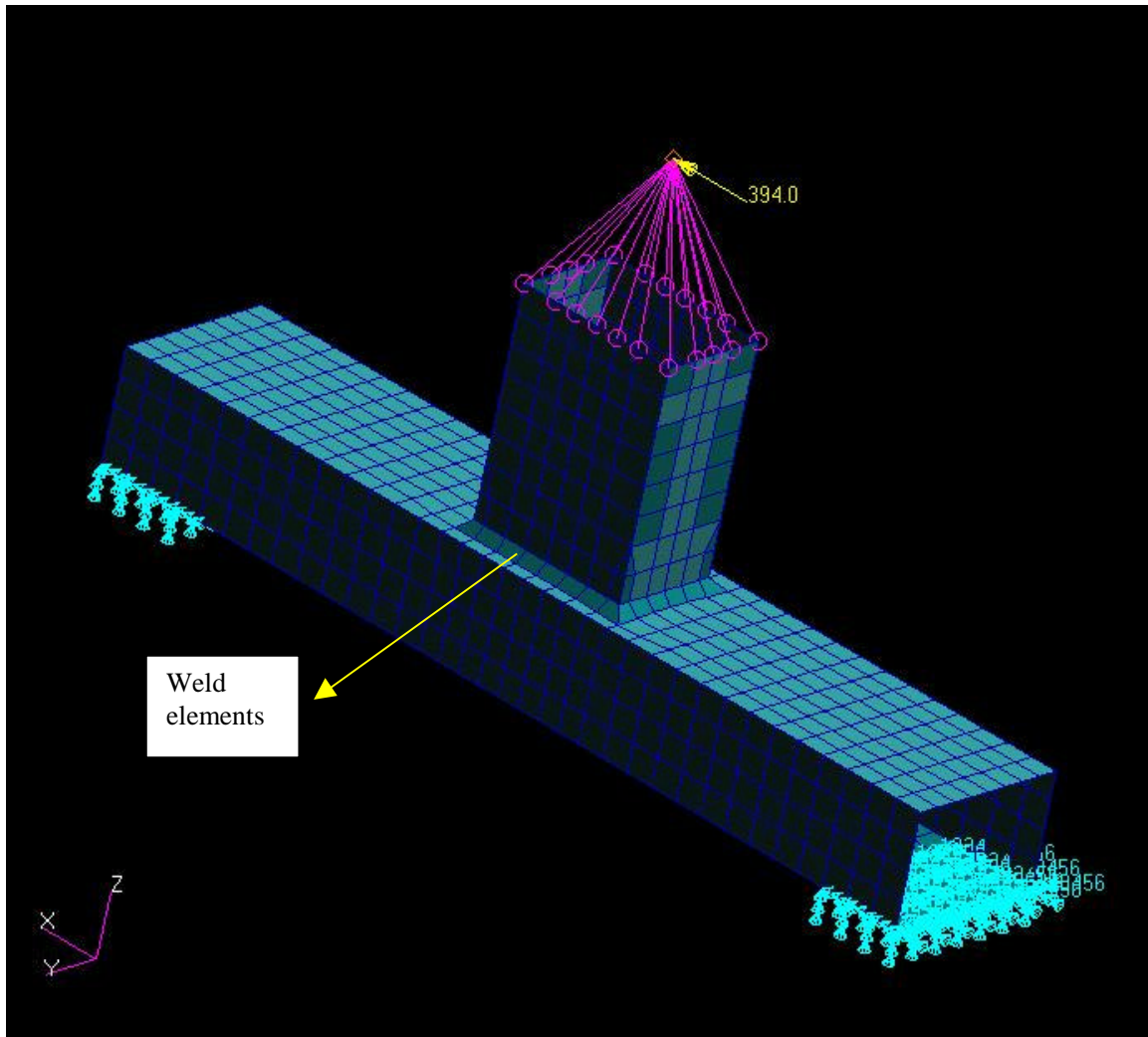


# Quick Start Guide

## Seam Weld Analysis

### Problem Description

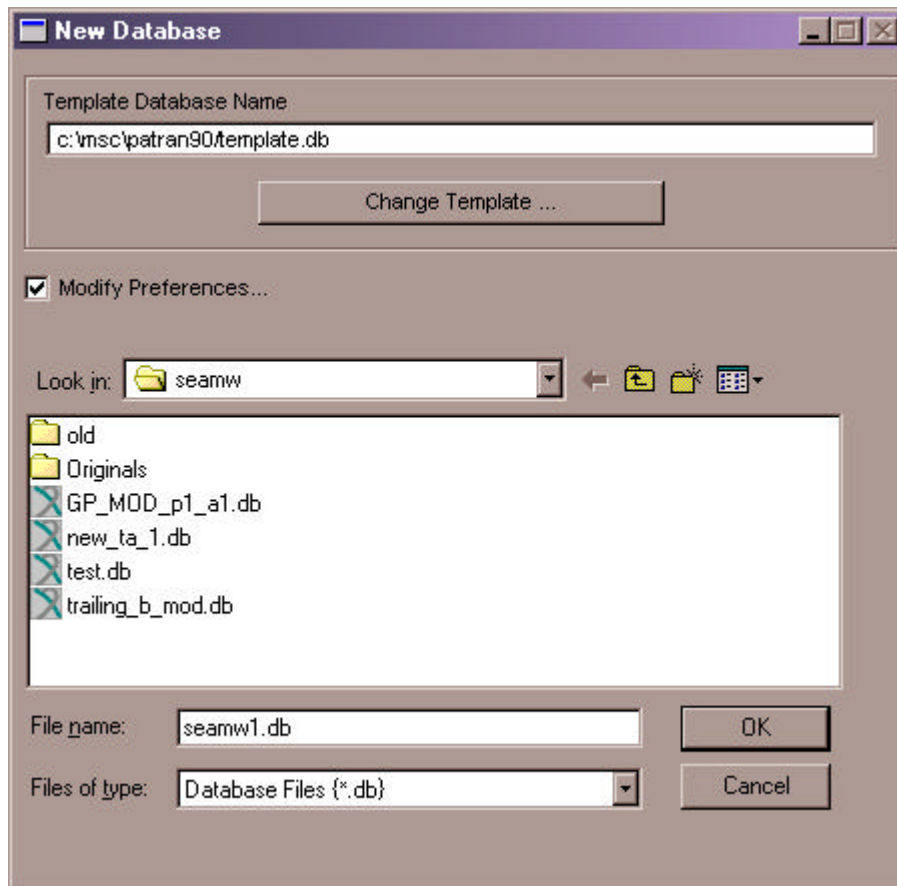
This simple example is used to demonstrate the use of Seam Weld Analysis in MSC.Fatigue. The .op2 file for the part is included in the delivery for users to familiarize themselves with the MAG Weld analysis module.



