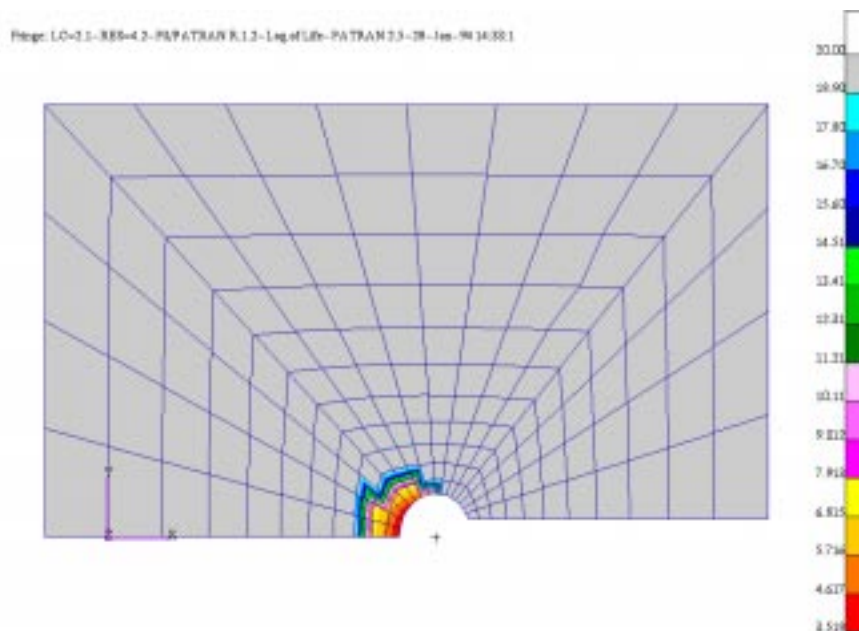


## LESSON 6

# *Crack Initiation Analysis of a Plate Assembly*



## Objectives

- To calculate the life to crack initiation of a plate assembly made of SAE1045\_450\_QT material undergoing a simple loading with a peak load of 22 KNewtons.
- To investigate design alternatives to achieve the desired design life of 10,000 operations. The alternatives considered are geometric and material changes.



## Problem Description:

The component we are designing is a bracket assembly which holds together two components. An elliptical connecting bar passes through the centers of the connecting parts. The bar does not apply any loading itself and is a clearance fit in the elliptic hole. The FE analysis was carried out to simulate the load applied to the plate during normal operation. (a force of 30 kN was applied in the Y direction).

A crack initiation calculation analysis must be carried out to ensure that the component meets the design life of 10,000 operations of the loading cycle depicted in Figure 1.

For proper execution of this exercise, make sure you are in the **ex06** directory of your PAT318 account. The P3/FEA results are stored under **plate1.res**, which again must be translated from **plate1.txt** using RESTXT. (While you are at it, convert **plate2.txt** also. You will be using it later also.) Enter PATRAN 3 and read the **plate1.out** Neutral file.

---



---

*Step 1 Problem Description*

## Exercise Procedure

Enter PATRAN 3, read in the neutral file and look at the stresses.

Q1: At what node does the highest stress lie and what is the stress?

A1:

\_\_\_\_\_

\_\_\_\_\_

\_\_\_\_\_

---



---

**Step 2 Time History Information**

The fatigue loading consists of a time history as shown in FIGURE 6.:

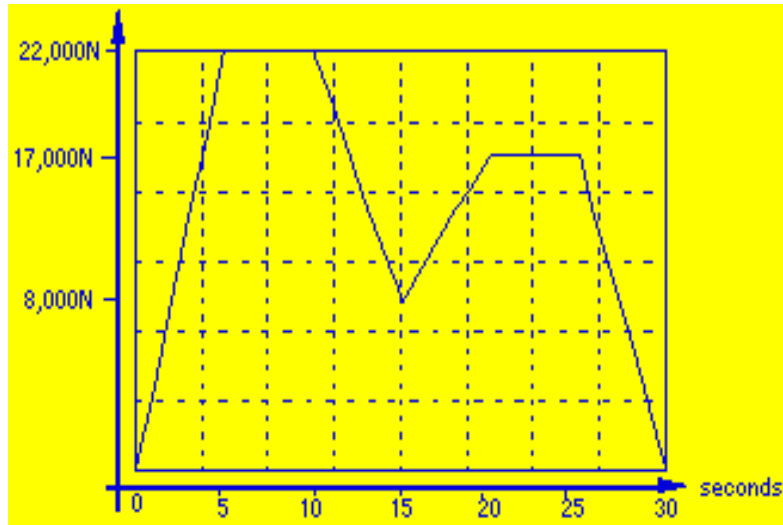


FIGURE 6. Loading Time History.

Create this loading history using PTIME. Use the **Enter X-Y points** option on the **Add** sub-menu. Set the equivalent units to 1 operation and call the time history PLATELOAD. The following are the steps it will take to create the time history. From the P3/FATIGUE main form:

<b>Loading Info...</b>	Open the Loading Information form.
<b>Static</b>	The results are static
<b>P3/FEA</b>	The results code is P3/FEA
<b>N/A</b>	Shell Surface is not applicable.
<b>plate1</b>	Select the P3/FEA job with the stress values for the fatigue analysis.
<b>Database Manager</b>	This will spawn the PTIME module.
<b>Enter X-Y points</b>	Create by inputting X-Y points.

