Engineering for Fatigue with FEA

CAE Software Processes

By: Gabriel F. Dambaugh, P.E., President, FEA Services LLC

Introduction

Since its inception in 2000, FEA Services LLC has developed and refined its core competency around real world fatigue and fracture analysis and mitigation. FEA Services continues to bring this expertise and experience to clients throughout several industries.

Over the past two decades we have seen significant advancements in the CAE technologies which are used as tools to more closely predict failure modes that a mechanical structure can succumb to during its in-service life cycle. A common failure mode of interest is fatigue.

These CAE technologies, primarily based on Finite Element Analysis (FEA) methods, have provided more opportunities for design Engineers and stress analysts to better predict and prevent in-service metal fatigue occurrences. When used properly, FEA and fatigue analysis at the design level serves to reduce a significant portion of costly physical prototype build and test iterations and reduces the likelihood of field failures.

This paper addresses specific aspects of some of the primary hands-on processes involving CAE software to be considered in assuring robust utilization of the technologies involved in a design level fatigue analysis and fatigue prevention effort:

- FEA Requirements
    - Directional accuracy
    - Nonlinear effects
    - Mesh refinement/fidelity
    - Sub-modeling
- Fatigue Theories
    - Stress life
    - Strain life
    - Fracture mechanics
- Fatigue Analysis Software
    - Hand calculations
    - Critical plane
    - Modern fatigue analysis software

Another aspect of the fatigue study process not addressed in detail here is the need to correlate analysis to testing and in-field service conditions. It is also vital to understand how the statistical nature of metal fatigue affects the real world results. *All of these aspects* of fatigue must be considered when implementing Engineering processes for fatigue analysis with FEA.

FEA Requirements

The first key to a successful FEA process related to fatigue is to develop Finite Element (FE) modeling techniques that are reliable. Here, "reliable" does not mean "perfect accuracy" regarding stress, strain and fatigue output vs. real testing. But rather, *a reliable FEA process means that the FEA results can be counted on to provide a correct comparison between one design to another, and then another, and so-on.*

To achieve this reliability, the FEA model does however depend on robust directional accuracy, which basically means that enough of the real world physics is taken into account at the external and internal boundaries of the model such that as the load changes or the design changes, the direction and relative magnitude of stress increase or decrease is captured by the models.

Nonlinear effects of the application loads, boundaries, materials and geometry are often overlooked with such assumptions such as, "We only need A to B results, so a simple linear analysis is enough." But the truth for most structures in the real world is that nonlinear effects will act to redirect the load path itself from "design A" to "design B" due to the design changes. This then alters the relative load vs. stress result in the critical areas of interest where fatigue is of concern. These relative alterations due to the nonlinear physics of the structure or environment are ignored when using linear FEA models. *In order to achieve directional accuracy to attain the required reliability of the critical fatigue results, and thus the all-important design comparisons, a solid understanding of nonlinear effects using nonlinear FEA is recommended.*

Regardless of the FEA system used, a critical technique that the FE Engineer must consider for fatigue analysis is mesh refinement. Simply put, when building a robust FE model intended for fatigue analysis, the mesh itself must be refined with greater fidelity than is normally required for other FEA purposes such as basic stress margins or deflection. *The required mesh fidelity is the fidelity at which the stress and strain results in the area of interest are insensitive to small mesh changes.*

FE modeling experience will help guide the Engineer to a robust mesh refinement, yet it is also possible to test the sensitivity of the mesh on any given model through iteration as follows:

Starting with the initial FE mesh, increase the fidelity of the mesh in the area of interest. Typically, the stress result will change with increased fidelity. Therefore, one should iterate with another mesh design of even greater fidelity. When the stress result no longer changes due to an increase in mesh fidelity, a robust mesh refinement is reached which is no longer sensitive to changes in the mesh.

Example of Mesh Refinement/Fidelity

In the following example of mesh refinement, we are taking data from one of our Automotive Industry DOE projects where nonlinear contact and mesh refinement proved to be of significant importance for subsequent fatigue analysis for a certain application.

Figure 1 shows a 3D section of one of the components of concern in the simulation, with the peak stress (red) in the fillet being the area of interest. The other 10+ parts in the FE model (FEM) are removed from the Figure due to the proprietary nature of the models.