The tubes are welded as shown and a point load of 394N is applied as shown. The plate thickness is 1.5mm.

### Loading the Model in Patran:

Start Patran or the stand alone version of MSC.Fatigue and select **File / New**. Enter a new job name as **seamw1.db** as shown in Figure 1.



**Figure 1**

On the Model Preferences form select a default Nastran | Structural run as shown in Figure 2.

The image shows a 'Model Preferences' dialog box for a file named 'seamw1.db'. It contains several settings: 'Tolerance' is set to 'Based on Model'; 'Approximate Maximum Model Dimension' is set to '10.0'; 'Analysis Code' is set to 'MSC/NASTRAN'; and 'Analysis Type' is set to 'Structural'. There are 'OK' and 'Reset' buttons at the bottom.

Model Preferences For:  
seamw1.db

Tolerance  
 Based on Model  
 Default

Approximate Maximum  
Model Dimension:  
10.0

Analysis Code:  
MSC/NASTRAN ▼

Analysis Type :  
Structural ▼

OK Reset

**Figure 2**

Now load the data file by selecting the *Analysis* option from the menu bar. The Analysis form is shown in Figure 3. From the Analysis form select the following properties:

- Action = Read Output 2
- Object = Both
- Method = Translate

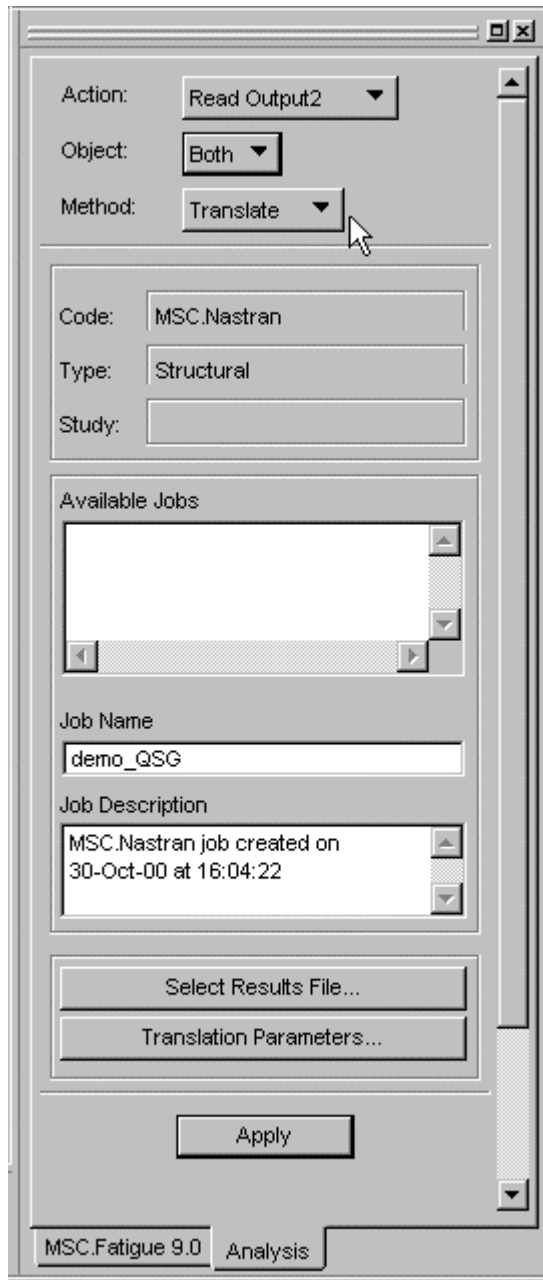


Figure 3

Click **Select Results File** and select the OP2 file, seamw.op2. It must be noted that this .op2 file contains the results from a NASTRAN run using the stress(cubic) case control command and PARAM,SNORM,45. These are required to extract correct nodal results at all nodes.

Click Apply and wait for the model to load.

### Creating a Weld group:

In order to carry out the fatigue analysis of the weld we need to create two groups, one containing the weld elements and the other containing the plate elements. To demonstrate the unique group extraction capability of this module, we will only create the weld group and use the default group for the plate group.

First, let's create a group containing the weld elements. From the main menu select **Group / Create** and fill in the form as below:

- New Group Name = weld
- Group Contents = Add Entity Selection
- Entity Selection = el 1078:1105

Click on **Apply** to define the group, the group should be highlighted in the plot.

