DesignLife Theory Guide

About this Document

Copyright Notice

Warranty Notice

While HBM makes the product as reliable as is reasonably possible, HBM does not warrant that the product will function properly under all hardware platforms or software environments. Certain combinations of third party software and/or manufacturers modifications to hardware and software may impact upon the operation of HBM software.

HBM has tested the software and reviewed the documentation but HBM makes no warranty, implicit or explicit, with respect to the product, its quality, performance, merchantability, or fitness for a specific usage. This software and documentation are licensed "as is" and you—the licensee—assume the entire risk as to their quality and performance when you use the product.

Liability Notice

HBM will not be liable for damages arising from the results, direct or indirect, special, incidental, or consequential, of the licensee's usage or misusage of the product, even if advised of the possibility of such damages. In particular, and without prejudice to the foregoing, HBM has no liability for any programs or data stored or used with HBM software, including the costs of recovering such programs or data.

Feedback

We value your feedback. If you have any documentation questions, comments, or concerns, please e-mail us at: techdocs@hbmncode.com

Customer Support

For issues involving the installation or use of your software, please e-mail us at: ncodesupport@hbmprenscia.com

or use the nCode website, www.ncode.com, to find support information for your country.

Contents

Figures

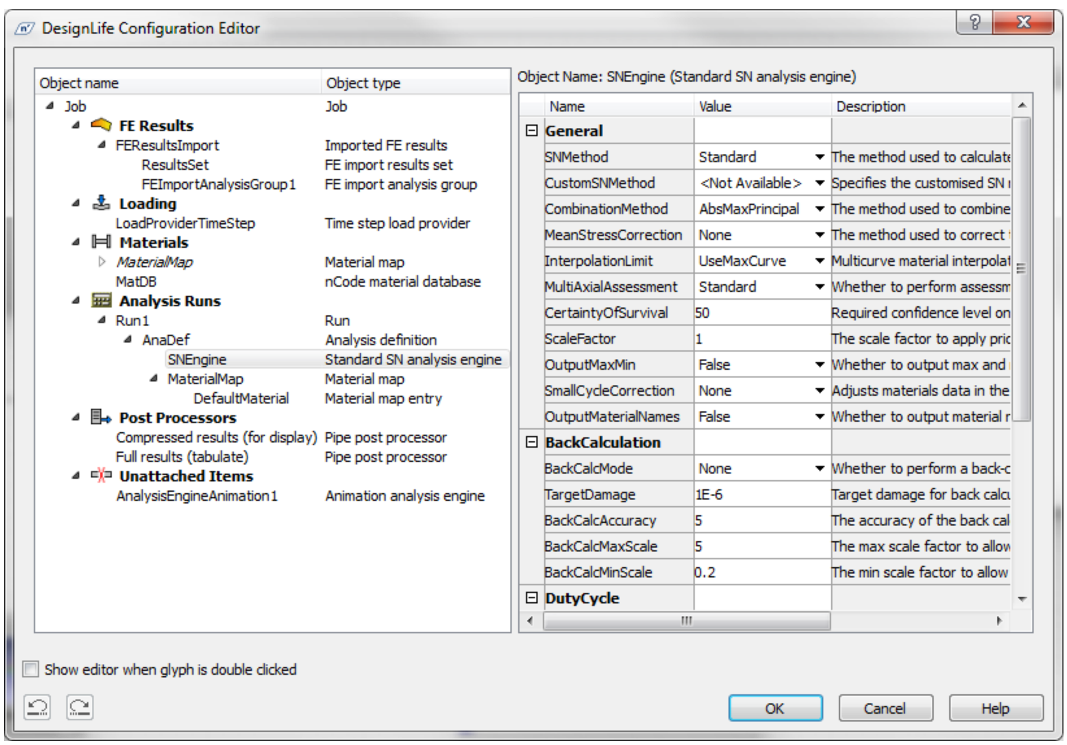
Tables

1 Introduction

nCodeDT provides the fatigue analysis “engine” for DesignLife and other, third-party, fatigue analysis products. This document describes the methods nCodeDT uses to make FE-based fatigue calculations. It concentrates on the way those FE-based fatigue analysis methods are implemented in nCodeDT, and provides details of any methods that are specific to nCodeDT.