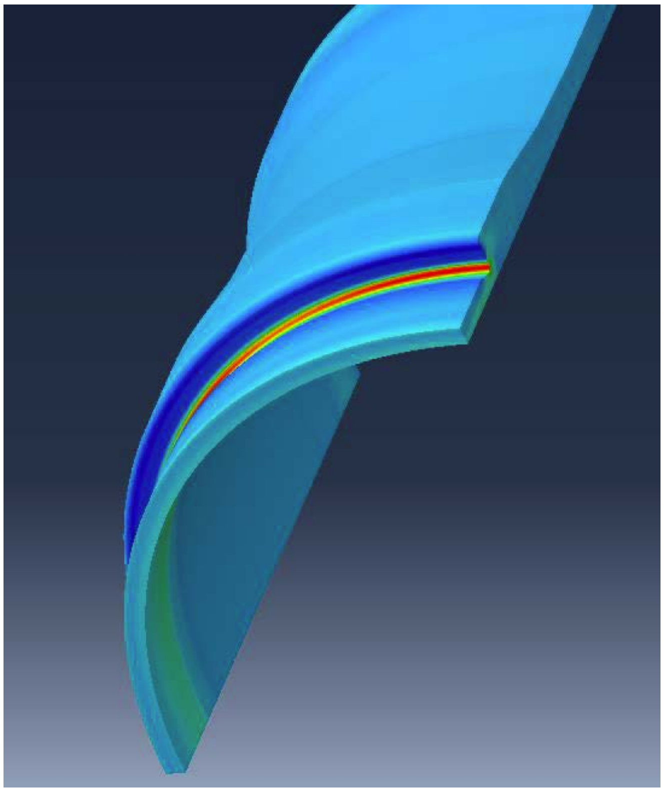


Figure 1. Stress in area of interest.

The three levels of mesh refinement schemes made to a critical fillet in the FEM, rev 0, rev 1 and rev 2 are shown respectively in Figures 2, 3 and 4.

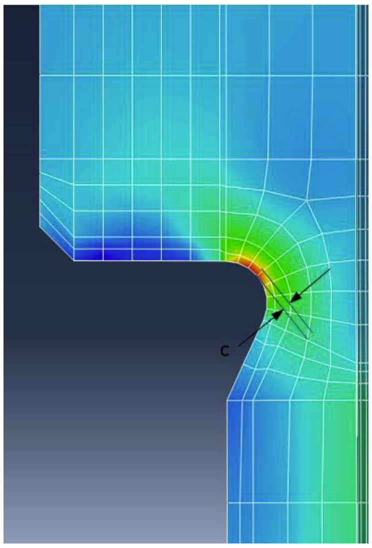


Figure 2. Rev 0 Mesh Lowest Fidelity (course mesh).

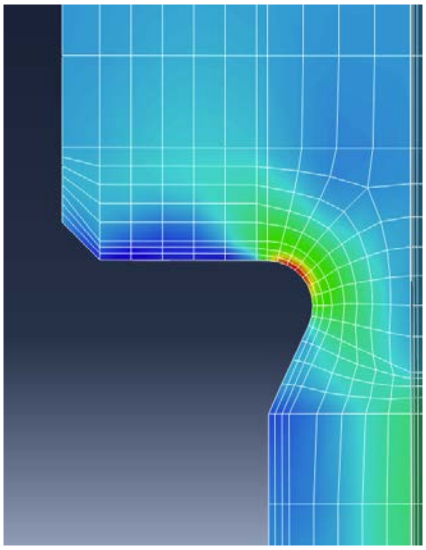


Figure 3. Rev 1, “c” is reduced for Moderate Fidelity.

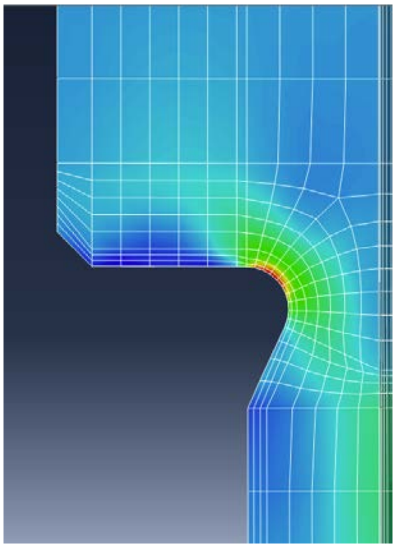


Figure 4. Rev 2, mesh is further refined for Higher Fidelity.

As shown in Table 1, the level of stress observed tends to increase as the mesh density (fidelity) increases, but only up to a certain point. The main result to note is not the magnitude of stress, but rather we find that as the mesh fidelity changed, the apparent “best design” by way of comparison changed as well.

Specifically, with the original (rev 0) mesh, design “A” seems to be the better of the design choices, but as the mesh is refined through to rev 2, it becomes clear that design “B” would actually be expected to survive more cycles under fatigue loading, and design “A” now looks to be the worst out of all four designs.

Design Revision	Tensile Stress Output in Fillet (pressure unit proprietary)	
	Mesh rev 0 (course)	Mesh rev 2 (refined)
A	730	944
B	740	812
C	800	924
D	800	890

Table 1. Stress output comparison with mesh refinement.

The take-away from this example is that a higher degree of mesh fidelity is required of the FEM in the areas of interest when considering fatigue analysis, even when the analysis is done only to compare one design to another.

In larger FE models, or in models with a high level of complexity, the added run-times related to a mesh refinement study can be a major issue for the Engineering team regarding budget or project timing. One way to economically achieve a reliable stress (or strain) result for fatigue analysis is with the use of sub-models.

A sub-model is in itself another FEM which is used to make a more refined analysis around a smaller area of an otherwise coarsely meshed larger FEM. The sub-model is essentially a smaller portion of the original (global) FEM. Like cutting a smaller piece out of a larger part, the outer boundaries of the sub-model geometrically lay within the spatial coordinates of the global model. The driving forces or displacements applied to the sub-model boundaries are taken directly from the results of the original global model, along with any remaining internal forces within the sub-modeling space.