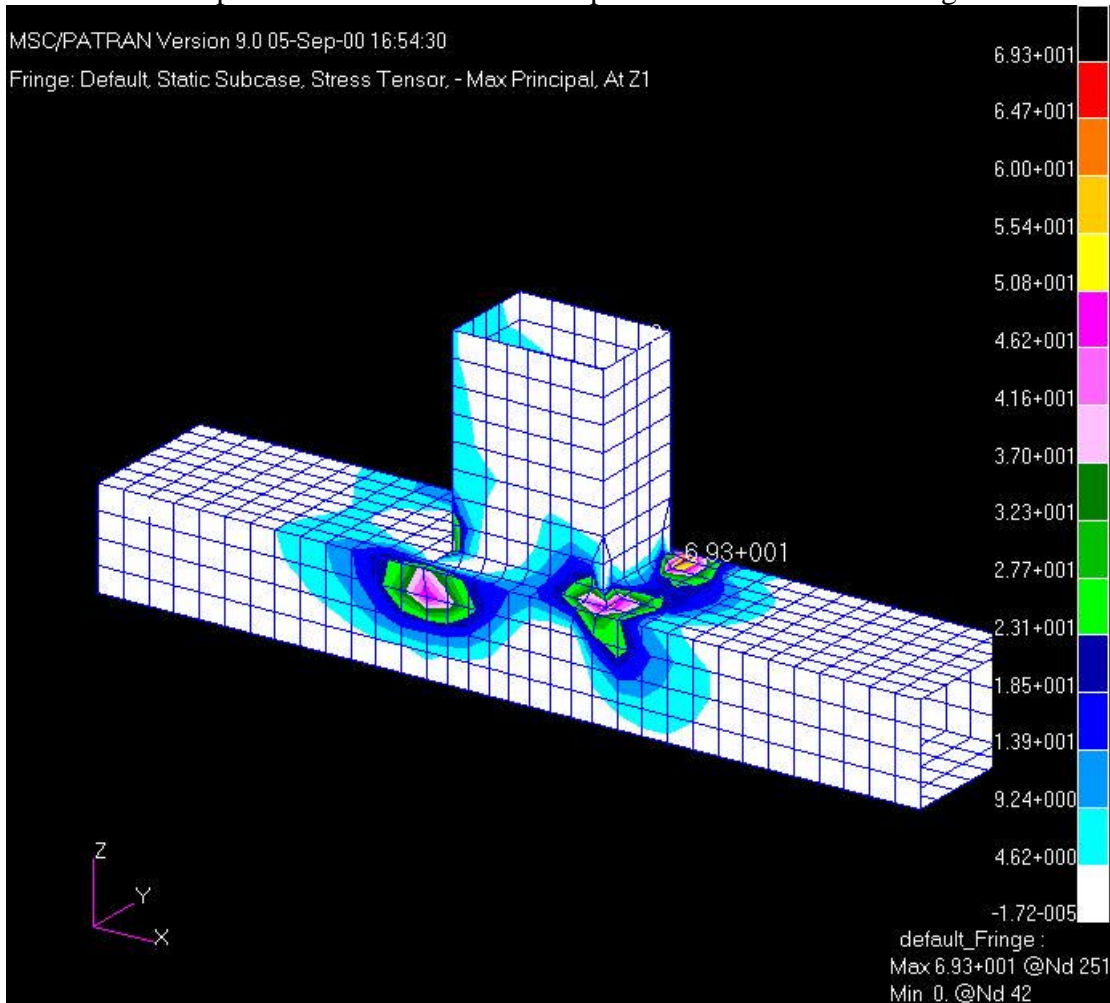


The image shows a software dialog box titled 'Create'. At the top, the 'Action' dropdown is set to 'Create'. Below this, the 'Current Viewport' field contains 'default\_viewport'. A 'Filter' field contains an asterisk '\*'. A list box titled 'Existing Group Names' contains 'default\_group' and 'weld'. The 'New Group Name' field contains 'weld'. There are two checkboxes: 'Make Current' (checked) and 'Unpost All Other Groups' (unchecked). The 'Group Contents' dropdown is set to 'Add Entity Selection'. The 'Entity Selection' field contains 'el 1078:1105'. At the bottom, there are two buttons: '-Apply-' and 'Cancel'.

**Figure 4**

### 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. The Maximum Principal stresses are shown in Figure 5.



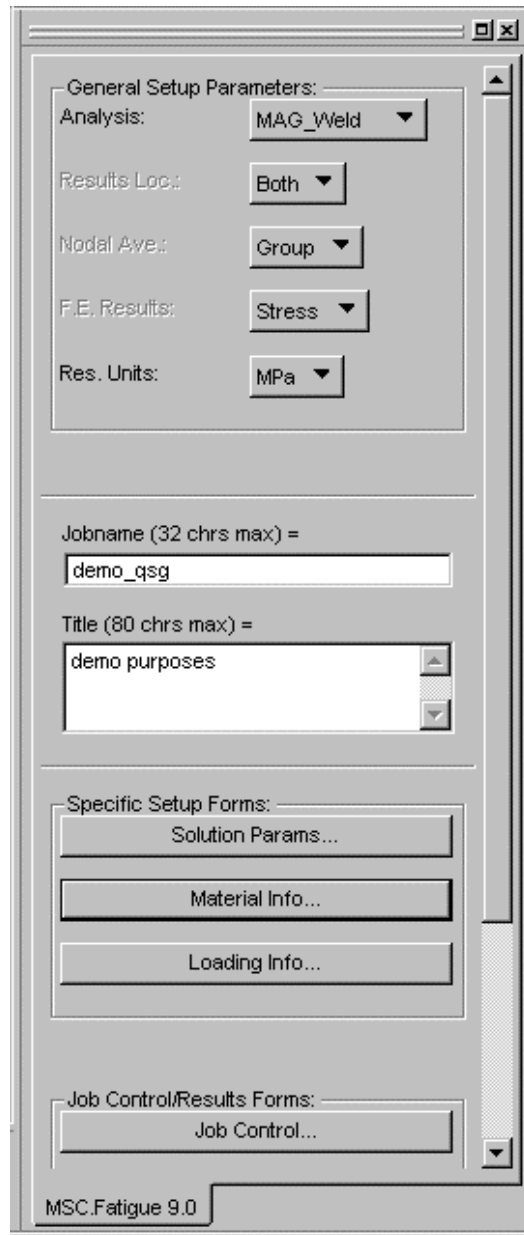
**Figure 5**

### Setting Up the Fatigue Analysis:

To set up a fatigue run select **Tools / FATIGUE**, from the main menu. Add the following properties to the form:

- Analysis = MAG\_Weld
- Jobname = seamw1

The form is shown in Figure 6.



The screenshot displays the 'General Setup Parameters' dialog box in the MSC.Fatigue 9.0 software. The dialog is organized into several sections:

- General Setup Parameters:** This section contains five dropdown menus:
  - Analysis: MAG\_Weld
  - Results Loc.: Both
  - Nodal Ave.: Group
  - F.E. Results: Stress
  - Res. Units: MPa
- Jobname (32 chrs max) =**: A text input field containing 'demo\_qsg'.
- Title (80 chrs max) =**: A text input field containing 'demo purposes'.
- Specific Setup Forms:** This section contains three buttons:
  - Solution Params...
  - Material Info...
  - Loading Info...
- Job Control/Results Forms:** This section contains one button:
  - Job Control...

The version number 'MSC.Fatigue 9.0' is visible in the bottom-left corner of the dialog box.

**Figure 6**

Note that results location, nodal averaging and FE results are grayed out. The results locations are at nodes, averaging is performed using the elements, except the weld elements connected to an analysis node and only FE nodal stress results are permitted.

We now have to set up the specific fatigue parameters.

### Solution Parameters:

Click on the *Solution Parameters* button. Set the *Mean Stress Correction* to OFF. The form is shown in Figure 5.

