3 FINITE ELEMENT MODELLING

The simulations were done by constructing the finite element models in Abaqus and importing it to fe-safe fatigue analysis software to estimate the fatigue life. It is very important to use the most suitable material models, parameters and boundary conditions to idealize the actual condition. Fatigue analysis is sometimes called the five box trick which includes material, loading, geometry inputs and analysis and results. (Hossain & Ziehl, 2012) The 3 main inputs for the finite element model is given in Abaqus FEA to analyse and get the stress strain histories for stabilized cyclic response. The fatigue material properties are provided to fe-safe software and the stress strain histories are imported to fe-safe software to analyse fatigue.

3.1 Abaqus FEA

3.1.1 Introduction

Abaqus is a finite element software which is capable of modelling dynamic/static behaviour. There are three core software products that can be used to model structural components.

1. Abaqus /CAE- (complete Abaqus graphical user interface) it is a software application that can be used to both the modelling and analysis.(pre-processing and post-processing)
2. Abaqus/Standard – This is a general-purpose finite element *analyser* with implicit integration.
3. Abaqus/Explicit - This is a special-purpose Finite-Element *analyser* that uses explicit integration to solve highly nonlinear systems with many complex contacts under transient loads.

Abaqus/CAE is divided into 11 modules. Each module defines a logical aspect of the modelling process. The modules help building the model from which Abaqus/CAE generates an input file. It is submitted to the Abaqus/Standard or Abaqus/Explicit analysis product. Then the analysis product performs the analysis, and generates an output database. Eventually the visualization module of Abaqus/CAE shows the results of the analysis.

3.1.2 Material properties

Material properties of monotonic tests cannot be used to simulate cyclic loading conditions. Therefore cyclic properties must be obtained and used for the simulations.

When a material is subjected to cyclic vibrations under fixed strain range elasto-plastic strain an initial softening or hardening behaviour will be shown. After a certain number of cycles of initial hardening or softening (figure 3.1) the material stabilizes, producing a stabilized hysteresis loop (loop closing behaviour). (Abaqus, 2013)

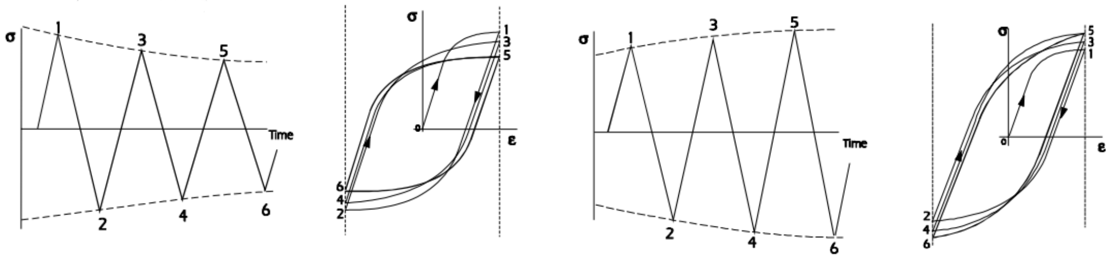

Figure 3.1: cyclic softening (left) and cyclic hardening (right) for a constant amplitude cyclic strain

The stabilized cyclic response is employing much of the fatigue life of the material. Near fracture response again become transient. (STL, 2002)

The material hardening behaviour can be idealized in Abaqus under plastic properties. The 3 main types are isotropic kinematic and combined.

1.) isotropic

The isotropic hardening model is useful for cases involving gross plastic straining or for cases where the straining at each point is essentially in the same direction in strain space throughout the analysis. Therefore this is not suitable for cyclic behaviour. Isotropic hardening describes the change of the elastic range.

2.) Kinematic

The kinematic option uses the linear kinematic model to define a constant rate of cyclic hardening. This is suitable to simulate the inelastic behaviour of material which are subjected to cyclic loading. Kinematic hardening describes the translation of the yield surface in stress space. When temperature dependence is omitted, the evolution law used here is the linear Ziegler hardening law

3.) Combined

The combined feature can be used to define non-linear kinematic hardening and isotropic hardening together. The combination of the isotropic component together with the nonlinear kinematic component can be used to predict shakedown after several cycles. (Cyclic hardening with Plastic shakedown)

. The Bauschinger effect (increase in tensile/compressive yield strength occurs at the expense of compressive/tensile yield strength) and plastic shakedown (plastic deformation ceases after a number of initial cycles and the response goes back to pure elastic with some state of residual stress) can be modeled with both models, but the nonlinear isotropic/kinematic hardening model provides more accurate predictions. (Abaqus, 2013)

It is very important to use proper a kinematic/combined hardening model instead of isotropic model and cyclic material properties instead of monotonic properties to idealize the actual condition.

3.1.3 **Boundary condition and loading**

Abaqus FEA uses steps to identify the changes in loading, boundary conditions, or when specific output requests are required. The analysis method of the step can also be defined. There are 2 procedure types,

1.) General

The nonlinear effects in the model can be included in general analysis step. The starting condition for each general step is the ending condition from the last general step, with the state of the model evolving throughout the history of general analysis steps as it responds to the history of loading.

There can be 3 sources of nonlinearity

I. Material nonlinearity -
   Nonlinear material behaviours such as nonlinear elasticity and plasticity can be added

II. Geometric nonlinearity
   The nodal displacements are considered to be significant so taken in to account for analysis.

III. Boundary nonlinearity
   The nonlinearities in the boundaries such as contact friction, springs etc are considered.

2.) Linear perturbation

The response in a linear analysis step is the linear perturbation response about the base state.

The loading in finite element models can be either force, pressure, temperature, displacement, or a number of other types. There are different kinds of loading in Abaqus each using a different type of time history. For example, RAMP and STEP define how and when the loading is applied during a given step. Otherwise Amplitude can be defined as well.

Idealisation of actual boundary conditions are very important to simulate the actual condition. Mechanical boundary conditions such as connector displacement, displacement /rotation restraints can be defined.

3.2 FE-safe

Fe-safe is a fatigue analysis software which must be used alongside FEA software to estimate where fatigue crack will occur when fatigue failure happens and the factor of safety on working stresses.

Fe-safe has 2 methods to analyse fatigue when the components are subjected to elasto-plastic vibrations

1. Using elastic FEA(elastic block) results with Neuber's correlation

The advantage of this method is that stress histories can be given in the form of loading datasets in fe-safe software itself. But the problem is that the stress redistribution at element nodes are neglected when Neuber's correlation is used. Therefore this method may predict higher number of cycles to failure than the actual condition when large region of the component being analysed is subjected to plastic stresses.

2. Using elasto-plastic FEA results (elastic-plastic block)

This method is suitable for any elasto-plastic stress history since the plasticity is taken in to account in the finite element analysis. But stress history cannot be given changed as desired to multiples as in elastic block method. Elasto-plastic FEA should be performed for every stress history required.

The stabilized cyclic response modelled in Abaqus FEA should be properly picked out of the stress strain datasets to perform the analysis.

Fatigue properties of the material can be defined and other fatigue inducing properties such as surface finish, residual stresses should be carefully chosen. Finally the algorithm to perform fatigue analysis is chosen. "Brown-miller equation gives most realistic fatigue life estimates for ductile metals" (Fe-SAFE, 2002,7-20) Brown miller equation is applied to 3d elements in fe-safe using critical plane method. The most damaging plane is identified by applying the equation to planes at 10º intervals between 0º and 180º from the surface of the component. (Fe-safe, 2002)