Generally, the main purpose of making the sub-model analysis is to consider the same loading condition as was made for the global model initially, but with more attention to the smaller areas of interest (i.e. with high fidelity meshing suitable for fatigue analysis). The FEA program, Abaqus, from Dassault Systèmes Simulia Corp., has unique sub-modeling advantages over other programs which is highlighted in the following example.

Example of Sub-Modeling

In the following example of sub-modeling, we will consider one of our heat exchanger forensic analyses. The Abaqus global model/sub-model process was used and actually determined the same worst-case area for fatigue as compared to real field data. Once this process was established, the manufacturer was then able to check which design modifications would make the most sense to mitigate future concerns with their product's fatigue life.

Figure 5 shows the temperature gradient as applied to the global FEM of the heat exchanger. The discrete mesh grid is not shown in the Figure for clarity. The thermal gradient is determined either within Abaqus using heat transfer or CFD, or from the customer's own heat transfer program.

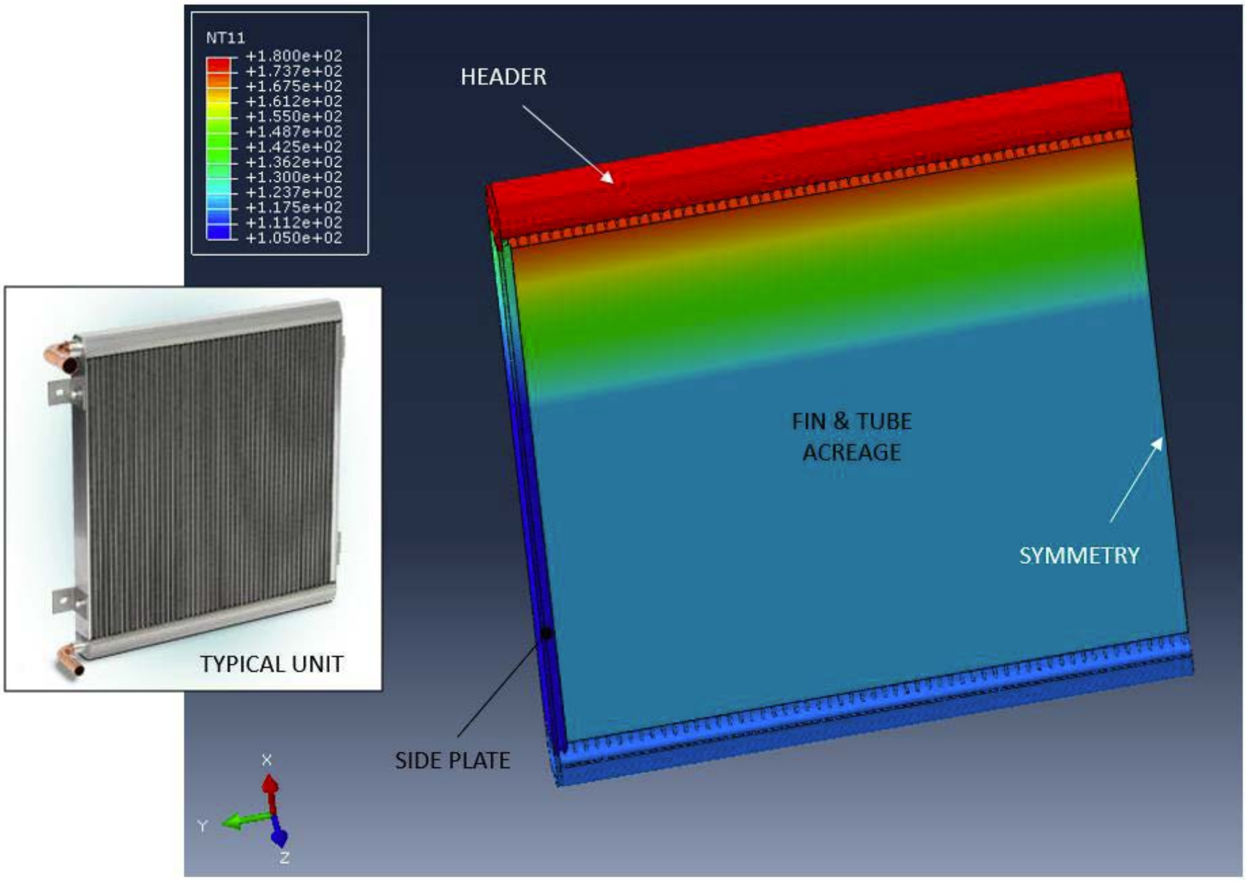


Figure 5. Heat exchanger global FEM and thermal map.

Since non-uniform thermal gradients produce mechanical strain energy, the system predictably deforms under the stress. Figure 6 shows the global model result where the stress tends to concentrate in the upper-left corner and on the outermost tube. However, upon review of the mesh density in this area, (Figure 7) we determine the stress results are likely not good enough for a reliable fatigue analysis due to the relatively course mesh density of the tubes.