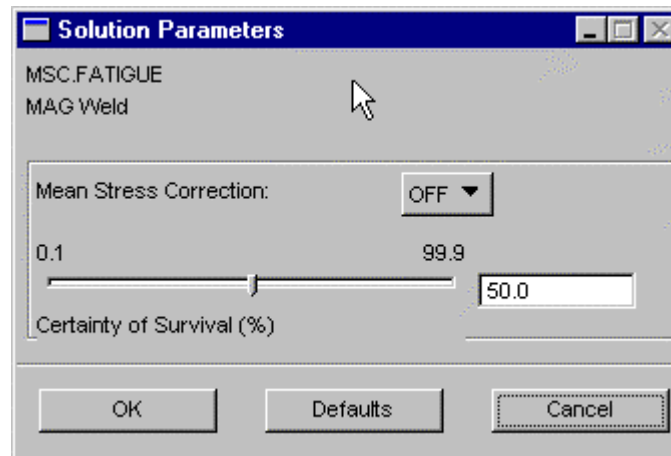
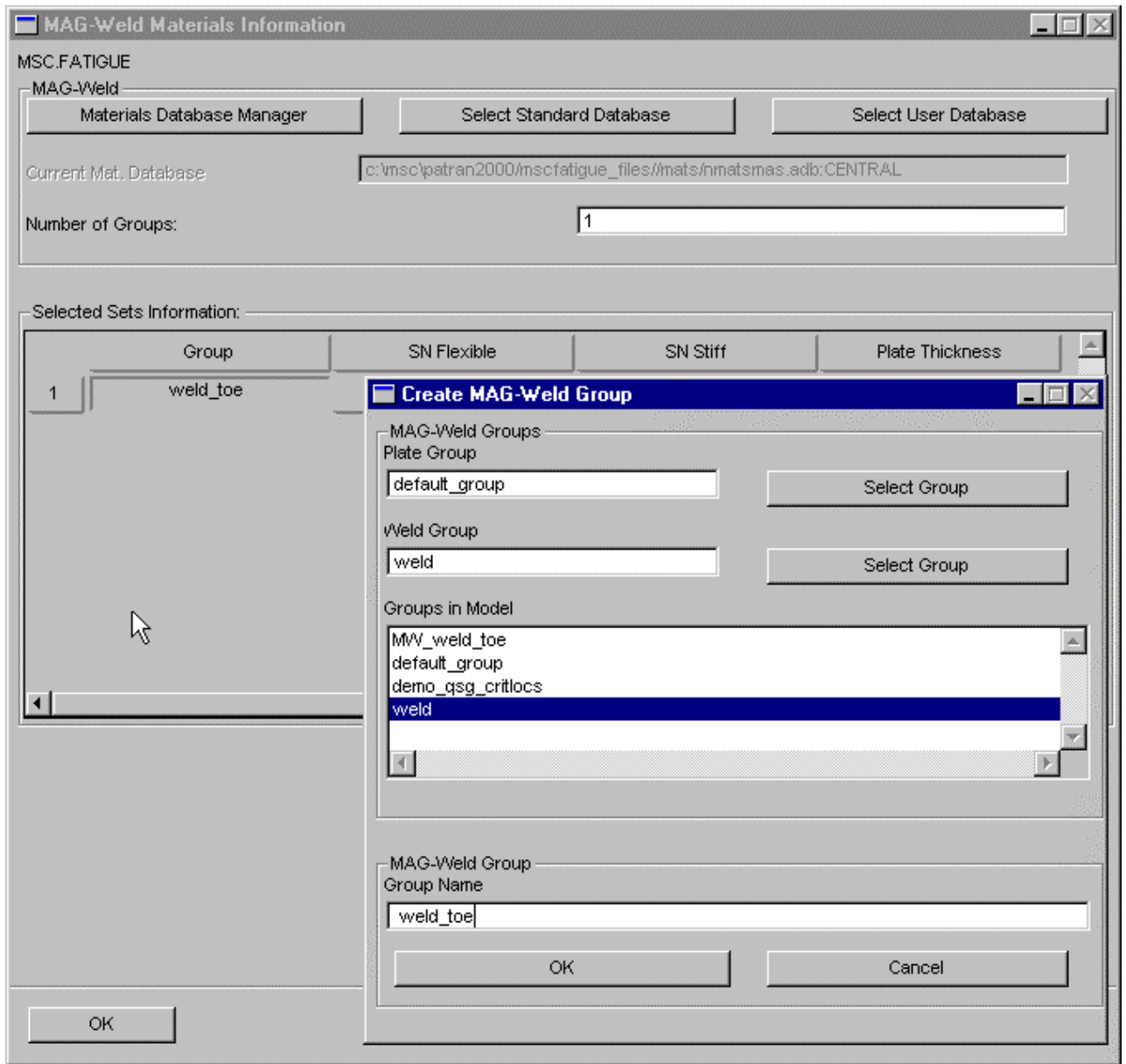


Figure 5

### Material Information:

This form is used to create the seam weld analysis groups and select the fatigue properties of the weld material. Fatigue analysis is carried out at the toe of the weld using the nodal stresses. To make life easier for the user, the software will automatically create a group for these nodes. The user needs to enter the weld group and the plate group (or the default group) and the software creates a group of the shared nodes between these. From the materials form, click on the *group* entry. The form titled, "Create Weld Group" should now appear and this is shown in Figure 6. Select the appropriate weld and plate groups and create a new group titled weld\_toe. (N.B. internally MSC.Fatigue appends "MW\_" to the beginning of this name.)



**Figure 6**

A new group containing the nodes along the weld toe is now posted on the materials form. Complete the form by assigning the following properties:

- SN Flexible = SEAM\_WELD\_FLEX
- SN Stiff = SEAM\_WELD\_STIFF
- Plate Thickness = 1.5
- M1/M2 Ratio value = 2.5
- Multiplier = 1.0
- Offset = 0

The completed form is shown in Figure 7.

MSC.FATIGUE

MAG-Weld

Materials Database Manager      Select Standard Database      Select User Database

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

Number of Groups: 1

Selected Sets Information:

Group	SN Flexible	SN Stiff	Plate Thickness	
1	weld_toe	SEAM_WELD_FLEX	SEAM_WELD_STIFF	1.5

OK      Defaults      Cancel

**Figure 7**

Click Ok to continue. The created group is automatically posted as shown in Figure 11. Note that *only the toe elements* are extracted which is the unique feature of this module.

