrical modeled with cquad4 elements ( $t=0.01\text{in.}$ ) meshed using a 10 in. by 10 degree increment mesh. The tube is clamped at each end and point loads of 0.5 lb. have been applied at 10-degree increments in separate subcases on the peripheral nodes as shown in Figure 1. The reason for the application of

the load of this magnitude is due to the fact that the wheels module, treats the output stresses in KSI units and any significant stresses in the plates will lead to erroneous interpretation of reported lives.

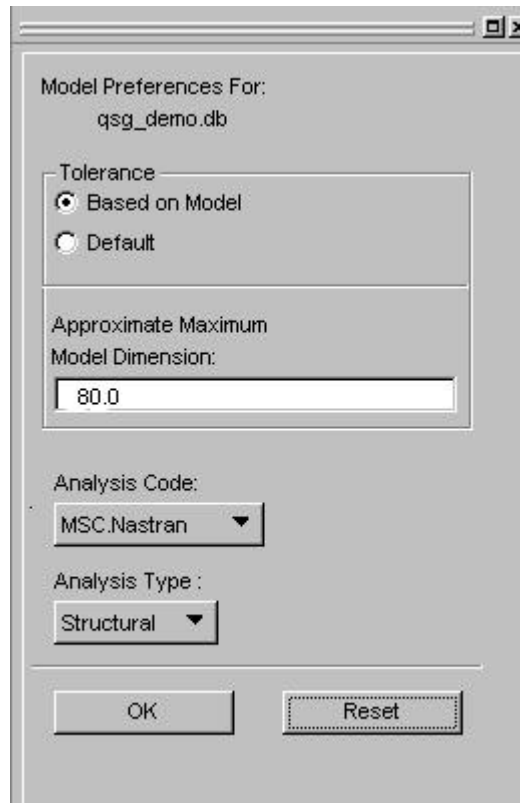
### **Loading the Model in Patran:**

Start Patran or the stand-alone version of MSC.Fatigue and select **File / New**. Enter a new database name as **qsg\_demo.db** as shown in Figure 1.



**Figure 1**

On the Model Preferences form select a default Nastran | Structural run and set size to 80 as shown in Figure 2.



**Figure 2**

Now load the data file by selecting the *Analysis* option from the menu bar or use the Analysis button in pre-post standalone. The Analysis form is shown in Figure 3. From the Analysis form select the following properties:

- Action = Read Output2
- Object = Both
- Method = Translate
- Jobname = demo\_qsg

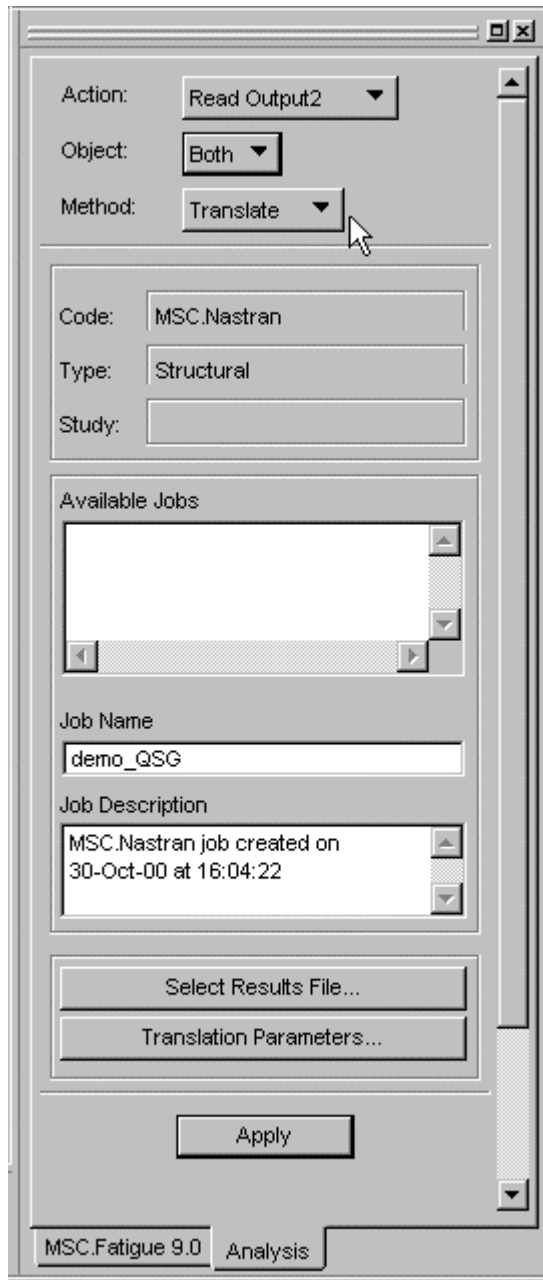


Figure 3

Click **Select Results File** and select the op2 file, cylinder\_model.op2.

Click Apply and wait for the model to load.

### Viewing the Stress Results:

Select **Results** from the toolbar. Choose your results to view and click **Apply** to see the results from the .op2 file. In the figure below, maximum principal 2D results for the first 9 sub cases are shown, that represent a 80 degree “rotation” of the load in 10 degree increments.

