Model set up for MSC Nastran Embedded Fatigue (NEF)

Introduction

Structures and components are designed to endure in extreme loads. However, when subjected to cyclic loadings they can fail from loads much lower than the extreme ones. As in reality structures and components are very unlikely to be subjected only to static loads, fatigue analyses are integral to the creation of durable products. Traditionally, to undertake a fatigue analysis, two separate solvers would be needed, one for stress analysis and one for the fatigue analysis. This approach has a significant impact on performance and complexity.

With the Embedded Fatigue analysis module of MSC Nastran (NEF), stress and fatigue calculations can occur in one simultaneous operation offering considerable benefits in performance and simplicity. Facilitating further this process, BETA CAE Systems SA, ANSA pre-processor offers the functionality to setup fatigue analyses for NEF in one database, and in a single environment.

The Process

This is accomplished through a robust yet fast and user friendly process the steps of which are listed below:

1. Stress analysis set up.
2. Definition of material properties.
3. Definition of fatigue properties.
4. Definition of cyclic fatigue loadings.
5. Definition of parameters.

Example case

The shock tower of a vehicle body is subjected to two forces and a moment as shown in the figure.
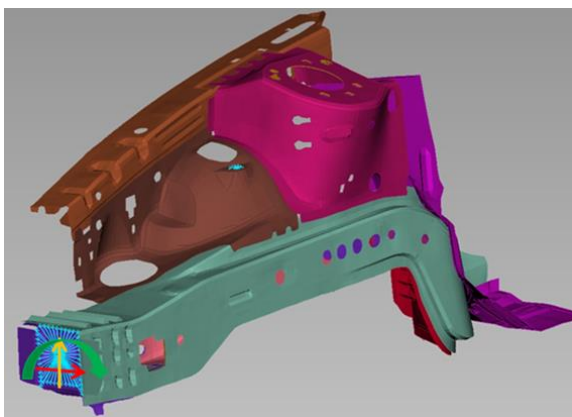

Figure 1: The shock tower of a vehicle body is subjected to two forces and a moment.

First, the model was set up for a static linear solution and afterwards cyclic variations are assigned to each load in order to perform the fatigue analysis. Physical and material properties are assigned as well. The proper parameters are defined for the solution and the preparation process of the example models is completed by exporting into *.nas file.

Stress analysis set up

Before performing the fatigue analysis, the model has to be set up for a static analysis. As a first step, the boundary conditions are defined and subsequently the loads are applied.
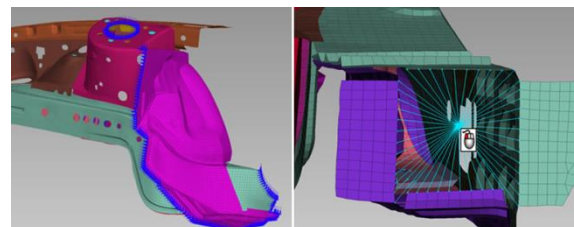

Figure 2 Definition of boundary conditions

At this point, the header for the definition of the SOL 101 solution is set up. In addition to SOL 101, the SOL 103, SOL 112 and SOL 200 are also fully supported. The process includes the specification of the solution type, the boundary conditions to be used in this type of analysis, as well as the results

(output requests section) that will be exported into the Nastran file (.nas). Each of the three applied loads is defined in different subcase. After the definition of fatigue keywords the header is extended.

Definition of material properties

Subsequently, the values for Ultimate Tensile Stress (UTS) and for Yield Stress (YS) are defined and the Strain-Life approach that will be followed is selected.

Subsequently the following parameters are defined for the Material; Strength Coefficient [Sf], Strength Exponent [b], Ductility Exponent [c], Ductility Coefficient [Ef], Cyclic Strain Hardening Exp [n], cyclic Strength Coefficient [K'], Cut-off in Reversals [Nc], etc.
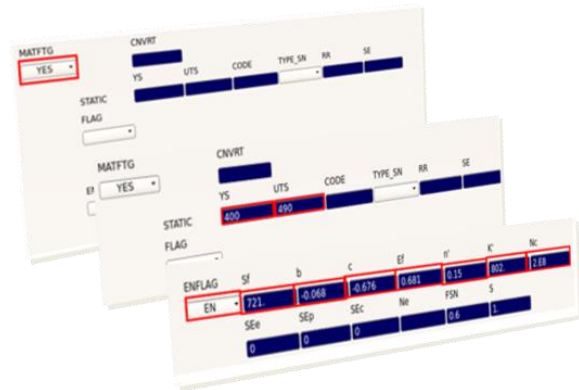