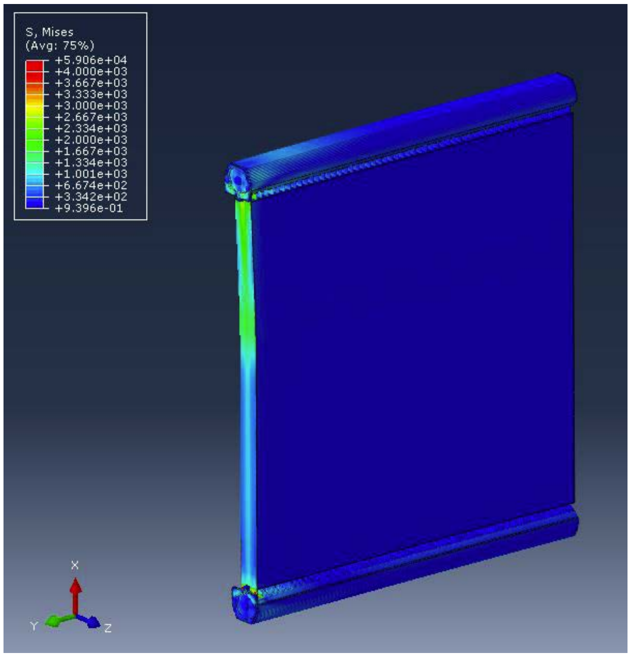


Figure 6. Stress plot of global model due to thermals.

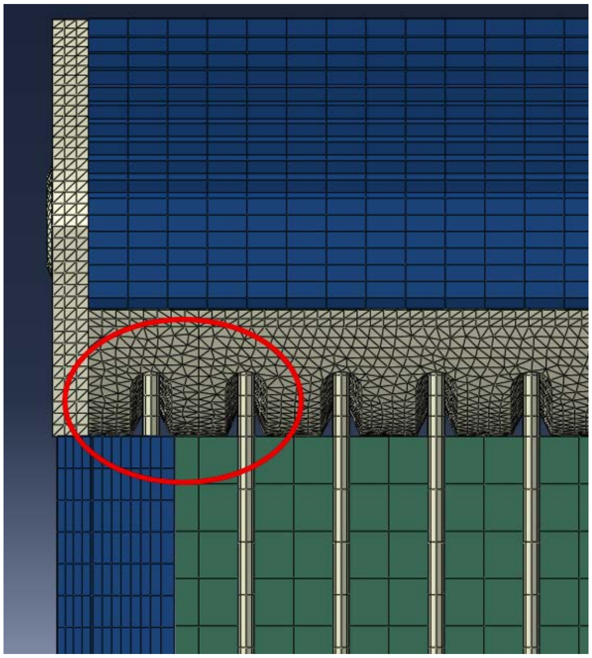


Figure 7. Local mesh density is too course for reliable fatigue analysis and comparisons.

Key point: Economics. This is a relatively large FEM. The global model is significant in size with approximately 10 million DOF such that an average run time is on the order of 48 hours given a typical FEA software and hardware configuration. If the global FEM is recreated with a highly refined mesh, even if the refinement is contained within the general area of interest (e.g. the corner area), there would potentially be an extreme run time expected. Hence, as shown in Figure 8, a much more refined sub-model is developed.