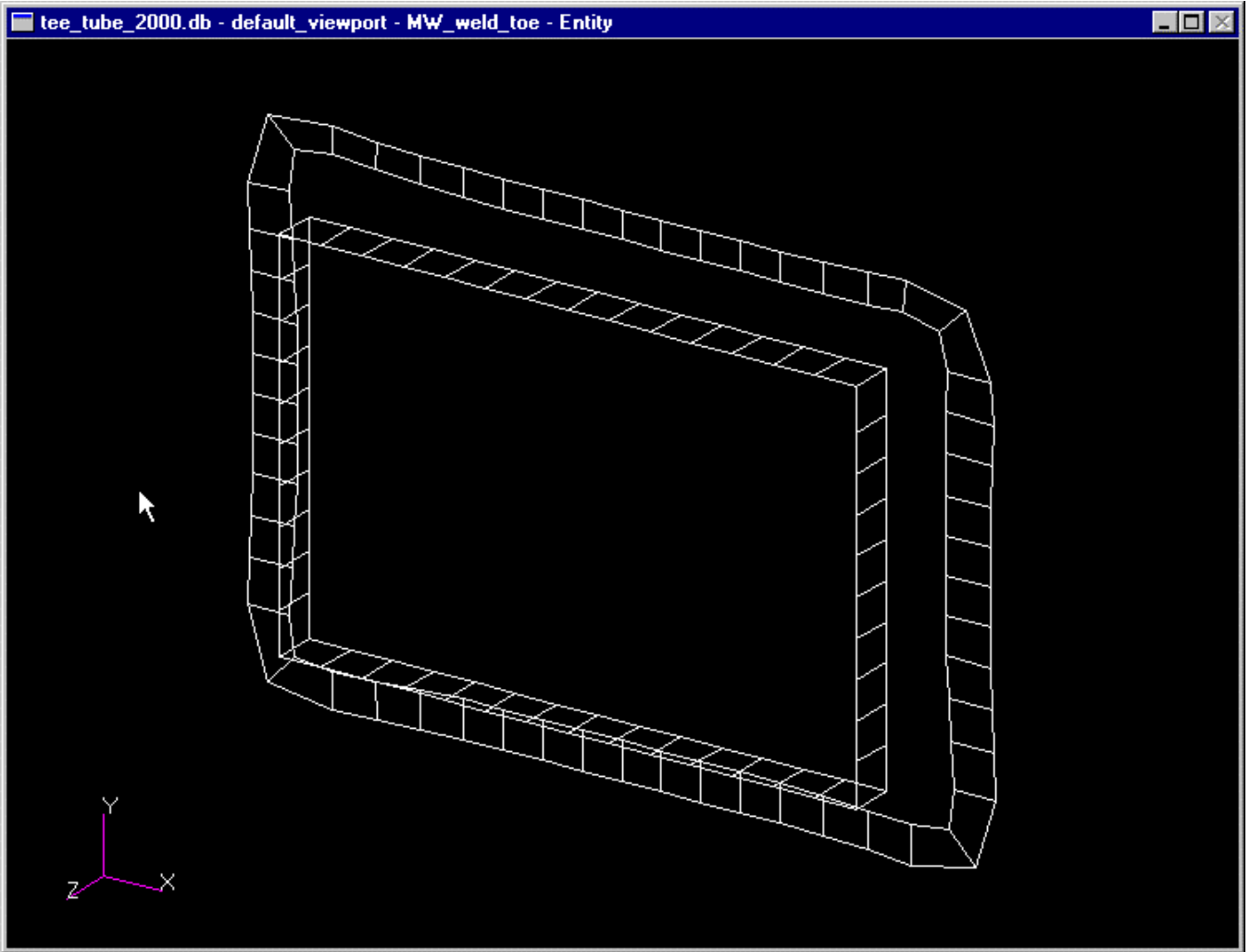


Figure 8

## Loading Info:

Click the Loading Info Button to see the loading form shown in Figure 9.

The screenshot shows the 'Loading Information' dialog box for MSC/FATIGUE. The window title is 'Loading Information'. The main area is divided into several sections:

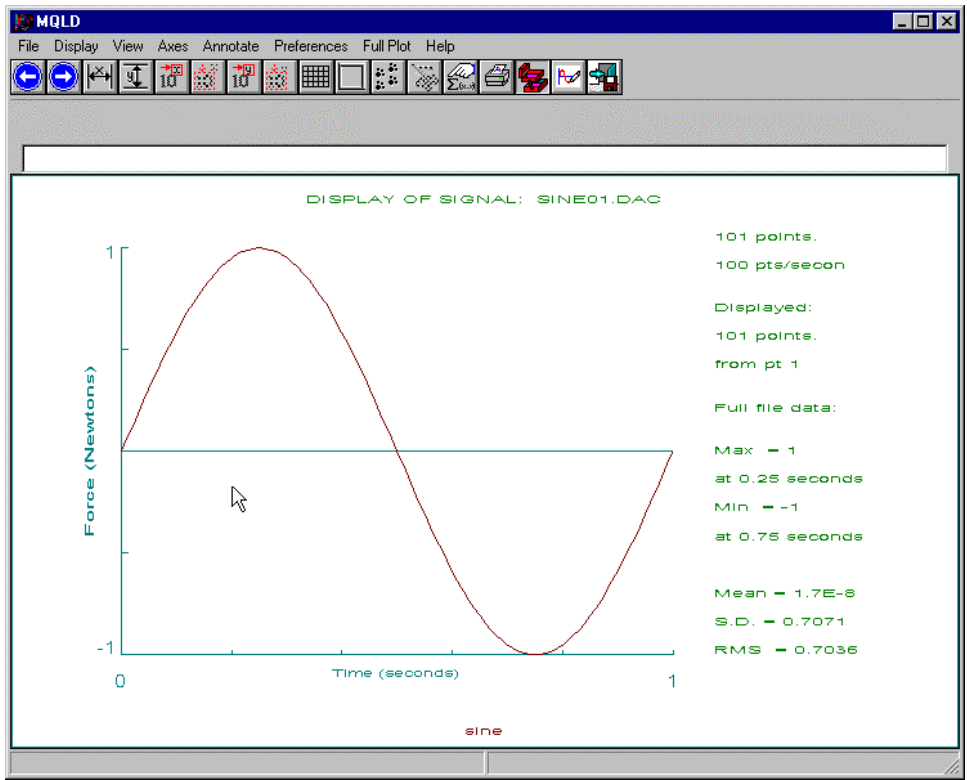
- Results Parameters:**
  - Results Type: Static
  - Results From: Database
  - Surface: Top
  - Strain Type: Tensor
- Results Transformations:**
  - Transform to Basic
- Directories:**
  - Time History Manager
  - Select Standard Directory
  - Select User Directory
- Current Time Directory:** /
- Number of Static Load Cases:** 1
- Fill Down OFF
- Selected Static Load Cases:**

Load Case ID	Time History	Load Magnitude
1		

At the bottom of the dialog are three buttons: OK, Defaults, and Cancel.