Figure 3: Defining the material properties in ANSA

Definition of fatigue properties

The next step is the creation of the fatigue properties and the definition of the elements and their associated fatigue properties that will be considered for the analysis.
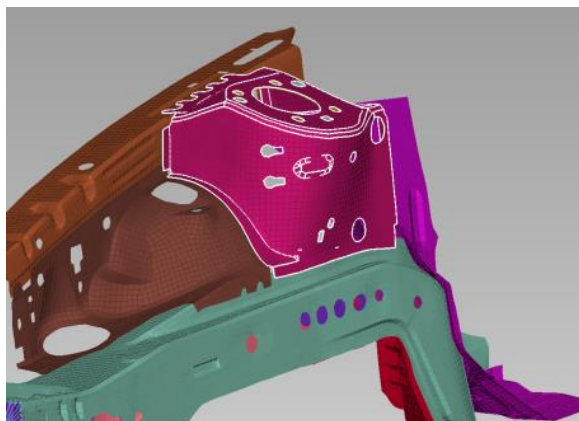

Figure 4: Definition of fatigue properties

Definition of cyclic fatigue loadings

Fatigue analysis should have a definition of the cyclic nature of the loading. This is accomplished using the bulk data entries: FTGLOAD, TABLFTG, FTGEVNT and FTGSEQ.
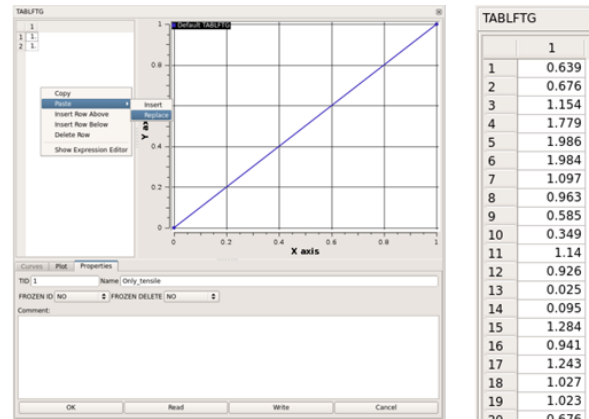

Figure 5: Definition of loads

In this example, loading time variations are applied to each subcase using the linear static FE results and the loading conditions are grouped into events in order to act simultaneously.
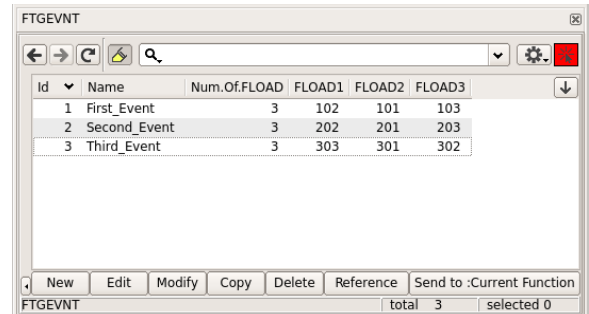

Figure 6: Defined load events

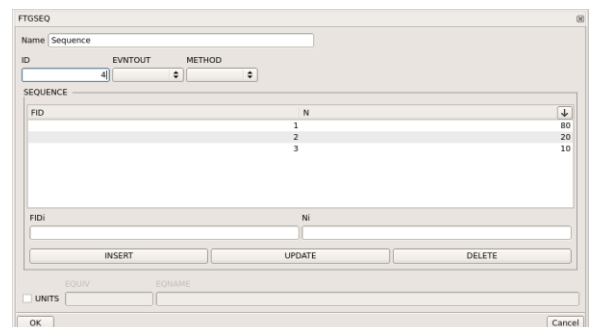Finally, a repeat factor is defined in order to express the entire event.

Figure 7: Defining events' repeat factor

Definition of fatigue parameters

In order to finish the setup, the fatigue parameters must be defined. The main use of this entry is to specify the method of the fatigue analysis (Stress-Life or Strain-Life), and other parameters as well. The Strain-Life method (e-N) was followed in this case.
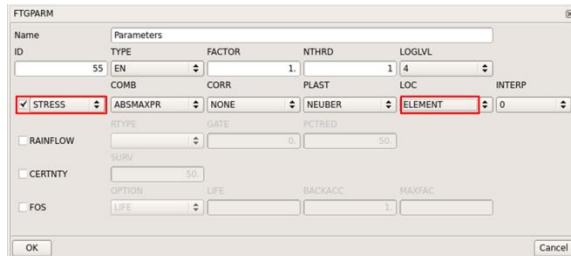

Figure 8: Defining the fatigue parameters

Lastly, the header is extended in order to complete the model and export it.

Review of the results

After running the analysis in Nastran, the user can evaluate the results in µETA post processor. Many fatigue results types are available, such as, Life, Damage, maximum/minimum strain for E-N method, factor of safety etc.

Figure 9: Results review in µETA

Benefits

By employing ANSA for Nastran fatigue analyses, numerous benefits are realized that greatly facilitate the CAE processes for fatigue analyses including:

- Facilitation of the process as it can be performed in a single environment.

- Significant time reduction for large fatigue and durability simulations.

- Capability to perform fatigue analysis only in specific part of the model.

- Smaller file sizes.

- The fatigue process is far more transparent as it offers the opportunity to engineers to include a fatigue calculation with every stress run.

For more information contact
BETA CAE Systems S.A.
Email: ansa@beta-cae.gr