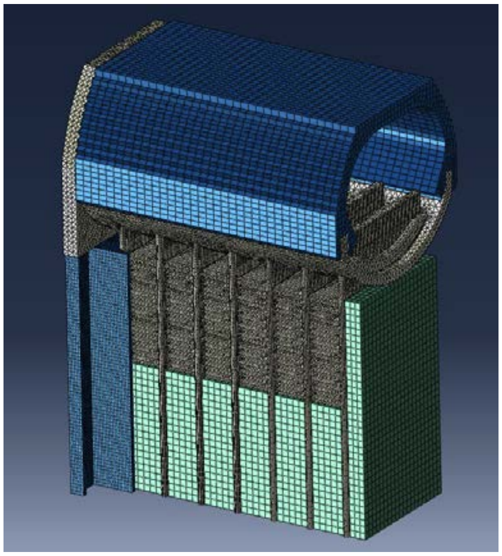
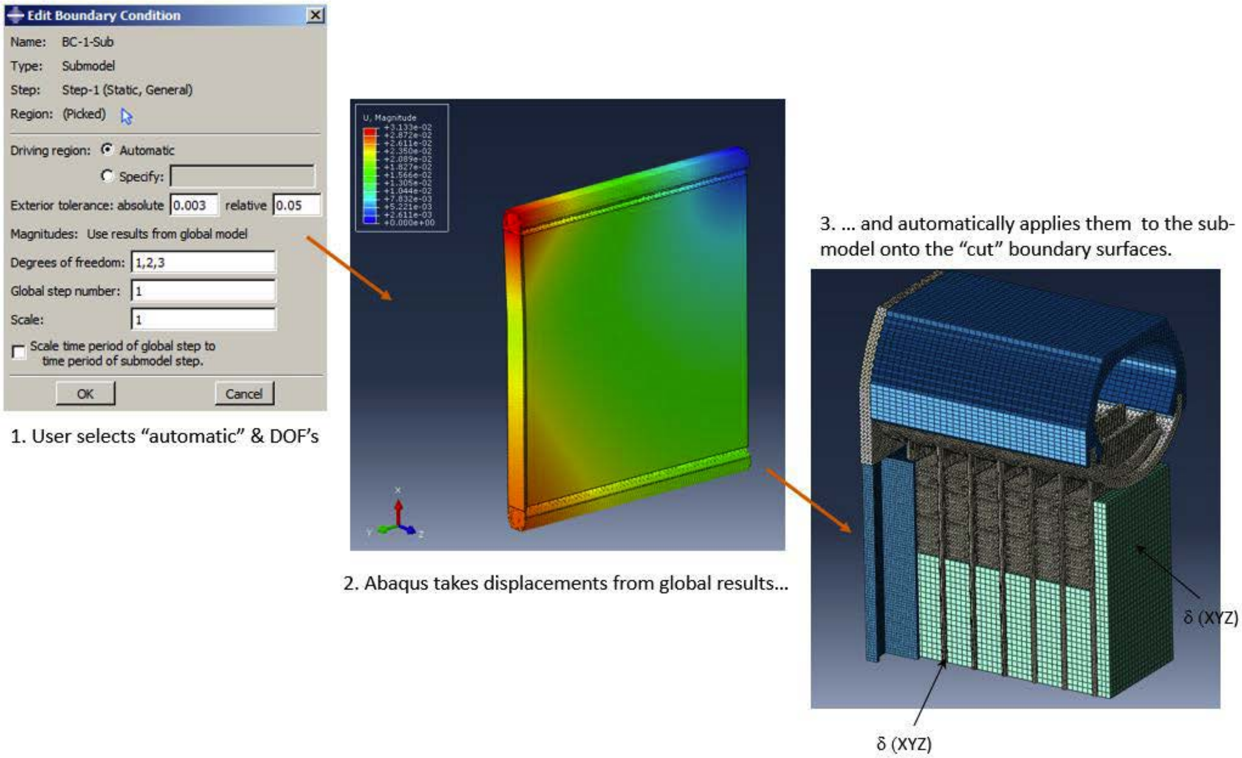


Figure 8. Sub-model with significant mesh refinement.

Creating the sub-model in Abaqus is generally a quick process, since the geometry is already in place and in the same XYZ space as what is needed. We simply “cut away” the geometry that is not necessary for the fatigue analysis. Once the sub-model is meshed, *the process to finish setting up the analysis is done within minutes when using Abaqus because the transfer of forces or displacements is performed <u>fully automatically</u> by the Abaqus software.*

Figure 9 shows the basic process of how displacement results from the previously run global model are automatically transferred into the new sub-model with just a few simple GUI based commands. The user is <u>not</u> required to pre-designate node sets or geometric areas at any time. In fact, the global model and sub-model meshes do not need to match or even contain the same element types.



Edit Boundary Condition

Name: BC-1-Sub

Type: Submodel

Step: Step-1 (Static, General)

Region: (Picked)

Driving region: ● Automatic ○ Specify:

Exterior tolerance: absolute 0.003 relative 0.05

Magnitudes: Use results from global model

Degrees of freedom: 1,2,3

Global step number: 1

Scale: 1

☐ Scale time period of global step to time period of submodel step.

1. User selects “automatic” & DOF’s

2. Abaqus takes displacements from global results…

3. … and automatically applies them to the sub-model onto the “cut” boundary surfaces.

Figure 9. Process to automatically transfer global results (δ) to the sub-model.

Abaqus also offers perhaps the only automatic sub-modeling routine which also has full nonlinear capability, such as metal plasticity. It is important in some applications to capture the inelastic strains for use with a strain life assessment of fatigue. Figure 10 shows the sub-model stress result of interest which indicates a high stress at the location shown. This location precisely matched the fatigue initiation point due to thermal cycling as found in real heat exchangers from the field per Figure 11.

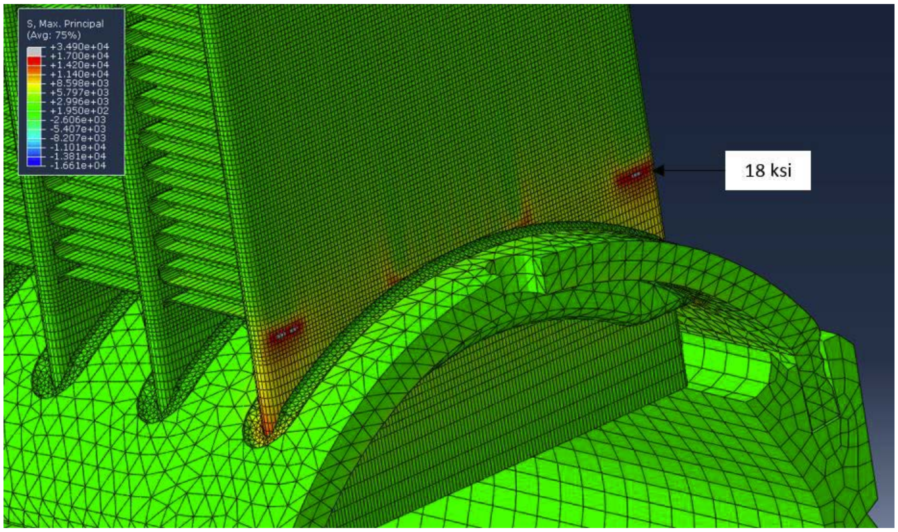


Figure 10. Sub-model stress result of interest with enough fidelity for fatigue analysis (side plate removed from view for clarity).

