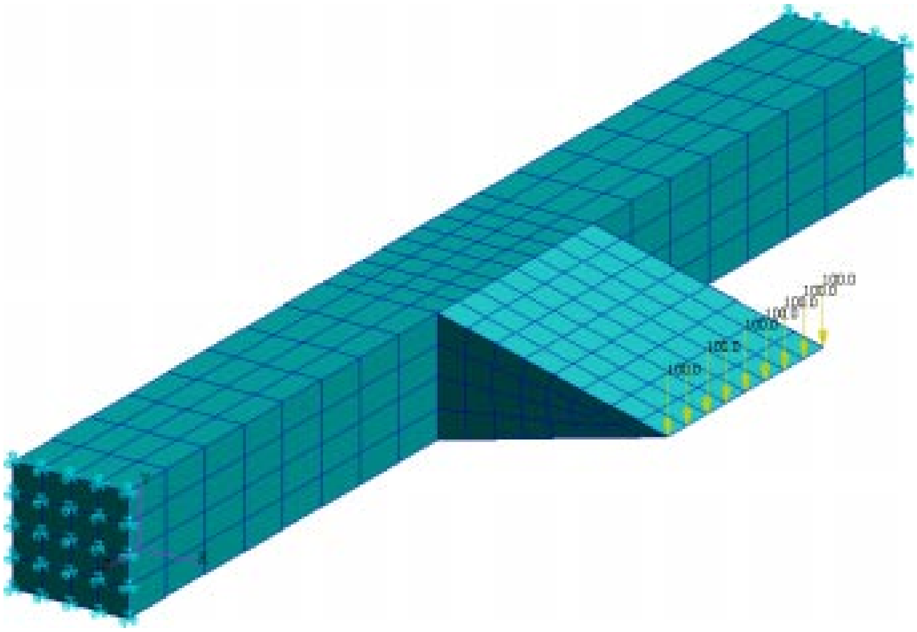
LESSON 4

Total Life Analysis of a Simple Bracket



Objectives:

- Use the S-N method to estimate fatigue life.
- Enter materials data into the database and create a simple time history.
- Review the P3⁄FATIGUE files and understand their functions.
- Perform Post-processing of P3⁄FATIGUE Results.

Problem Description:

This exercise is concerned with the fatigue analysis of a welded structure the dimensions of which are shown in FIGURE 4.

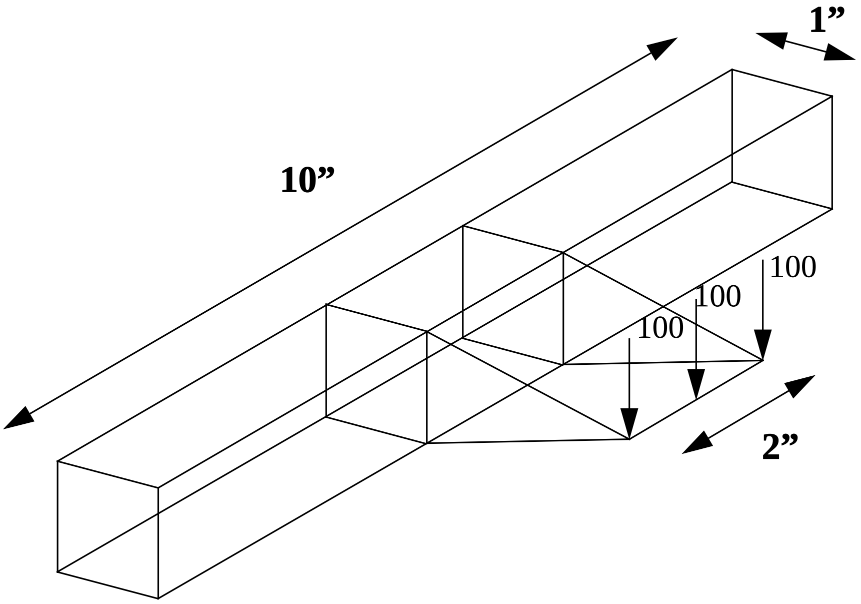


FIGURE 4. A Simple Bracket with Weld.

A simple bracket as shown above has a design life of 7 years (61320 hours). Loading occurs at the end of the short welded-on section and the component is constrained at both ends of the main bar. Because failure is known to occur at the weld, the FE modelling at the load point and the stresses found there can be ignored for the purposes of this exercise. The load applied in the model was 900 lbs. and the results obtained from P3/FEA in a file called **bracket.res**. (This file must be created from the file **bracket.txt** using the PATRAN utility RESTXT.) The component was tested under constant amplitude fully reversed conditions to produce S-N data as follows in Table 4-3.