This puts you in the data record creation window. Then input the following data:

Filename	<b>PLATELOAD</b>
Description 1	<b>EX6</b>
Description 2	<b>Optional.</b>

## LESSON 6

---

Load type	<b>Force</b>
Units	<b>Newtons</b>
# of fatigue units	<b>1.0</b>
Fatigue equiv unit	<b>Operations</b>
Sample rate	<b>1</b>

Press **F1** to accept the data then enter the Y values. Note that the time (X values) are not important.

0	1st Y value.
22000	2nd Y value.
22000	3rd Y value.
8000	4th Y value.
17000	5th Y value.
17000	6th Y value.
0	Last Y value.

Press **F1** key to accept the data.

Plot the PLATELOAD time history, if you so desire, to verify that it is correct. When you are satisfied with this step, exit from PTIME back to the P3/FATIGUE loading information form in PATRAN.

1	Set the Number of Static Load Cases to one.
1	Load case number in P3/FEA job.
<b>PLATELOAD</b>	Enter the modified version of the SAETRN time history with the type of Force and Units of Newtons.
<b>30000</b>	In the Load Magnitude databox, enter the value of one.
<b>OK</b>	Click on the OK button to close the form.

The first choice of material is SAE1045\_450\_QT since it is already used for another component in the assembly, it can be easily machined and it may be heat treated. The 450

---

---

**Step 3 Materials  
Information**

signifies that the Brinell hardness number and the QT stands for quenched and tempered. The surface finish from the machining is GOOD MACHINED.

There is no surface treatment. Since the material already exists in the materials database it is not necessary to actually run PFMAT, the materials database manager.

At this point, we may set up the P3/FATIGUE job by filling out the P3/FATIGUE main form.

---



---

**Step 4 Setting up the P3/FATIGUE job**

<b>Crack Init.</b>	The <i>Analysis</i> type that we are performing is Strain Life or crack initiation analysis
<b>Nodal</b>	Use stress <i>Results</i> at nodes.
<b>Stress</b>	Use stress <i>Tensor</i> of the results.
<b>MPa</b>	Stress <i>Units</i> as prescribed from the P3/FEA job are MPa
<b>plate</b>	Enter the <i>jobname</i>
	Use the Title to give a description of the job. (E-N analysis of the plate assembly)
<b>Solution Params...</b>	Click on the Solution Params button in the P3/Fatigue form.
<b>S-T-W</b>	Set the <i>Mean Stress Correction</i> option menu to Smith-Topper-Watson.
<b>50</b>	Set the <i>Design Criterion (%)</i> to 50. Click on the OK button to close the form.

Now the job setup is complete. Submit the job as follows:

---



---

**Step 5 Submitting the P3/FATIGUE job**

<b>Job Control</b>	Click on the Job Control button in the P3/FATIGUE form.
<b>Full Analysis</b>	The Action option menu should be set to Full Analysis.
<b>Apply</b>	Submit the analysis job.

The job is now submitted and the job progress can be monitored using the **Monitor Job** option. Note that various processes will be executed and the whole job is not complete until the **Fatigue analysis completed successfully** message appears.

To review results, you must first click on the results button and then select List Results option. Click on the **Apply** button. Enter the design life of 10,000 Operations when the P3FATIGUE module PFPOST appears and then display the **Most damaged nodes** as done in previous exercises.

---

---

*Step 6 Reviewing Results*

Q2: What is the life of node 12? Is our design life of 10,000 repeats met?

A2: \_\_\_\_\_  
\_\_\_\_\_  
\_\_\_\_\_

We can examine the extent of the damage zone by carrying out a contour plot. This shows the damage zone is concentrated around the notch. Exit PFPOST. Read the results file in by changing the Action option menu to Read Results. To view a fringe plot of the life estimates, you must click on the Results toggle in the Main Window.

**Log of Life, Ops** Pick **Log of Life, Operations** from the Select Results listbox.

**Plot** Make the fringe plot.

Q3: What options are there to improve the life?

A3: \_\_\_\_\_  
\_\_\_\_\_  
\_\_\_\_\_

We can carry out an investigation of alternative geometries by suggesting that the local stress concentration could be reduced by a given factor. Go back the P3FATIGUE main menu and select the **Optimize** option. This will start the design optimization program for strain life or crack initiation analysis.

---

---

*Step 7 Design Optimization*

Q4: What scaling factor will meet our design life of 10,000 repeats of the loading? Do you remember what option will do this for you.

A4: \_\_\_\_\_  
\_\_\_\_\_  
\_\_\_\_\_

We have already modified the notch geometry, replacing the elliptic notch with a circular one. The PATRAN Neutral file for the new model is **plate2.out** and the P3/FEA results are stored under **plate2.res**. The \*.txt files must be translated by PATRAN's utility program RESTXT from the file **plate2.txt** as done before with **plate1.txt**. Quit from DESOPT and close this database. Open the new database and import the neutral file. Examine this model and the stresses computed. A new P3/FATIGUE analysis should be setup and submitted.