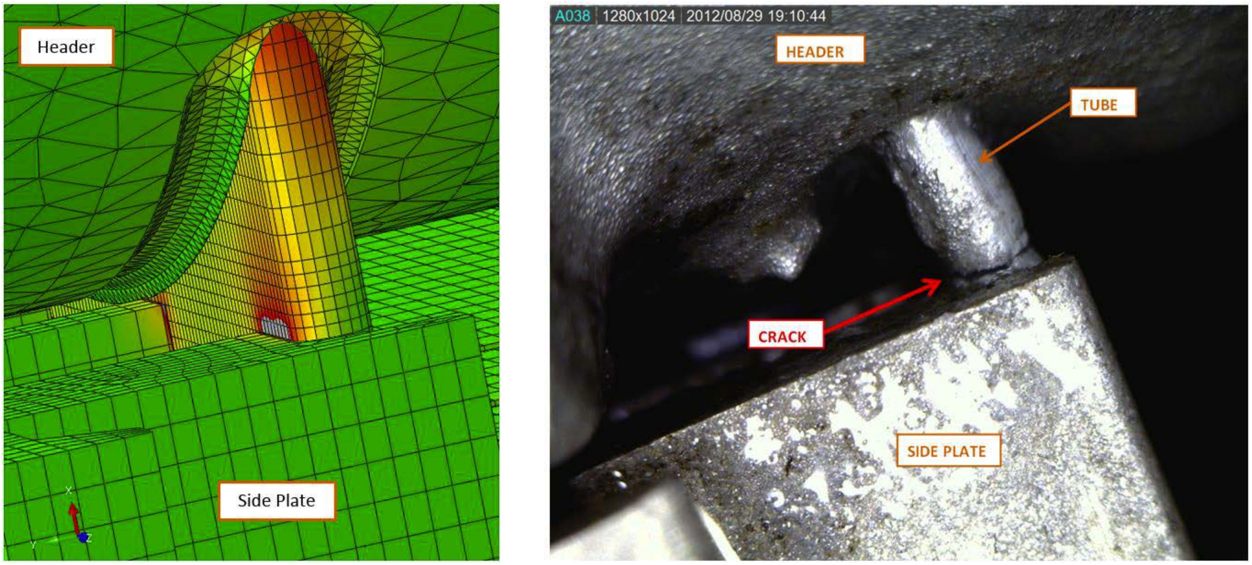


Figure 11. Side-by-side view of stress results compared to field result.
Gray color on right indicates elements/volume stressed beyond yield.

The end result was that the customer made design iterations to mitigate the fatigue issue such that a significantly reduced stress (approximately 50%) was found from the respective FEA's. Two years after this FEA work was completed, we understand that the field service life issues have in fact been mitigated. Using the sub-modelling technique within Abaqus for a refined mesh fidelity, sufficient for a reliable stress result, was paramount in accomplishing this fatigue analysis task.

Fatigue Theories

Once the requirements have been met for reliable stress results from the FEA, the next step in a successful FEA process related to fatigue is to translate the stress data into a proper fatigue assessment. Here, “proper” does not mean perfect accuracy regarding a resulting cycle count made from a single fatigue calculation. “Proper” rather means that an appropriate fatigue theory and calculation method is used such that *consistent correlations* (e.g. calculation vs. real testing) are eventually made in the test arena and that the comparison between designs on the relative expected fatigue life is proven.

An important point is that for many fatigue assessments, the fatigue analysis correlations are *eventually* made with real testing, since it is realized that an iterative process for a given application is required. This is due to the vast complexity of real fatigue variables. A fatigue theory is just that: A theory which is primarily based on observation and empirical curve fitting.

The three main top-level fatigue theories generally considered for metals are:

1. Stress life
2. Strain life
3. Fracture mechanics

Stress Life

Stress life, a long time standard, is considered most useful when “infinite” life or otherwise high cycle fatigue (>100 K cycles) is to be obtained. It is perhaps the easiest fatigue theory to understand and the test data is direct with many material test libraries in existence. From an analysis point of view, this is a good choice when the stress remains in the elastic region. Stress life data is often represented by S-N data plots as shown in Figure 12.