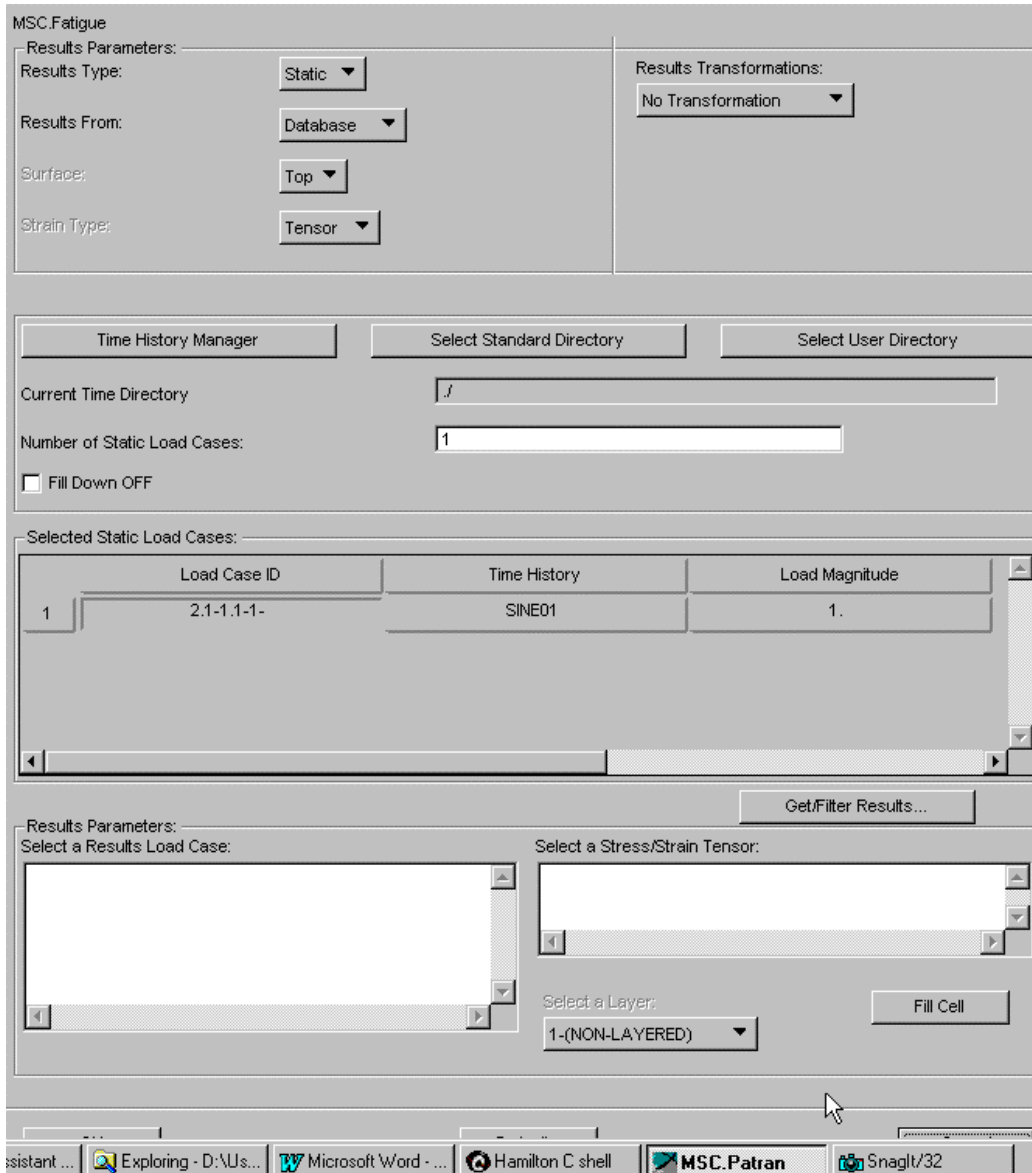
**Figure 9**

This form allows us to specify the loading environment. In this example we have carried out a quasi-static FE analysis on a single load case with a point load of 394N. We now need to determine how long the weld will last when subjected to a time varying load with a maximum of 394N. The input time history will be scaled proportional to that in the file titled sine01.dac, this is illustrated in Figure 10.



**Figure 10**

From the Loading Info form, Figure 11 click on *Load Case ID*, the following section is added to the bottom of the form:



**Figure 11**

Click **Get/Filter results** and tick the '**Select All Results Cases**' button. Click **Apply** and notice the FE load case appear in the form.

Select the load case and the stress tensor and click on **Fill Cell**, the load case is now entered into the table.

For the time history entry select SINE01 from the table.

Complete the table with the following properties:

- Load Magnitude = 1.0
- Scale Factor = 0.5
- Offset = 0

The completed form is shown in Figure 12.

Click **Ok** to submit the loading information.