For proper execution of the exercise, make sure you are in the **ex04** directory of your PAT318 account.

Table 4-3 Weld S-N Properties.

Material Properties	SI	Imperial
Young's Modulus, E (MPa,ksi)	205800	29850
UTS (MPa,ksi)	700	101.5
S-N Data	Component	
Stress Range Intercept, SRI1 (MPa,ksi)	10710	4680
Slope b1,	-.33333	-.33333
Transition life, NC1 (cycles)	1E7	1E7
Slope b2,	-0.2	-0.2
Fatigue Limit, FL (MPa,ksi)	10	1.45
Standard error, SE	0.2	0.2
R-Ration of test, RRAT	0.5	

The stress for the S-N curve was measured using strain gauges at a point one quarter of an inch from the weld on the main bar and 5 inches from each end of the bar.

In service the component experiences loading of 3000 lbs. in the direction of the FE loading and 7000 lbs in the reverse direction. This happens once every 30 minutes.

Only 4% failure rate should be allowed.

Exercise Procedure

Step 1 Using PATRAN

Enter PATRAN 3 and read the Neutral file **bracket.out** using the following procedure:

p3 Start PATRAN3 from the UNIX prompt.

File/New Database Open the File pull-down menu. Select New Database from the pull-down menu.

Bracket Enter the name Bracket in the New Database Name databox. Click on the OK button. When the New Model Preferences form appears, change the preference to P3⁄FEA, click on the OK button to close the form.

File/Import In the Neutral Files listbox, select the file bracket.out. Click on the OK button to begin the model import operation. Acknowledge the Question form application NEUTLOD.

Look at the stress results from P3/FEA. (This requires the knowledge of using PATRAN 3. Ask the instructor if you have difficulty.) Also plot the forces and constraints. You will need to change the view also. We have provided the following steps for your guidance.

Analysis Open the analysis form.

Read Results Set the Action option menu to read results.

Select Results File In the Select File form, select the file bracket.res from the Available Files. Click on the OK button to close the form.

Apply Read the results file into the database.

Load/BCs Open the Load/Boundary Conditions form. Change the Action option menu to Plot Markers.

Select all the Load/BC sets to be plotted to the default_group. Click on the Apply button.

Finally, use the Results Display form to examine the results and answer the following question.

Q1: Which node corresponds to the measurement point for the S-N curve?

A1:

____

____

____

Now that you have examined the stress results, you may proceed to set up a fatigue analysis job. To do that, you must select P3/FATIGUE from the Main Window.

Applications Open the Applications pull-down menu.

P3/FATIGUE Open the P3/FATIGUE form.

Mask In the Main P3⁄FATIGUE form, select mask for the Module Driver selection.

Q2: How do you input the S-N and materials data?

A2:

____

____

____

Q3: What type of analysis is this?

A3:

____

____

____

Step 2 Materials Data

S-N Comp. The *Analysis* type that we are performing is Material Nomial Stress Life

Nodal Use stress *Results* at nodes.

MPa Stress *Units* as prescribed from the FEA job are MPa

bracketsn Enter the *Jobname*

Use the Title to give a description of the job. (S-N analysis of bracket specimen)

Solution Params… Click on the Solution Params button in the P3⁄FATIGUE form.

Gerber	Set the *Mean Stress Correction* option menu to Gerber.
50	Set the *Design Criterion (%)* to 50.
OK	Click on the OK button to close the form.

Q4: Why are the units of stress important?

A4: ______________________________

____

____

Q5: What does “Design Criterion” mean?

A5: ______________________________

____

____

Q6: Where are the FE results from?

A6: ______________________________

____

____

Q7: Why is the value of the FE loading required?

A7: ______________________________

____

____

Now you will specify the material data given in Table 4-3.

Material Info…	Click on the Material Info button in the P3⁄FATIGUE form.
Database Manager	Click on the Database Manager button in the Materials Information form.

This will take you to material database management. This exercise of entering material data has been explored in the previous two exercises and is only described here. Do the following:

Find the **C**reate option. The program may ask for a password to enable entry into the protected database. You do not know this password, so it will create a local database which you may edit by using the **return** key (with no password). You may call your data set anything not already existing in the default database, ***but remember its name for later use***.