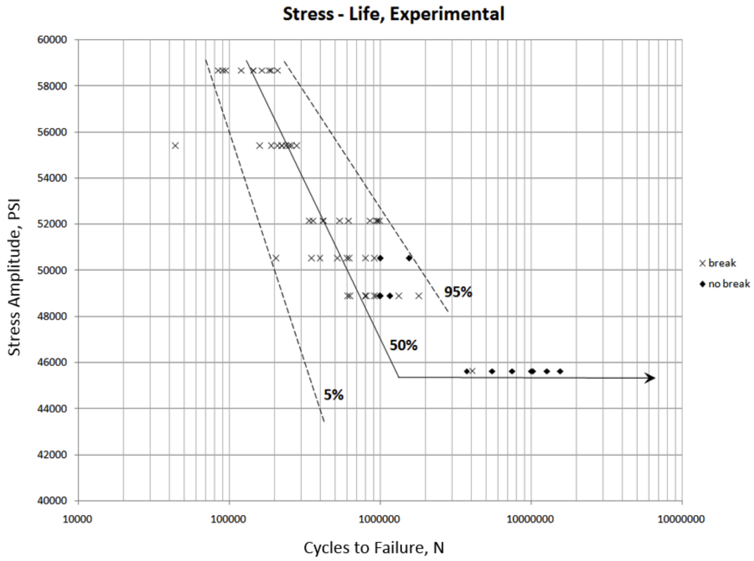


Figure 12. Typical S-N data with statistical reliability bands.

Strain Life

Strain life is a focus of more modern fatigue theories and is based on elastic *and* plastic responses. Hence, it is a good choice when metal plasticity in the area of interest is expected to be greater than yield. Test data for a specific alloy can be obtained economically. Modern fatigue analysis software also supports the concept of the critical plane using strain life. The common Smith-Watson-Topper theory is given below in Equation 1.

$$\sigma_{\max}\ \varepsilon_a = \frac{(\sigma_f')^2(2N_f)^{2b}}{E} + \sigma_f'\,\varepsilon_f'\,(2N_f)^{b+c}$$

Equation 1. Smith-Watson-Topper Strain Life Theory

σ_{max} *and* ε_a: *applied stress and strain (from FEA)*
E= *elastic modulus*
$\sigma_{f'}$ *and* $\varepsilon_{f'}$: *material alloy strength parameters from testing*
b *and* c: *strain life exponents from testing*
N_f= *cycles to failure initiation*

As with any of the three main fatigue theory topics, there is a great variety of more detailed theories developed over the years made by researchers who continue to dive deeper into the physics and real world results of how the many variables affect the outcome of fatigue. For example with strain life, a few of the popular theories to consider are:

- Coffin-Manson relationship
- Masing's hypothesis
- Smith-Watson-Topper (SWT)
- Morrow's mean stress correction
- Brown-Miller
- Fatemi-Socie
- McDiarmid's criterion

Fracture Mechanics

Fracture mechanics is primarily the study of cyclic crack growth once a crack has already been produced on the material surface as opposed to stress life or strain life theories which are based on the cycle count required to *initiate* a fatigue crack. Linear elastic fracture mechanics (LEFM) is used extensively in the Aerospace industry and has strong potential to benefit other applications.

Figure 13 shows a crack growth rate (da/dN) curve which is empirically obtained material data from coupon testing that is required for LEFM techniques.

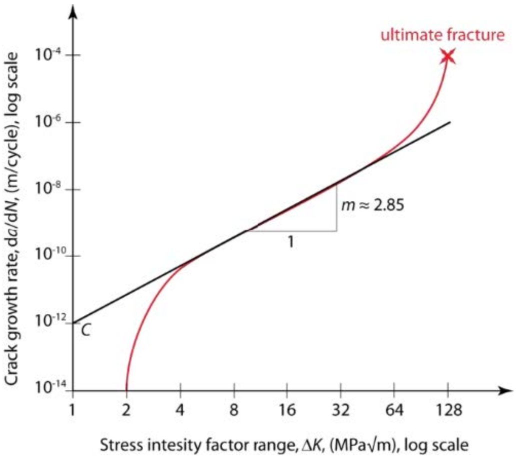


Figure 13. Typical da/dN plot.

Although we will not dive into the details of working with LEFM for fatigue approximations, it is important to note that the stress intensity (ΔK) as referenced in the Figure above can be determined by a comprehensive FEA package, such as Abaqus, for any given complex part. Furthermore, the emerging XFEM technology within Abaqus actually simulates crack growth and growth direction in a more automated way within the FEA run itself.

Fatigue Analysis Software

Following the FEA development and choice of which fatigue theory to consider for the fatigue assessments, the next primary decision to make is whether to use manual "by hand" calculations, or a more comprehensive modern CAE based fatigue software package. Whether done manually, or with fatigue analysis software, the goal of the "Fatigue Analysis Code" (see Figure 14) is to provide an analytic result of the estimated number of load cycles for fatigue failure initiation.

Hand Calculations

Hand calculations using basic textbook level methodology can be a good economical approach when the cyclic loading is simple. By "simple" we mean:

- There is a single load which cycles along only one line of action; or-
- Multiple loads are applied which are always synchronous, with no phase changes throughout the spectrum.

More complex hand calculations methods, such as the FKM guideline (VDMA Verlag) can also be employed to consider some of the typical fatigue related variables such as residual stress.

In many real world applications, the loading spectrum becomes more complex, whereby loads are in multiple directions and are not in-sync. (i.e. the phase relationship is not constant). In these cases, it is agreed among modern fatigue researchers that there is a critical plane at the point of fatigue initiation, which must be considered when making a fatigue assessment.