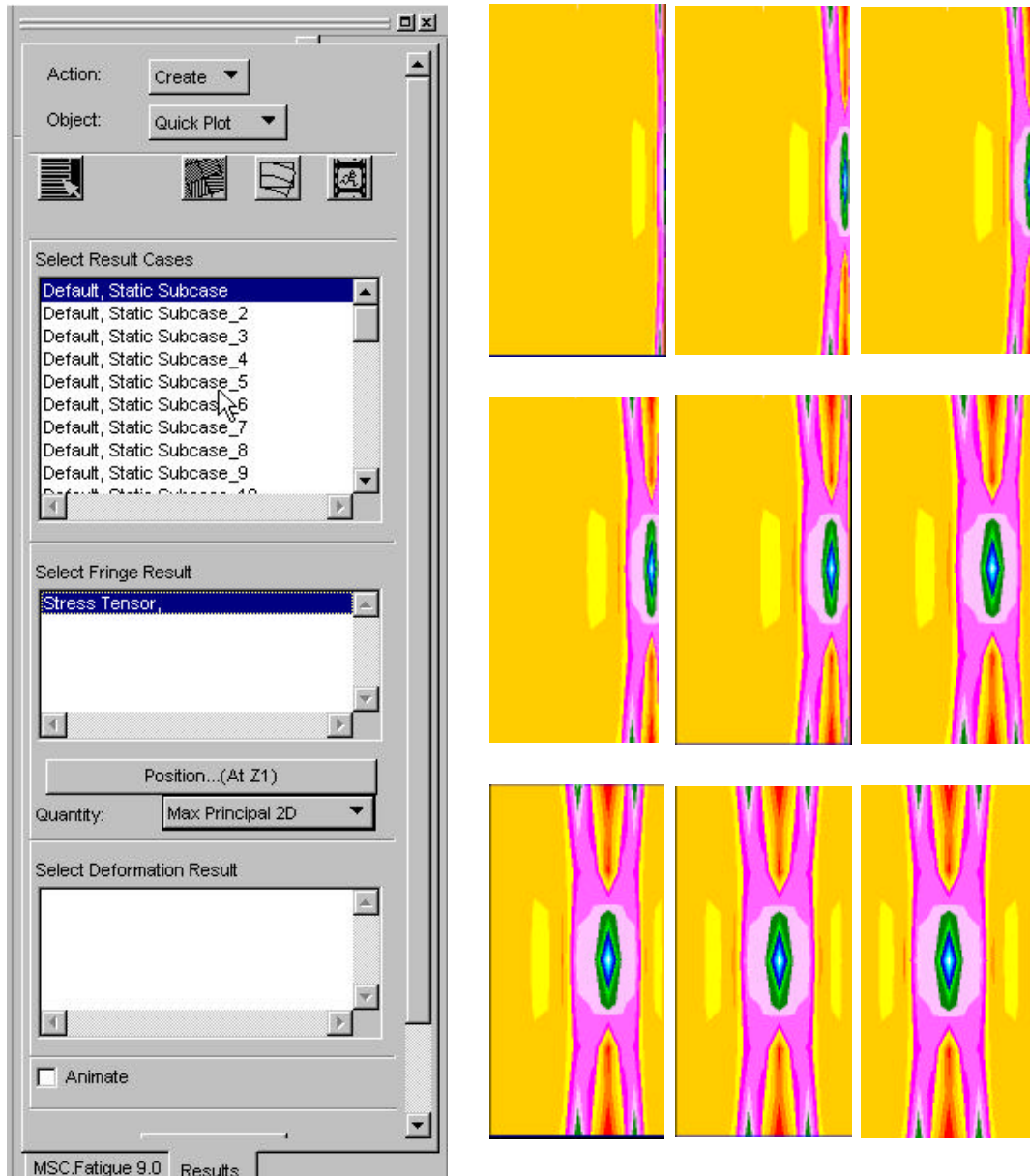


Figure 4

### Setting Up the Fatigue Analysis:

To set up a fatigue run select **Tools / FATIGUE**, from the main menu. Select Wheels and specify a Jobname:

- Analysis = Wheels
- Jobname = qsg\_demo

The form is shown in Figure 5.

General Setup Parameters:

Analysis: Wheels

Results Loc.: Node

Nodal Ave.: Global

F.E. Results: Stress

Res. Units: KSI

Jobname (32 chrs max) =  
qsg\_demo

Title (80 chrs max) =

Specific Setup Forms:

Solution Params...

Material Info...

Loading Info...

Job Control/Results Forms:

Job Control...