After creating the new material database, you will be put into the area where you may enter your own materials data. It consists of four pages (screens) of data entry. Input the name you want to call your material. The other fields in this window are optional. Press **F1** to accept the data.

The next page you will enter generic data in ***SI units***. Input the Young's Modulus of 205800 MPa and UTS of 700 MPa. Press **F1** to accept the screen data.

Since you will be performing a Total Life Analysis, you only need to input the stress data. This data is located in Table 1. After this data is inputted, press **F1** to accept the data. Press **F1** again when the LEFM screen is presented and leave everything blank. You will be told that the data has been entered into the database.

Q8: How would you display the S-N curve graphically?

A8:

____

____

____

When you are satisfied that the S-N curve is safely entered, exit the Materials Information form, and go back to the P3⁄FATIGUE menu in PATRAN.

OK Click on the OK button to close the form.

Now, you must exit PFMAT to return to PATRAN 3 for the remainder of the P3 job setup.

1 Set the Number of Materials to one.

<mat_name> Click on the *Material* table cell in the Selected Materials Information spread sheet. In the *Select a Material*: listbox, click on the material you just created. Your selection will be placed in the spreadsheet and its weld classification will be automatically entered.

Node 514	Enter Node 514 as the *Reference Node*. Type the <Return> key to accept the data.
default_group	In the *Select a Group* listbox, select the only group available; default_group.

Q9: Do you need to fill in all the fields?

A9:

____

____

____

Q10: How would you correct a mistake?

A10:

____

____

____

Q11: Which option do you use to describe the loading conditions?

A11:

____

____

____

Q12: What does “reference node” mean?

A12:

____

____

____

Q13: Which nodes should you select and how?

A13:

____

____

____

Hint: There are 3 hyperpatches on the main bar, in order 1, 3 and 2. The triangular hyperpatch is #4.

Step 3 Time History Information

At this time, you must select Loading Info… from the P3⁄FATIGUE main menu to specify the time variation of the loading. In the Loading information form, click on the Database Manger button to invoke PTIME module of P3⁄FATIGUE.

You can create the time history using the **Enter X-Y points** option under the **Add a time history** menu option. When the time history database is empty as it is when you first invoke PTIME in this exercise, you are automatically presented with the **Add at time history** submenu.

The **X-Y points** option allows you to type in the y coordinates for each turning or end point in the time history. Use **Y** values of **0,3000,-7000,0** for the time history and exit PTIME when finished.

The following are the detailed steps of time loading specifications.

Static	The *Results Type* is static
P3/FEA	The *Results From* is P3/FEA
N/A	Shell *Surface* is not applicable.
bracket	*Select the P3/FEA job* with the stress values for the fatigue analysis.
Database Manager	This will spawn the PTIME module.
Enter X-Y points	Use the X-Y points option.

Now fill out the information page with the following.

Filename	Use anything you want.
Description 1	Use anything you want.
Description 2	Use anything you want.
Load type	Load type is Force.
Units	Units are lbs force.
No. fatigue units	**0.5 -** this is equivalent to 1/2 hour since one repeat of the time history occurs every 30 minutes.
Fatigue equiv units	**Hours**.

Press the **F1** key to accept the screen. Next enter:

0	First Y value.
3000	Second Y value.
-7000	Third Y value.
0	Forth Y value.

Press the **F1** key to accept the screen. And return to the main PTIME menu.

Q14: What will the time history look like?

A14: ______________________________________

____

____

Q15: Which other options could create this time history?

A15: ______________________________________

____

____

Q16: What does “fatigue equivalent units” mean?

A16: ______________________________________

____

____

In our example, the design life is in years, but it is a good idea to make the number of fatigue equivalent units larger than this, so use hours. (This doesn't affect the actual calculation.)

Q17: How would you view the loading history graphically?

A17: ______________________________________

____

____

When you are satisfied that the loading history is OK, exit the loading history database manager, PTIME, and return to the P3⁄FATIGUE main menu.

On returning to PATRAN 3, select the time history desired for this run.

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

Step 4 Setting up the P3/FATIGUE job

You are now ready to set up the P3/FATIGUE job. Click on the Job Control button in the P3/FATIGUE main form. Submit a full analysis of the job. Enter the following keystrokes.

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 **Monitor Job** option in the Job Control form. Note that various processes will be executed and as such the whole job is not complete until the **Fatigue analysis completed successfully** message appears.

Q18: What important files are generated by a submitted job?