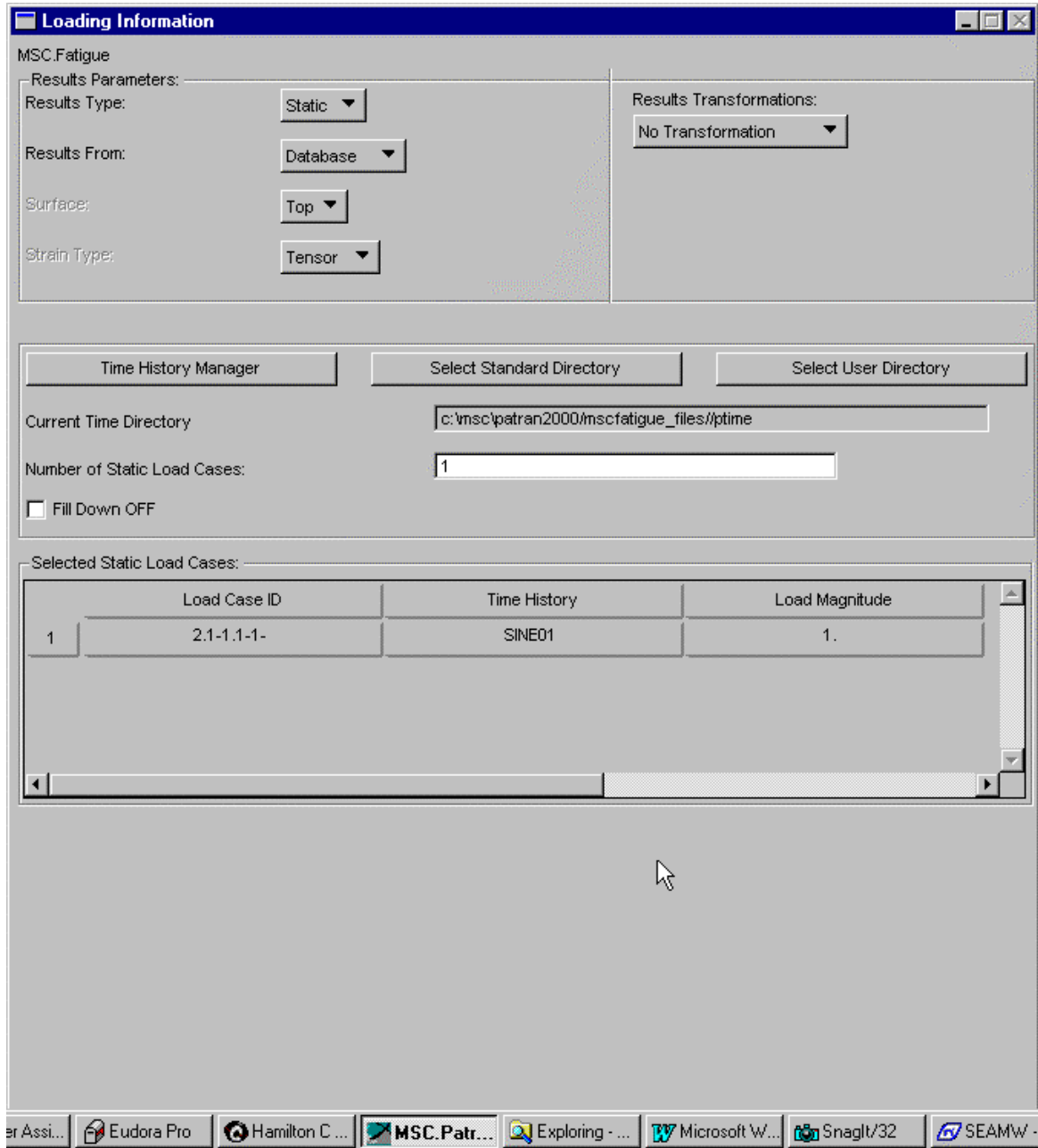


Figure 12

### 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.

### Listing the Fatigue Results:

Click on the **Results** button and select Action = **List results** from the form. Click on **Ok** until the following results summary form appears. The worst damage occurs at node 87 with 5236 repeats to failure. Click **Ok** and **exit** from the results listing program.

Node	Damage	Life Repeats of history	Life Repeats	Log Life	Log Damage	Log eq.	Stress state	R-ratio average
87	1.91E-4	5236	5236	3.719	-3.719	3.719	Stiff	0.4295
995	1.873E-4	5340	5340.4	3.7276	-3.7276	3.7276	Flexible	0.9646
129	1.74E-4	5749	5748.6	3.7596	-3.7596	3.7596	Flexible	0.9098
21	1.148E-4	8710	8710.3	3.94	-3.94	3.94	Stiff	0.2614
1025	1.098E-4	9105	9105.1	3.9593	-3.9593	3.9593	Flexible	0.9639
119	6.04E-5	16555	1.6555E4	4.2189	-4.2189	4.2189	Flexible	0.8968
605	4.526E-5	22093	2.2093E4	4.3443	-4.3443	4.3443	Stiff	0.4214
539	3.269E-5	30593	3.0593E4	4.4856	-4.4856	4.4856	Stiff	0.3725
629	3.012E-5	33198	3.3198E4	4.5211	-4.5211	4.5211	Flexible	0.9126
619	2.919E-5	34262	3.4262E4	4.5348	-4.5348	4.5348	Flexible	0.9104
1072	1.705E-5	58638	5.8638E4	4.7682	-4.7682	4.7682	Flexible	0.9668
151	1.277E-5	78336	7.8336E4	4.894	-4.894	4.894	Flexible	0.939
968	1.271E-5	78669	7.8669E4	4.8958	-4.8958	4.8958	Flexible	0.9678
139	7.418E-6	1.348E5	1.348E5	5.1297	-5.1297	5.1297	Flexible	0.9341
88	5.087E-6	1.966E5	1.966E5	5.2935	-5.2935	5.2935	Stiff	0.4624
1	3.097E-6	3.228E5	3.2284E5	5.509	-5.509	5.509	Stiff	0.4701
651	1.815E-6	5.51E5	5.5103E5	5.7412	-5.7412	5.7412	Flexible	0.9318
639	1.783E-6	5.609E5	5.609E5	5.7489	-5.7489	5.7489	Flexible	0.9886
606	1.622E-6	6.165E5	6.1651E5	5.7899	-5.7899	5.7899	Stiff	0.4489
528	1.418E-6	7.054E5	7.054E5	5.8484	-5.8484	5.8484	Stiff	0.4559
999	2.322E-7	2.001E6	2.0011E6	6.4773	-6.4773	6.4773	Flexible	0.9377

Figure 11

### 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.

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

- Select Result Cases = Seam Weld Analysis, seamw1
- Select Results = Log of Life ( )
- Click Apply

The seam weld fatigue fringe plot is shown in Figure 12. Results may also be displayed with Marker plots in Insight.

The worst life results is shown as  $\log(3.7)$ ,  $\approx 5236$  repeats to failure.