**File/New Database** Initialize new database.

**Plate2.db** New database name.

**File/Import** Invoke neutral file utility.

**plate2.out** Neutral file name.

You may look at the stress results again if you wish. The node with the highest stress is now Node 13.

**P3/FATIGUE** Enter P3/FATIGUE menus.

Define Job menu. If the default jobname is **plate**, use it. If no default is available and it asks you to define a job name, type **plate** and use the existing parameters. We will be using the existing job setup with some minor changes.

All job setup parameters are already defined. You may see this by opening the menus at this same menu level.

**plate2** The only entry that needs to be changed is the file name containing the FE results. The new file for results is denoted as **plate2.res**.

**Job Control** Click on the Job Control button in the P3/FATIGUE form.

**Full Analysis** The Action option menu should be set to Full Analysis.

**Apply** Begin the analysis submit.

The job is now submitted and the job progress can be monitored using the **Monitor Job** option. Note that various processes will be executed and the whole job is not complete until the **Fatigue analysis completed successfully** message appears.

---

*Step 8 Submitting the P3/FATIGUE job*



Look at the results as you did with the previous model, **plate1**. Note that the results from the previous fatigue analysis have been overwritten. If you do not wish to do this then you should use separate jobnames.

Q5: What is the life at the critical node? What is the node ID. Can you determine this node without contouring or listing results?

A5:

---

---

---

The estimated life is significantly above the design life so that we will now consider options available to reduce the cost of manufacture. Using DESOPT, the design optimization module, re-analyze using an alternative material. Try the materials SAE1561\_234\_HR and SAE1018\_106\_HR. These materials are hot rolled, not heat treated and are cheaper than SAE1045\_450\_QT.

Q6: How do you accomplish this using DESOPT. Are both of these materials suitable for this plate?

A6:

---

---

---

Try using the **search for a Worse material** option under the **Material optimization** main menu pick to see other possible materials that will meet the design life.

It has been suggested that the machining will not always be of the highest quality. Reset all the original parameters by selecting the **Original parameters** option from the main menu. Then answer the following:

Q7: What is the effect on life of an AVERAGE machining surface finish (machining defect) for material SAE1045\_450\_QT? How do you get DESOPT to tell you this?

A7:

---

---

---

Q8: Use the BACK life calculation facility to obtain the largest  $K_t$  that is permissible for the design life. How is this done?

A8:

---

---

---

Just to be safe, to improve the wear characteristics in the notch, nitriding will be used. Use an AVERAGE machined surface.

Q9: What is the effect of nitriding on fatigue life? How do you get DESOPT to do this?

A9:

---

---

---

Quit from DESOPT and from PATRAN when you are through. Note that number of files have been created under your job heading. They are essentially the same for crack initiation as for the total life method, i.e., (.fin, .fnf, .fes, .fpp, .fef, .msg, .sta)

- A1: Node 12, stress = 2085 MPa (Max Principal).
- A2: Approx. 3500 operations of the loading history, NOT ENOUGH! (design life is 10,000 operations)
- A3: There are a number of modifications which we can investigate. Alternative geometry at the notch, different material, different surface finish and/or treatment.
- A4: When the DESOPT module appears, first enter the design life of 10,000 Operations and have the program automatically extract the worst case node. Press the **F1** key when the first screen is filled out properly. Note that the same answer is given. Press the **return** key and you are put into the DESOPT main menu. From here you can select **Parameter optimization**, **Scaling factor**, and then **Recalculate**. Using this back calculated scaling factor we find the factor would be approximately 0.85. This is small enough for us to consider reducing the stresses by changing the geometry.
- A5: You can determine this by using DESOPT directly and have it automatically pick the worst node and do the re-analysis. The critical node is node 13 and the life is approx. 1,000,000 operations.
- A6: Select **Material optimization** from the main menu and then **Material change** from the resulting submenu and type in the new material or use the list mechanism by pressing **F3** or click on **OK** on the top header bar. Don't forget to **Recalculate**. SAE1018\_106\_HR has a critical node life of approx. 8000 operations which is below the design life and therefore not suitable. The safety factor is low when considering the scatter in fatigue failures.
- A7: After resetting all the original parameters, use the **Sensitivity analysis** option and turn on all **surface Finishes**. An average machined finish which could occur as the cutting tool wears will reduce the life by more than half to approx. 349,000 operations, but still well above the design life.
- A8: Again reset the original parameters and then pick **Parameter optimization** and then **Scaling factor**. A stress concentration factor of 1.68 would reduce the life estimate to 10,000 operations. In other words, a defect probably could not easily reduce the life to an unacceptable level.
- A9: Reset the original parameters, and this time simply select **Change Parameters** from the main menu and select **surface Finish** and change it to **AVERAGE**, then do the same thing and select **surface Treatment** this time and change it to **NITRIDED**. Nitriding the component with an average machined surface finish extends the life which is an attractive option since

we are well within the design life. As a matter of fact it gives it an infinite life (beyond cutoff).