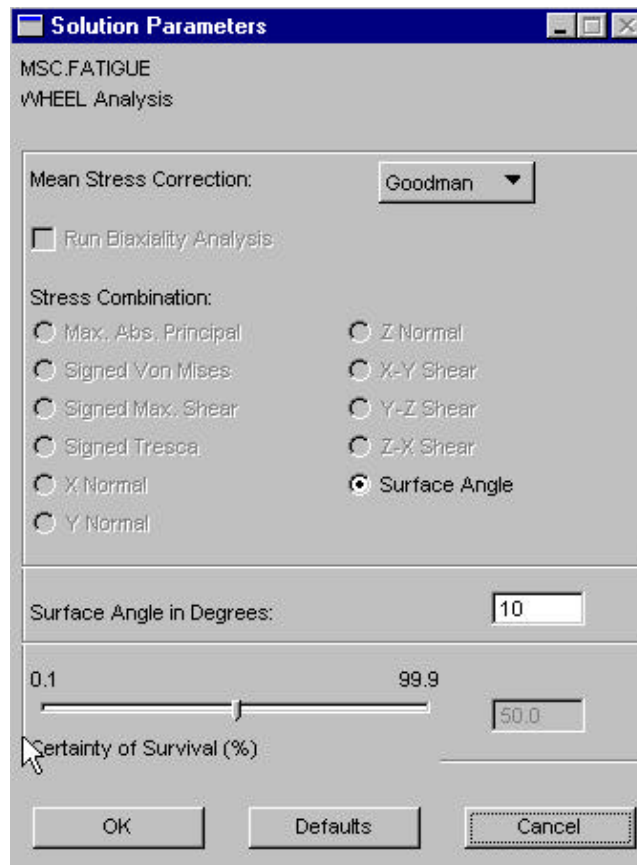
MSC.Fatigue 9.0

**Figure 5**

Element or nodal results may be processed and results averaged globally or on the selected group. FE results are restricted to stresses in KSI only. Users are cautioned about this limitation but future versions will allow different units.

### Solution Parameters:

Click on the *Solution Parameters* button. The form shown in Figure 6 is displayed. Set the *Mean Stress Correction* to Goodman.



The image shows a dialog box titled "Solution Parameters" for "MSC.FATIGUE WHEEL Analysis". The "Mean Stress Correction" is set to "Goodman". The "Run Biaxiality Analysis" checkbox is unchecked. Under "Stress Combination", "Surface Angle" is selected. The "Surface Angle in Degrees" is set to 10. A slider for "Certainty of Survival (%)" is set to 50.0. The dialog includes "OK", "Defaults", and "Cancel" buttons.

Figure 6

Biaxiality analysis or the stress combination methods are not available, since the Wheels module performs a critical plane S-N analysis using surface resolved stresses at every surface node. Although this is not necessary for this model, other models, in particular solid element models, will require stresses to be surface resolved. The option to generate surface normals is available from the job control menu.

The surface angle selects the increment at which analysis will be performed. In this case surface stresses will be resolved in 10-degree increments and an analysis performed at every 10-degree increment.

The certainty of survival is not selectable in this module – the S-N curve is used without any modifications.

## Material Information:

This materials form shown below In Figure 7. Select

- Material - 7075-HV\_T6
- Finish – No Finish
- Treatment – No Treatment
- Region – Default
- Select defaults for the remaining fields

MSC.FATIGUE

WHEEL Analysis

Materials Database Manager    Select Standard Database    Select User Database

Current Mat. Database: c:\msc\patran2000\mscfatigue\_files\mats\hmatmasmas.adb:CENTRAL

Number of Materials: 1

Selected Materials Information:

	Material	Finish	Treatment	Region
1	7075_HV_T6	No Finish	No Treatment	default_group

OK    Defaults    Cancel

Figure 7

## Loading Info:

Click the Loading Info Button to see the loading form shown in Figure 8.

MSC.FATIGUE

Enter Fatigue Equivalent Units: 1.0 repeats of the loading = 1.0 MILES

Parameters for Loading Conditions: Number of Loading Conditions: 5  Fill Down OFF No Transformation

	Loading Condition ID	Design Life (MILES)	Loading Factor
1			
2			
3			
4			
5			

OK Defaults Cancel

**Figure 8**

This form allows us to specify the loading environment. The default number of load conditions is 5 and for this module it is important to understand the concept of a load condition. The load condition represents a particular type of loading. The first load condition could be the set of sub cases for one revolution that define a straight roll, the second could be a set of sub cases for one revolution that define turning condition, and similarly for the third, fourth and fifth load conditions. A typical usage profile for a wheel is shown in the table below.

Loading Condition	Vertical Load [lb.]	Lateral Load [lb.]	Required Mileage
Straight Roll	11,300	0	42,500
15% Inboard Turn	11,300	1,700	3,250
15% Outboard Turn	11,300	-1,700	3,250
30% Inboard Turn	11,300	3,400	250
30% Outboard Turn	11,300	-3,400	250

In our example, 36 sub cases (loads at 10 degree increments – 360/10=36 sub cases) constitute the first and only load condition. The table below illustrates how the stress at every surface angle and rotational angle (this is merely the same load applied in the next subcase at the node corresponding to a 10-degree rotation) is extracted and damage computed for the load condition. The stress time history at an analysis node is a sequence of stresses extracted from each sub case for every surface angle for each rotational increment (sub case). In the table below the stress time history for the analysis node for a surface angle  $q$  is the column associated with the surface angle. It is also worth noting that since the stress time histories are created from the sub cases there are no .DAC files required for analysis.