The way that a fatigue analysis is carried out by nCodeDT and the methods used are controlled by the fatigue configuration (`.fej`) file, which defines the hierarchy and properties of the objects that comprise the fatigue job. This configuration and the object properties can be viewed and edited using the **DesignLife Configuration Editor**, which can be accessed from within DesignLife by selecting the **Advanced Edit** option on any of the CAE fatigue analysis glyphs. A typical example is illustrated below in Figure 1-1.

Fig. 1-1 Typical nCodeDT configuration



For basic information about all these objects and their usage, refer to the DesignLife User Guide. This DesignLife Theory Guide concentrates on the theory behind the operation of the different objects, and the meaning of the various properties. Its sections reflect the categorization of objects within the configuration:

- **FE Results** describes the handling of different types of FE results and the definition of analysis groups.
- **Loading** describes the different methods that nCodeDT uses to build the dynamic stress and/or strain histories needed for fatigue analysis.

- **Materials** describes the different types of material data that can be processed, and other settings that affect the handling of material property groups**.**
- **Analysis Runs** provides details of the implementation of fatigue analysis methods.
- **Results** handles the outputting of results produced by the various analysis engines.

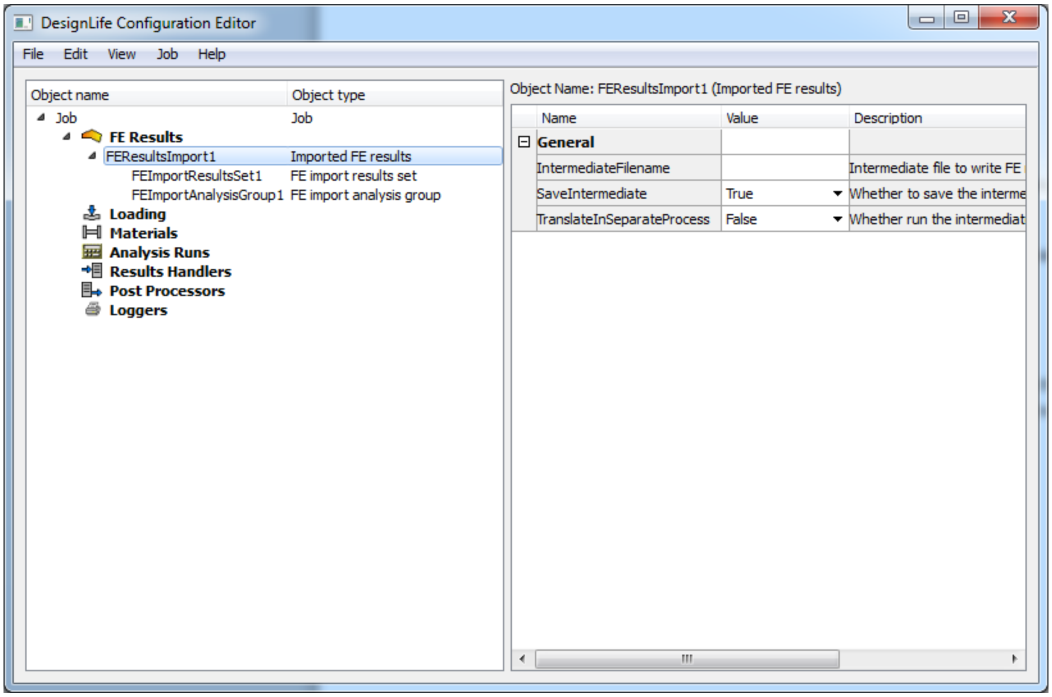
2 FE Results

nCodeDT uses FE analysis results to generate the fluctuating stresses and strains required for fatigue life prediction.

- Most fatigue analyses are carried out on the basis of elastic stresses.
- Elastic strains may also be used.
- In some special cases, it may be appropriate to use elastic-plastic strains and stresses from a non-linear analysis.
- Spot weld calculations are more typically based on element or grid-point forces, representing the cross-sectional forces in the spot welds.
- Seam weld calculations may use FE stresses directly, or alternatively, seam weld stresses may be derived from grid point forces.
- Temperatures may also be used.
- When analyzing composite materials, additional FE results other than the normal stresses and strains may be required—for example, averaged matrix or fibre stresses or strains, and the fibre orientation distribution tensor.
- "Virtual strain gauges" may be applied to the model and used to recover strain (and/or stress) histories for correlation with test.
- Normally, nCodeDT also requires the FE model geometry (for example, position of nodes and element connectivity).