A18:

____

____

____

Step 5 Reviewing Results

Use the **Results** option from the P3⁄FATIGUE menu. Enter option **List Results**. Click on the **Apply** button. This time enter a design life of 61320 Hours when you are presented with the first screen of PFPOST and then list the **Most damaged nodes.**

Q19: What is the estimated life of the component?

A19:

____

____

____

Q20: What do the different colors indicate?

A20:

____

____

____

Exit this program. Create a fringe plot of the results. Try a fringe plot from the contour option.

Results	Click on the results toggle in the Main Window.
Estimated Life	In the *Select Result* listbox, select the result that corresponds to estimated life.
Plot	Click on the Plot button.

Q21: How do you interpret this plot in the context of a welded component?

A21:

____

____

____

Step 6 Design Optimization

At this stage you should have answered the first objective of the exercise. Now we can use the **Design Optimization** option from the P3/FATIGUE main menu to answer the rest of the questions.

P3/FATIGUE	Open the Main P3/FATIGUE form.
Results	Click on the Results button.
Optimize	Set the Action to Optimize.
Node 514	Select a Node to be Node 514.
Apply	Click on the Apply button.

Q22: Which node should you analyze?

A22:

____

____

____

Q23: If the defaults are taken, does the Design Optimization module, DESOPT, give the same answer as the global analyzer?

A23:

____

____

____

To calculate what load reduction is required to achieve a 7 year design life, use the Back Calculation option on scale factor. Use the following keystrokes or option picks from the main menu of DESOPT.

Parameter optimization...Pick the **Parameter optimization** option which will display a sub-menu.

Scaling factor — Pick the **Scaling factor** back calculation option. Note that above the menu picks it indicates that this option has been turned on.

61320 — Enter a design life if you haven't already from the initial screen presented to you.

Recalculate — Do a recalculation.

Q24: What reduction in load is required?

A24:

____

____

____

Reset everything and do a Back Calculation on design criterion to determine the failure rate after 7 years. Use the following keystrokes or option picks from the main menu of DESOPT.

Original parameters — This resets all **Original parameters.**

Parameter optimization...Pick the **Parameter optimization** option which will display a sub-menu.

design **C**riterion — Pick the **design Criterion** back calculation option. Note that above the menu picks it indicates that this option has been turned on.

Recalculate — Do a recalculation.

Q25: What would be the failure rate after 7 years?

A25:

____

____

____

Now look at the effect of mean stress correction.

Original parameters — This resets all **Original parameters.**

Sensitivity analysis... — Pick the **Sensitivity analysis** option which will display a sub-menu.

Analysis types (all) — Pick the **Analysis types (all)** option. Note that above the menu picks it indicates that this option has been turned on.

Recalculate — Do a recalculation.

Q26: What difference in life does the Gerber correction have?

A26:

____

____

____

You have submitted the report to your manager, which has caused panic, and have been asked to re-analyze the component after using a modified welding technique, which is more expensive. After re-testing, a new S-N data set has been generated as shown in Table 4-4.

Table 4-4 New S-N Dataset for Weld.

Material Properties	SI	Imperial
Stress Range Intercept, SRI1 (MPa,ksi)	13950	2023
Slope b1,	-.29	-.29
Transition life, NC1 (cycles)	2E7	2E7
Slope b2,	-0.16	-0.16
Fatigue Limit, FL (MPa,ksi)	10	1.45
Standard error, SE	0.14	014

The base material properties, E and UTS, are unchanged.

Enter this modified S-N data into the database just as you did with the first dataset. Load both the first and this new data set into the materials database manager.

Q27: What do you notice when comparing the two data sets graphically?

A27:

____

____

____

Q28: Predict the approximate change in life from the graph.

A28:

____

____

____

Go back to the P3⁄FATIGUE menu. Change the nodal properties to use the new S-N data. Resubmit the job.

Hints: For safety, delete all active nodes before adding them back. Also, don't re-use the neutral file.

Q29: Is the new life satisfactory?

A29: ____

____

____

Use the nodal analyzer to repeat the design optimization analysis for the modified component.

Q30: What is the amount of overload that this component can take with the new weld method? What is the failure rate after 7 years?

A30: ____

____

____

A1: Node 514.

A2: Enter Material Database Manager, PFMAT, followed by the **C**reate option.

A3: Nominal Stress Life, component.

A4: Because the materials data is stored in metric units, the FE stresses must be converted to make the calculations compatible. The software will automatically convert the data to compatible units based on the value given in the P3/FATIGUE **Job Setup**.