Rotational Angle F = subcase #	Surface angle q				
	10°	20°	...	350°	360°
0° (subcase 1)	$\sigma_{1,10}$	$\sigma_{1,20}$	...	$\sigma_{1,350}$	$\sigma_{1,360}$
10° (subcase 2)	$\sigma_{2,10}$	$\sigma_{2,2}$	...	$\sigma_{2,350}$	$\sigma_{2,360}$
20° (subcase 3)	$\sigma_{3,10}$	$\sigma_{3,2}$	...	$\sigma_{3,350}$	$\sigma_{3,360}$
...	...	...	...	...	...
350°(subcase 36)	$\sigma_{36,1}$	$\sigma_{36,2}$	...	$\sigma_{36,350}$	$\sigma_{36,360}$
<b>Damage <math>D_{qL}</math></b>	$D_{1,L}$	$D_{2,L}$	...	$D_{36,L}$	$D_{36,L}$

The loading form allows user selectable units for reporting life. Here, approximately 500 repeats of the loading is equivalent to 1 mile. Enter the respective quantities in the cells as shown in Figure 10.

Change the number of load conditions to 1. The .op2 file contains results for 36 sub cases that will be used as the load condition.

Click on the loading conditions cell. The form shown below (Figure 9) is displayed.

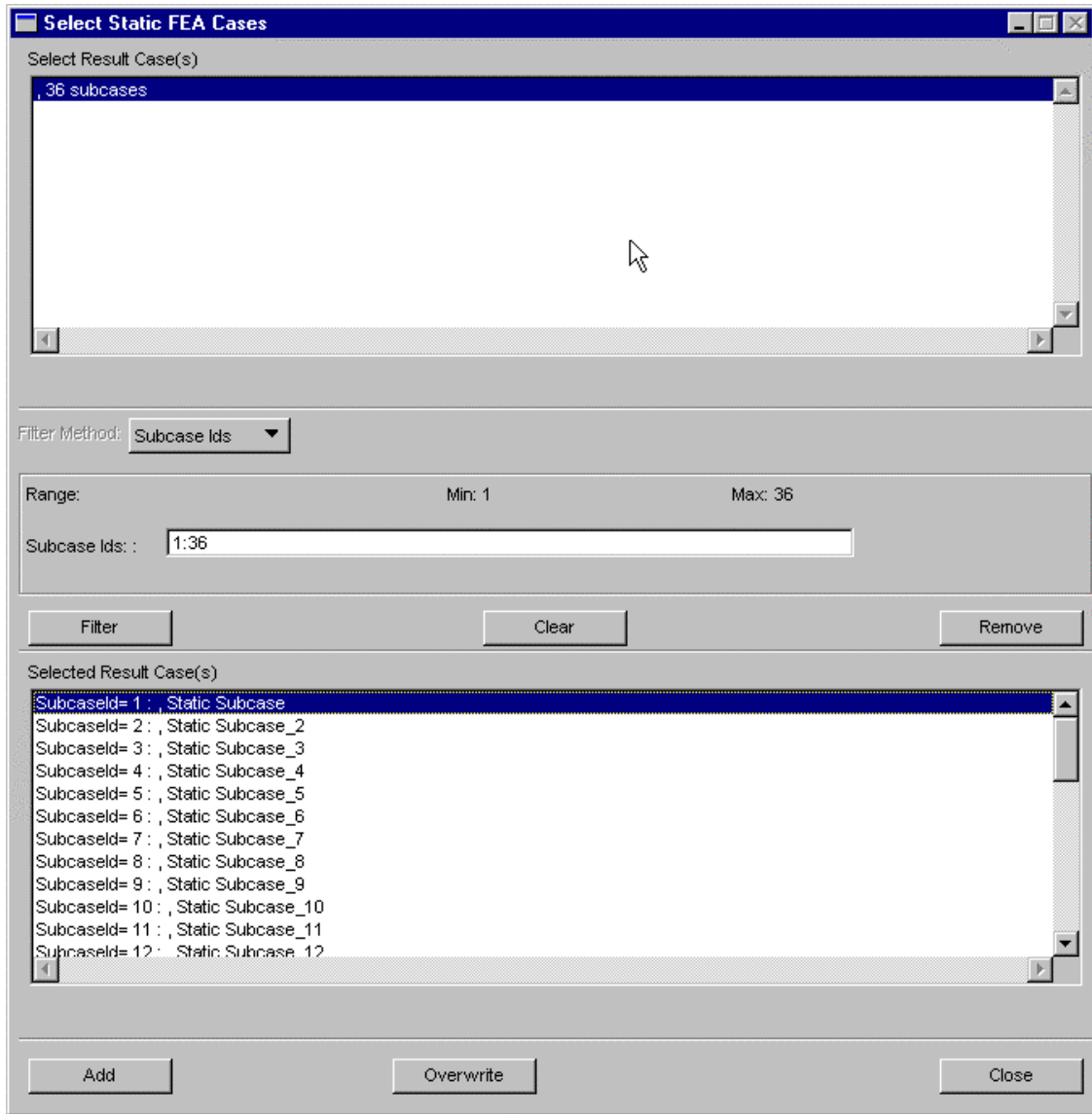


Figure 9

Select the default, 36 sub cases and click on the filter button to populate the bottom data box – here all the subcases are lumped into one but it is possible to remove subcases, that appear in the lower window, by picking the subcase and clicking on the remove button. The Clear button clears the entire selection while the remove button allows the analyst to pick and remove subcases. Note also that the filter method, for selecting results cases from the upper window is grayed out – more options will be added in the future release to allow users flexibility in selecting results cases.