Each nCodeDT configuration requires an **Imported FE results** object. Normally this will have at least one child **FE import results set** and one **FE import analysis group**.

Fig. 2-1 FE Results setup—required objects

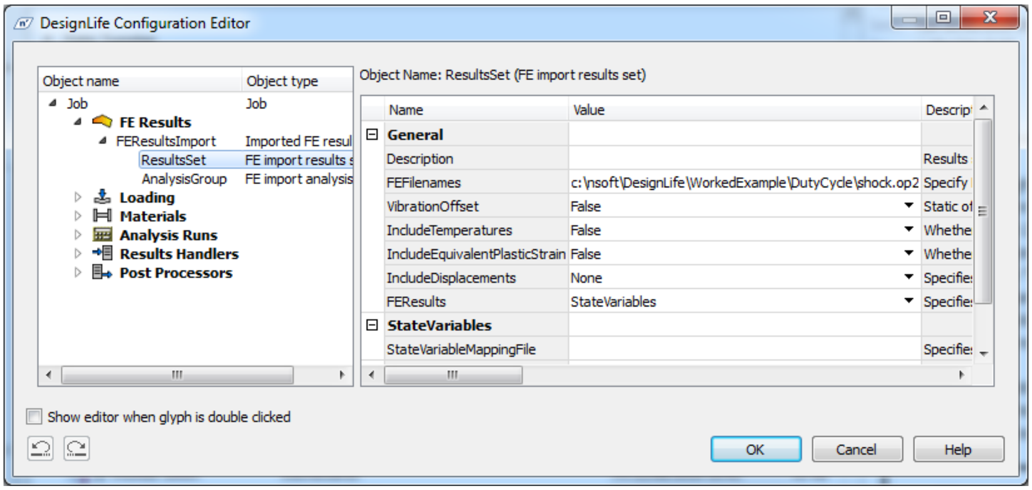


The **FE import results set** defines the FE results files that will be used for the job. These files should contain the necessary FE results and model geometry (FE). The **FE import analysis group** defines the type of FE data that will be used for the analysis, the groups of nodes and/or elements that will be processed, and other settings related to the handling of FE results. The only circumstance in which these two objects are not required is if there is an existing valid intermediate (FEI) file defined on the **Imported FE results** object, resulting from a previous successful translation.

2.1 FE Import Results Set

The FE import results defines the list of FE results files that will be used for the analysis. At least one of these must include the FE model geometry.

Fig. 2-2 FE import results set properties



The other properties are:

- **VibrationOffset**. This is set to True if the FE results file is to be used to provide a static offset loadcase for a vibration fatigue analysis. For all other applications, it should be set to False.
- **IncludeTemperatures**. This is set to True if temperatures need to be read in from the FE results. If temperatures are not required, this should be set to False.
- **IncludeEquivalentPlasticStrain**. This is set to True if equivalent plastic strains need to be read from the FE results. These can be used in conjunction with the Dang Van analysis engine. They allow the fatigue properties of the material to be modified to take into account the forming strain accumulated in the manufacturing process. The equivalent plastic strain should be available in the FE results for at least one of the load steps or loadcases. If equivalent plastic strains are not required, this property should be set to False.
- **IncludeDisplacements**. This property specifies whether to write nodal displacements to the intermediate file. Nodal displacements are required by the Animation analysis engine. The available options are:
  None: Do not include displacements.
  FullModel: Include displacement results from all nodes in the model.
  Outline: Only include displacements from nodes on the outline edges of the model (as seen in the *FE Display* glyph's *Outline* plot).
- **FEResults**. This is set to *Standard* if standard FE results are to be used. This is the normal setting and allows nCodeDT to recover standard FE results from the relevant output files.

Some FE analyses, however, may not use the standard data sets to store results values.

In particular, structural analyses for *composite* materials that have been carried out in a strongly coupled manner with *Digimat* (using Digimat to provide a user material model) will store the analysis results (stresses and/or strains and other important information such as material orientation tensors) in the output file as State Variables.
In this case this property should be set to *StateVariables*.