A5: The design criterion is expressed in terms of the percentage certainty of survival based on S-N data curve scatter. Failure rate can be determined by subtracting this value from 100. The design criterion is 96% (4% failure rate accepted).

A6: The FE results are from linear static P3/FEA analysis by the jobname of bracket. There are no shell elements and there is only one load case with ID of 1.

A7: Because the software normalizes the stresses to unit load before calculating the effect of the time history. It cannot automatically deduce this value from the database because it is sometimes not possible to measure the load in reality, so some secondary loading effect is used, for example displacement.

A8: Use the **Graphical display** option on the main menu.

A9: No, only E and UTS are mandatory. Partial data sets for strain, stress and LEFM are not allowed, but an entire section can be omitted.

A10: Use **F2** (or **PF2** on Tektronix terminals) to return to previous screens. For field editing, **Tab** clears to the end of the field, or just type over existing data. Insert mode can be entered by typing **#IN**, overwrite mode by typing **#OV**.

A11: Use the Time History Database Manager, PTIME.

A12: This is the node at which the stress was measured to generate the S-N data. The stress is usually not at the place where fatigue cracks will appear but at a remote or nominal area.

A13: We can choose any nodes on hyperpatches 1, 2 and 3. Ignore hyperpatch 4 because it contains stresses which are not of interest. Note that the reference node is automatically included. For the purposes of later discussion, select all nodes on all faces of HP's 1T3.

A14: The time history will have a maximum of 3000 lbs. and a minimum of 7000 lbs. We have no information about any other data values so we can assume that there are no peaks/valleys between these points be-cause frequency is not important. We can assume that only these 2 points are required. The 1/2 hour interval can be modeled using the “fatigue equivalent units” (see later) so we do not need to invent a time interval of 30 minutes.

A15: There are many ways to create this time history. All would give the same result. The simplest way is to use the **Wave creation** option to create a sine or triangle wave with 1 cycle. The mean would be 2000 and the amplitude 5000. There are two ways to do it via the **Graphical create** option. Either by using the **Append** menu with the **Ramp** option, or **Append** with **Waves**. And of course there is the **X-Y point** insertion.

A16: This is a term relating to the real value of 1 repeat of the time history. In our case, we can use 30 minutes, 1/2 hours, 1/48 days etc. The answer will be the same, of course, but we can choose the best parameter for our product.

A17: By choosing the **Plot a time history** option from the main PTIME menu. If the create option was used to generate the history, then the data would be seen at creation time.

A18: Files created by P3⁄FATIGUE are:

bracket.fin
bracket.fnf
bracket.fes
bracket.fpp
bracket.fef
bracket.msg

The .fin file contains setup information, the .fnf file contains DFEG information, the .fes file contains information from the .fin, .fnf and FE results files, the .fpp file contains time series and other information for each node, the .fef file contains results, and the .msg file contains a trace of the batch job for use in diagnosing errors in the batch process.

A19: The result is that the life of weld is approx. 7500 hours. Note that if all nodes has been analyzed, some of the most damaged node would have been at the load application point (i.e. at node 202) which would not have been true. These nodes have the highest stresses so the P3⁄FATIGUE damage allocation algorithms pick up this node as the one with which to associate the highest damage. For this reason, the color contour plot is a waste of time and misleading. The S-N dataset is only valid for the weld area.

A20: Red indicates a value below the design life, Yellow a value less than 3 times above design life and Green a value more than 3 times design life.

A21: This plot is not meaningful for a welded component. We know that the failure is at the weld, so it is only the shortest life which is of interest to us.

A22: The node used is irrelevant because all nodes use the reference node stresses. Lives at other nodes are computed by a ratio of stress relative to the reference node. This is how a contour plot can be created.

A23: Yes, it better.

A24: The back factor is approximately 0.5, which means the loads must be halved in order to meet the required life. The number of hours in 7 years is 61320, excluding leap years.

A25: The failure rate after 7 years would be greater than 99%.

A26: Very little - no correction gives a life of approx. 7800 hours.

A27: The new graph lies above the original. This means that the life will be longer with the modified weld. Remember comparisons need to be made at constant stress; i.e., horizontally.

A28: By reading the first curve, you can find the stress range corresponding to 7800 cycles. If you then read the second curve at the same stress, you can see that the new life will be approximately 10 times the old life. This doesn't take account of mean stress or design criterion.

A29: The new data gives a life of approx. 115800 hours, or 13.2 years.

A30: The load factor is approximately 1.19, which means the product would survive sustained 19% overload of the major cycle. The failure rate at 7 years would be less than 0.1%.