Since, we are using all subcases we will accept them by clicking on the add button. The overwrite button allows the analyst to change the selection in case a mistake is made. Click on the close button to accept and load the Loading condition ID cell on the loading information form as shown in Figure 10.

MSC.FATIGUE

Enter Fatigue Equivalent Units:

500 repeats of the loading = 1.0 MILES

Parameters for Loading Conditions:

Number of Loading Conditions: 1  Fill Down OFF No Transformation

Loading Condition ID	Design Life (MILES)	Loading Factor
1	1000.	0.30000001

Get/Filter Results...

Select Loading Condition Results:

1, 2 ( 1:36 )-

Select a Stress Tensor:

1.1-Stress Tensor,

Select a Layer:

1-At Z1

Fill Cell

OK Defaults Cancel

Figure 10

Since we previously set the life reporting units to miles, we will set the design life to 1000 miles. The design life is the target distance and allows reporting a factor of safety on the calculated life.

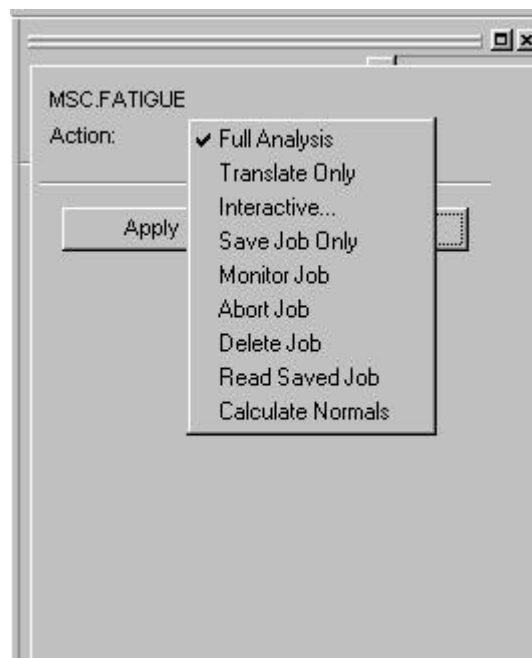
Set the design life to 1000 miles.

Set the loading factor to 0.3. Since the stresses on the .op2 file are used in units of KSI it is necessary to provide this scaling to get the stresses to a reasonable limit.

### **Fatigue Analysis:**

Click on the ***Job Control*** button and select ***Full Analysis*** form the form. Click on Apply to submit the fatigue analysis. The job can be monitored by setting Action = Monitor Job. When the job has completed click ***Cancel*** to return to the main Fatigue form.

Note also the availability of the Calculate Normals option – this would be used to generate surface normal files for solid element models.



**Figure 11**

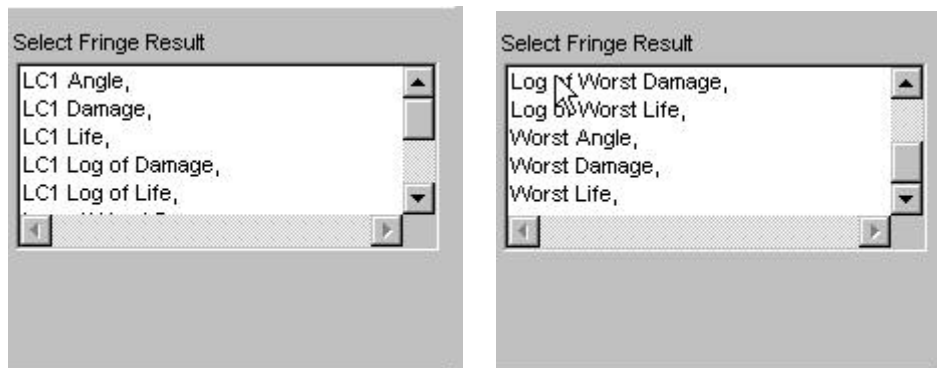
### **FEF file output results:**

The jobname.fef file contains the following items.

1. Node
2. LC1 Angle – angle of lowest life at the node for load condition 1
3. LC1 Damage – worst damage at the node for load condition 1
4. LC1 Life – life in equivalent units at the node for load condition 1
5. LC1 Log of Damage – log of damage at the node for load condition 1
6. LC1 Log of Life– log of life at the node for load condition 1
7. Log of Worst Damage- For a single load condition, this is identical to (5). For multiple conditions this is the log of the sum of damage from all load conditions.
8. Log of Worst Life – Log of (11).
9. Worst Angle -- For a single load condition, this is identical to (2). For multiple conditions this is the surface angle which has the highest accumulated damage from all load conditions.
10. Worst Damage - For a single load condition, this is identical to (3). For multiple conditions this is the sum of damage from all load conditions.
11. Worst Life - For a single load condition, this is factor of safety. For multiple conditions this is sum of the factors of safety of each load condition.

### **Plotting the Fatigue Results:**

From the Results menu change Action = **Read Results**. Click on **Apply** then **Cancel** to return to the main Fatigue form. The results are now loaded into the MSC.Fatigue database. The jobname.fef file contains the results that may be plotted as shown in Figure 12.