Critical Plane

Critical plane methods resolve the stresses/strains on all of the planes at a point on the part to determine the worst-case plane at which fatigue cycling is the most damaging. When considering the FEA output, the typical method of using (only) the worst-case tensile (Max Principal) stress at a point on the part being analyzed will at times result in an erroneous fatigue result when loads/stresses are out of phase. *By using a critical plane method for spectrums which have varying phase relationships, the mistake of taking the peak Max Principal stress at a single snap-shot in time on the part at the point of fatigue concern is avoided.*

Modern Fatigue Analysis Software

Modern fatigue analysis software, such as fe-safe™, is necessary to use because the complexity of the critical plane method, combined with the need to calculate damage over many points from an FEA model, makes this task impossible to do manually by hand. The detailed advantages of fe-safe™ are beyond the scope of this paper, yet in our experience it is very simple to set up even with complex spectrums, runs very quickly with FEA output, and has provided us excellent comparisons to real testing.

A simplified flow chart of recommended processes for fatigue analysis, sans the Product Data Management (PDM) aspects of managing the data, is shown below in Figure 14.

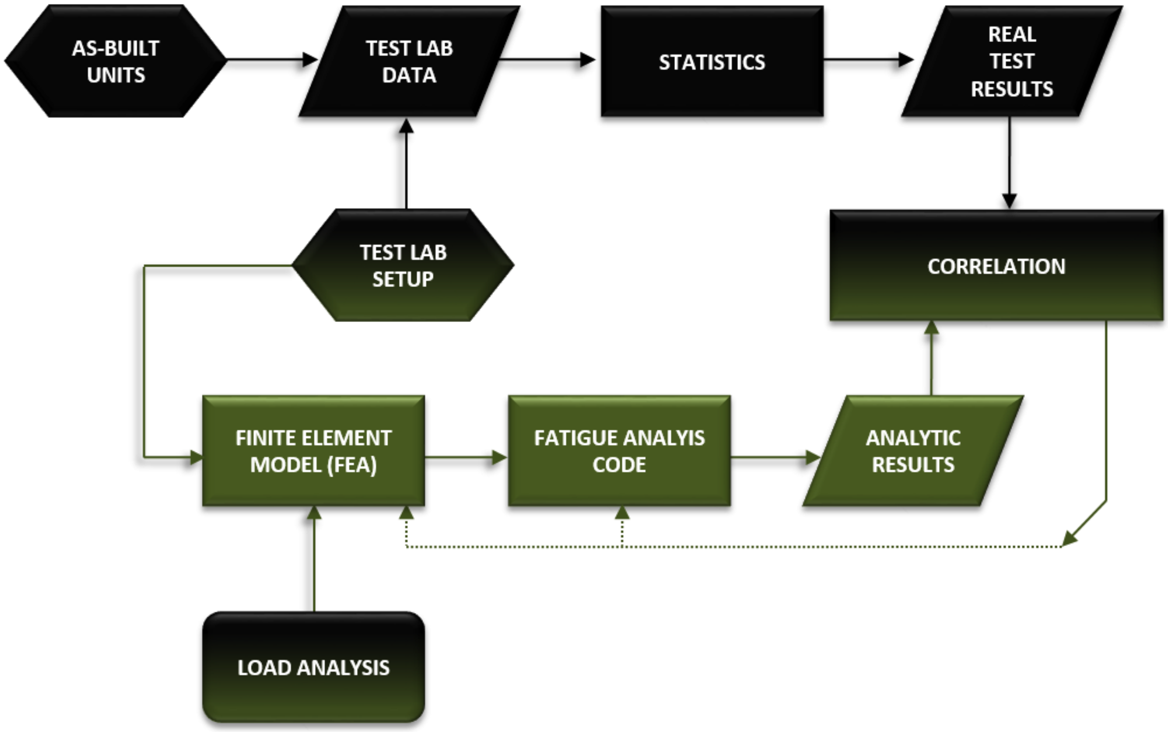


Figure 14. Simple flow chart: Engineering for fatigue with FEA.

Summary

Top level insights into some of the important aspects of Engineering for fatigue with FEA to reduce fatigue occurrences in the field are provided herein. Given the constant market pressure for reduced product weight and cost, more accurate Engineering tools are becoming a business requirement.

While the focus of this paper is the use of the CAE software tools and processes such as with FEA or fatigue analysis software, an understanding of fatigue correlation and statistics are other concepts which must also be mastered for the best real world results.

Regarding Engineering for fatigue with FEA, some main points are:

- A proper load path in the FEA may require nonlinear modeling in order to assure good directional accuracy.
- The FE mesh must have sufficient refinement for robust reliability. In Abaqus, sub-modeling provides a sophisticated yet simple means for mesh refinement, even with initially complex and/or nonlinear models.
- The choice of an appropriate fatigue theory is important.
- With complex loading spectrums, understanding critical plane analysis and using modern fatigue analysis software is often necessary to more accurately predict life cycles.
- Initial fatigue results will often require comparison and correlation.

For clarifications or questions regarding the content of this paper, please do not hesitate to contact us via email at info@feaservices.net. Additionally, a more detailed white paper specifically regarding high strength steel fatigue can be downloaded from our site: www.feaservices.net/downloads.htm