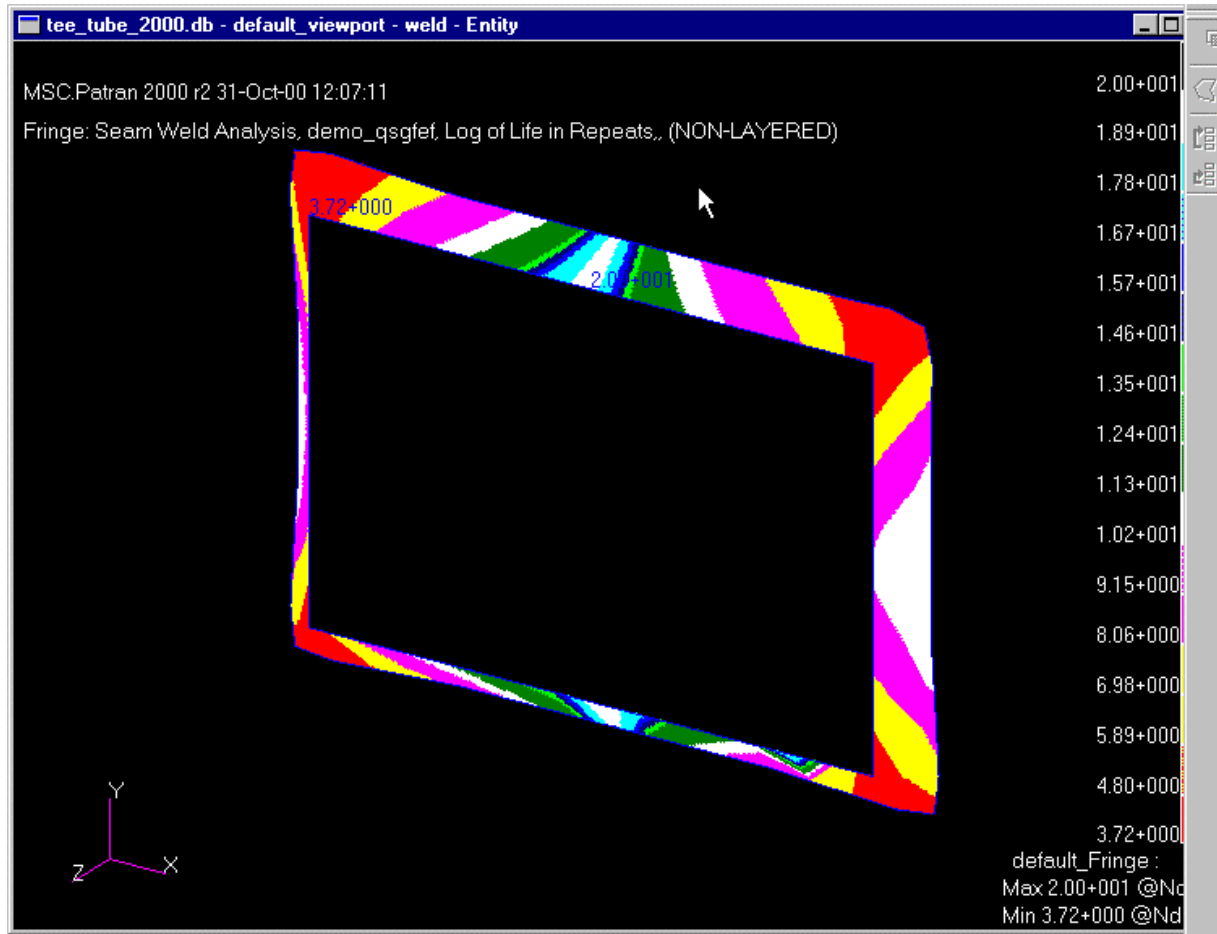


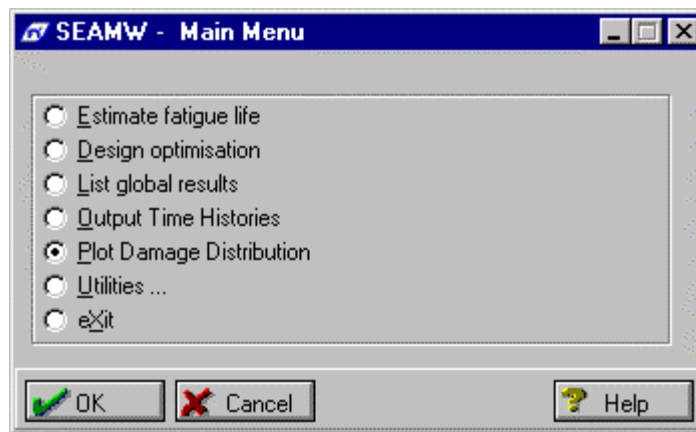
Figure 12

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

Apart from the normal output (items 1 – 7) the Mag-Weld module provides statistical information on the stress state at each analysis node (items 8-11). Items 2 – 11 can be plotted from the Mag-Weld main menu (figure 13).

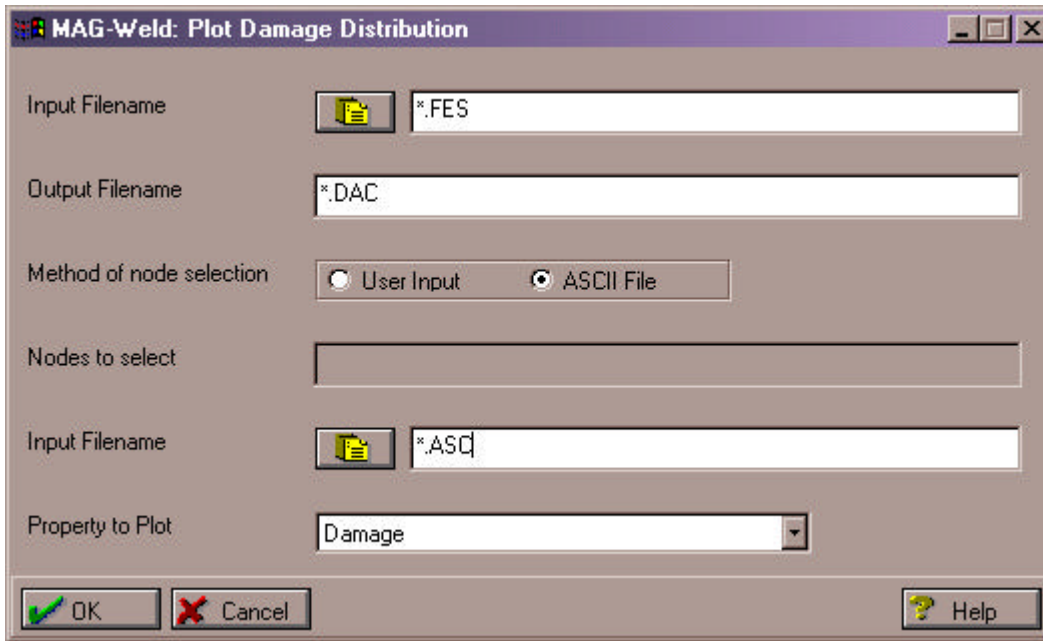
1. Node
2. Damage
3. Life repeats of history
4. Life in Repeats
5. Log Life
6. Log Damage
7. Log Life in Equivalent units
8. Stress State (Stiff or flexible)
9. r-ratio average (or stress ratio)
10. r-ratio range (or stress ratio)
11. r-ratio standard deviation (or stress ratio)
12. Element thickness

### **Plots from the MAG-Weld Main Menu:**



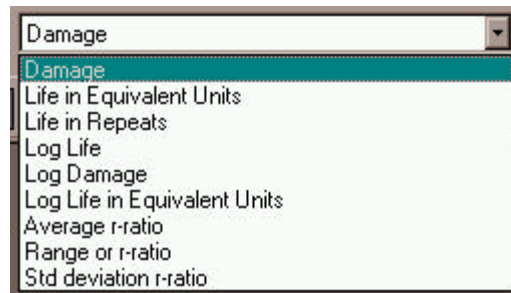
**Figure 13**

Selection of the Plot Damage Distribution produces the GUI shown in Figure 14. Nodes can be listed using standard input, e.g. (1 100 2), or alternatively from an ASCII file where each node number is listed on a separate line in the required order.



**Figure 13**

The available properties are those in the FEF file and are shown in Figure 14.



**Figure 14**

The software then reads the list and extracts the required values from the FEF file and creates a DAC file for plotting in MQLD. The x axis increment is unity.