**Figure 12**

### **Plotting the Fatigue Results:**

Select Results from the main Toolbar and fill in the form as follows:

- Select Result Cases = qsg\_demofef
- Select Results = Log of Life ()
- Click Apply

The log of life fatigue fringe plot is shown in Figure 13.



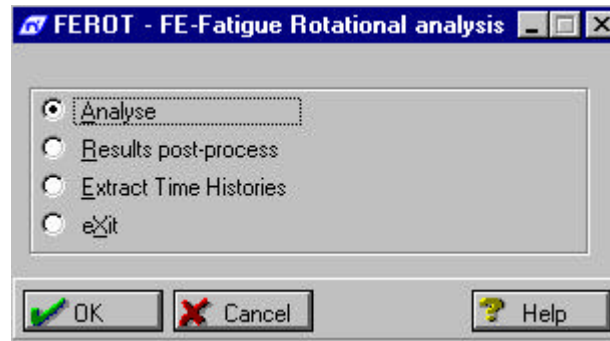
Figure 13

The fringe plots show the expected results – since the boundary conditions do not change and the loading is identical for every subcase, the maximum and minimum principal stresses for every subcase are identical (see fringe plots in Figure 4). Consequently, we see progressive concentric circles of increasing life from the lowest in the middle of the cylinder (area of maximum stress under the applied load) to highest in between the constrained ends and the middle.

Plots of the other items are not shown here – the user may want to exercise the other options by selecting them from the list shown in Figure 12.

### **Wheels Interactive menu:**

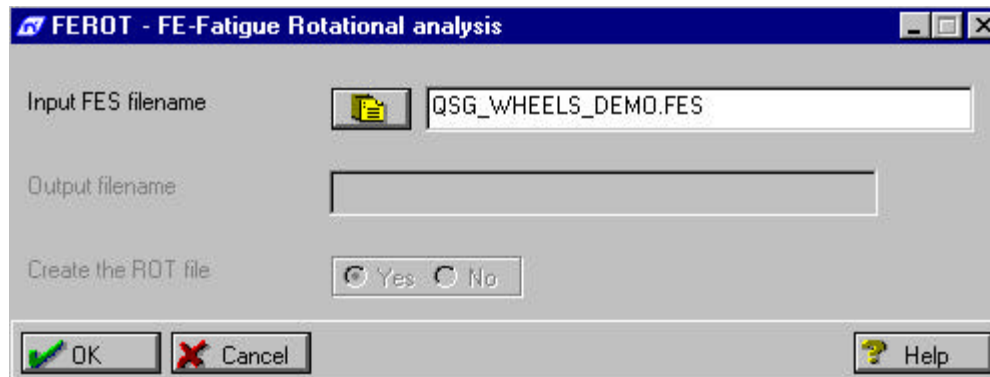
From the Job Control menu, select Interactive to display the Wheels Interactive menu (Figure 13).



**Figure 13**

The available options are:

- Analyse - Select Analyse and the following menu is displayed.



**Figure 14**

Select a .fes file that has been previously created. The output filename defaults to Jobname.fef but an alternative name may be supplied.

Click on “Yes” to create the ROT file. This file provides the user with useful information on the stress contribution and damage for each load condition at the analysis nodes for each rotational increment of the wheel (remember this is dependant on the number of subcases that were specified to define a revolution) and each surface angle increment. For large models, this listing can be quite extensive and the option is provided to turn this off.

A sample output of the .rot file is shown in Figure 15

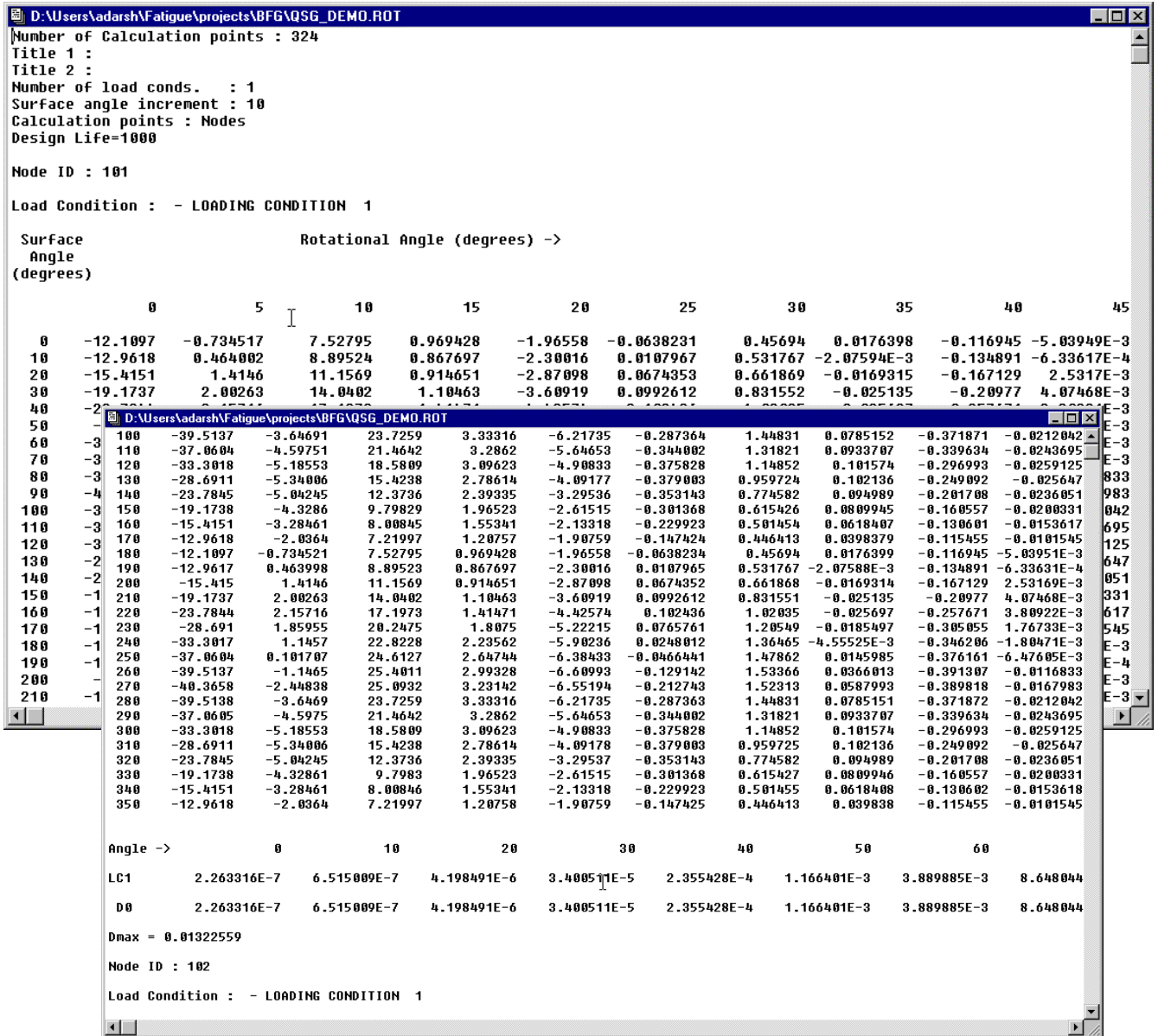


Figure 15

Each cell represents a stress from the surface angle and the subcase (rotational angle). Each row is a stress time history for the node at the particular surface angle and may be plotted by using the extract time histories from the interactive menu (Figure 13). The lower figure shows the damage values for each surface angle increment for load condition 1 and the maximum damage. The row labeled D0 shows the damage sum from all load conditions for each surface angle increment.

➤ Results Post-process

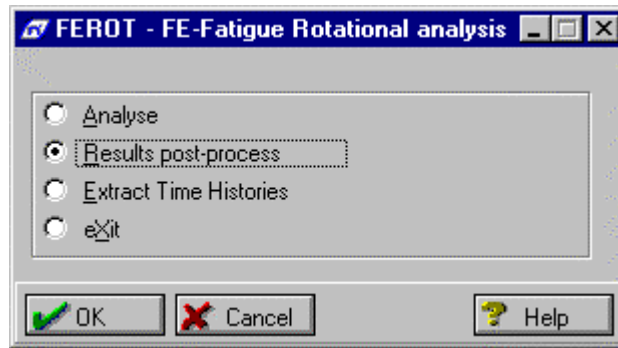


Figure 16

The following is displayed when this option is selected

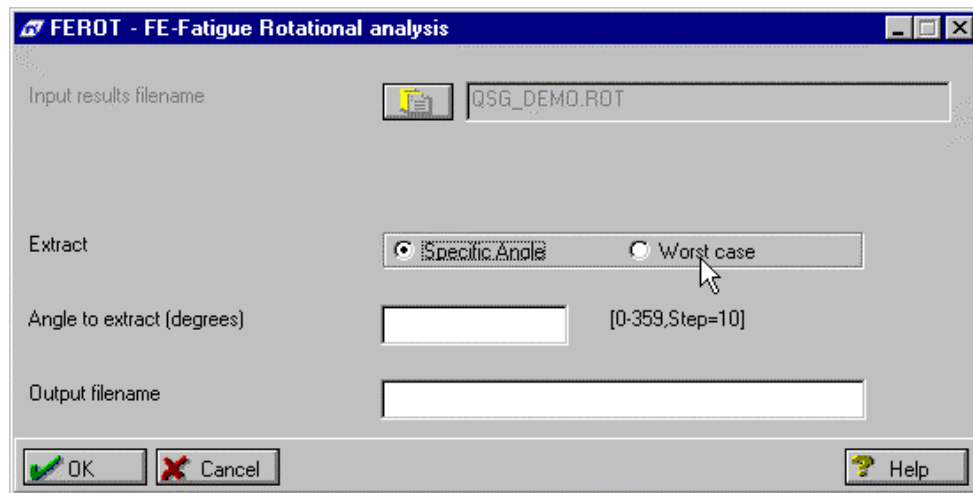


Figure 17

This menu allows further post-processing of the results that requires the .ROT file for input to generate a new output results file (username.fef file). This file can either be listed or read back into the MSC.Fatigue database by specifying the same Jobname in the main form (figure 5) as the output filename. The extraction options are: -

- Specific Angle – Extract results at a specific angle at all analysis nodes. Since the surface angle selected for our example case was 10 degrees, results may only be extracted for angles in ten-degree increments up to 360 degrees.
- Worst Case – Extract the worst (lowest life, maximum damage) results for all conditions including the worst case (accumulated damage from all conditions) results.

- Extract Time Histories – This option allows the extraction of stress time histories (.DAC files) for all load conditions at user defined nodes and surface angles.

Click on Extract Time Histories (Fig 16). The following is displayed.

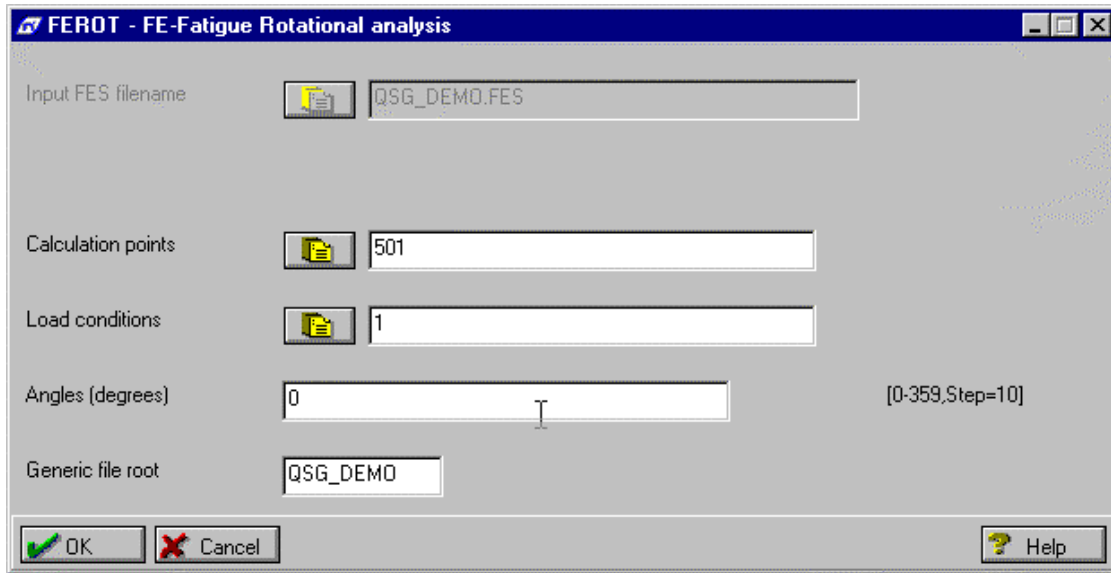


Figure 16

Click on the Input FES filename browse button and pick the FES file.  
Click on the Calculation points browse button to display the calculation nodes. Enter the node number.  
Click on the load conditions browse button to display the available load conditions.  
Enter the load condition number.  
Enter the surface angle at which the stress time history is desired.  
Enter the output filename. An extension will be added to the filename that identifies the node number, load condition and the surface angle. A plot of the stress time history will be automatically generated as shown in Figure 17.

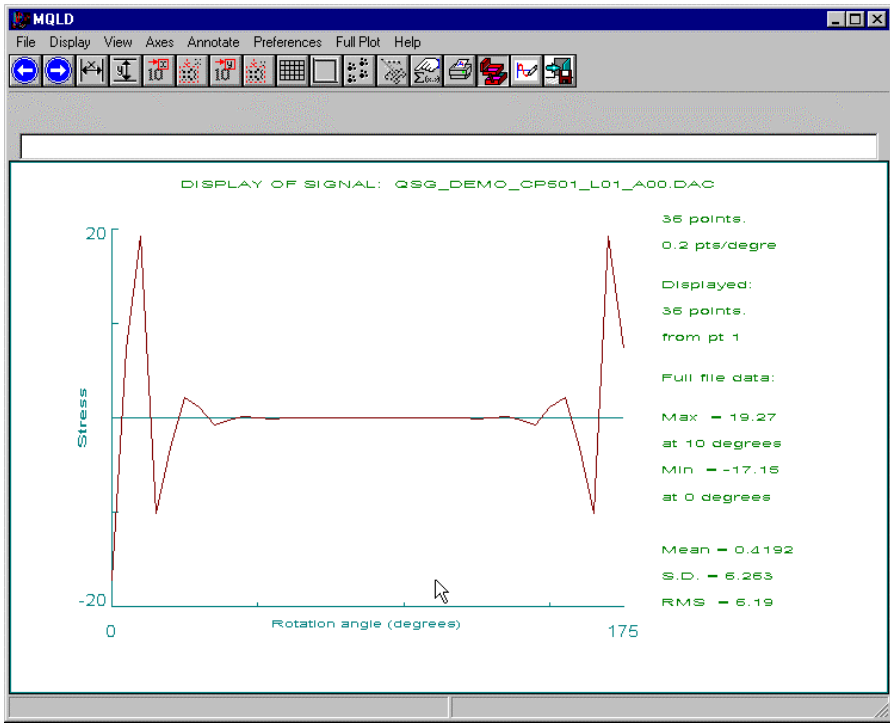


Figure 17