If *FEResults=StateVariables*, another property will become available: *StateVariableMappingFile*.
In general, state variables may be used to save any scalar value, so to store a 3D stress tensor, 6 state variables are required. In order to use state variables, nCodeDT needs to know how to map the state variables to the required stress and strain tensors.
Usually, there will be sufficient information stored with the state variables to allow them to be mapped automatically, and in this case, this property may be left blank.
Otherwise, the name of the Digimat interface file (e.g., .aba or .ans file in the case of ABAQUS or ANSYS) should be entered here, including the path if required. This contains the information necessary to identify the stress and strain components stored as state variables.

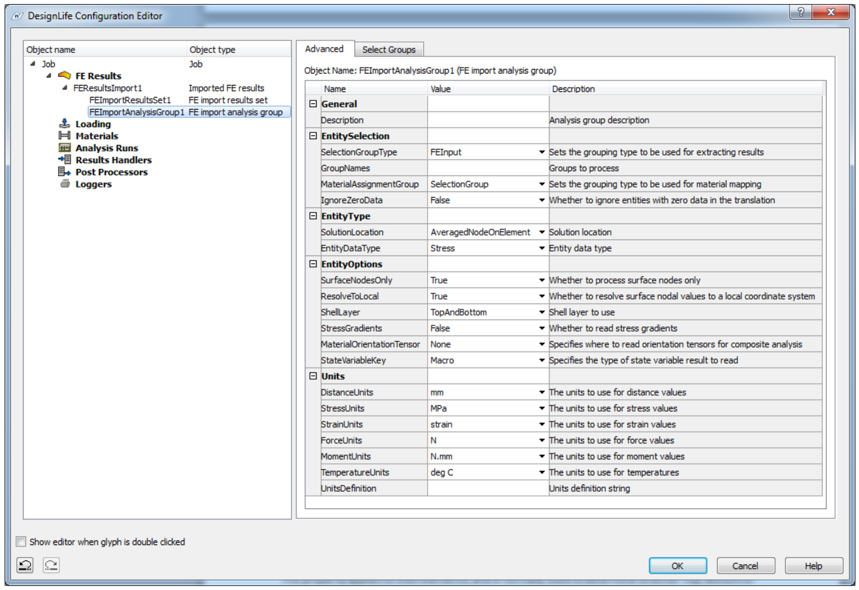
2.2 FE Import Analysis Group

This section addresses the properties of the FE Import Analysis Group. An FE import analysis group in nCodeDT is a group of FE entities (nodes, elements) for which the FE results will be translated and analyzed in the same way. On the basis of the settings of the FE import analysis group, nCodeDT translates data from the FE results files and writes them to an intermediate (FEI) file for onward processing.

An FE import analysis group in nCodeDT is a group of FE entities (nodes, elements, or virtual gauges) for which the FE results will be translated and analyzed in the same way.

Each analysis group can be based on a number of subgroups, or selection groups. These selection groups may be of different types, depending on the SelectionGroupType setting.

Fig. 2-3 FE import analysis group properties



2.2.1 SelectionGroupType

SelectionGroupType defines the type of subgroup or selection group used to define the analysis group. SelectionGroupType can have the following settings:

Table 2-1 SelectionGroupType settings

FEInput	Use the group information defined in the FE Input glyph. If a subset of the model is displayed in the FE Input glyph, for example by property set or material group, or by picking nodes/elements and labelling them in a user-defined group, this information is passed on via the metadata. If the whole model is displayed in the FE Input glyph, all FE entities of appropriate type will be included in the selection group.
Property	FE entities are grouped by property set. This is a very convenient way of grouping FE entities. For example, when analyzing a car body structure using NASTRAN, there may be very many different panels in the vehicle body, each of which will be represented by a different NASTRAN property set.
Material	Groups are identified according to the FE material assigned to each entity.
ElementSet	Element sets are sets of elements that are defined and exist in the FE results files. See also the note at the end of “Modeling Guidelines, Grouping and Required FE Results” on page 300 about the change in default settings (later versions of MSC.NASTRAN, may produce op2 files that do not contain the necessary element set information).

For all these types of groups, enter the names of the groups under GroupNames, use a wildcard (*) or select the groups interactively using the dialog on the “Select Groups” tab.

There is one other possible setting for SelectionGroupType, which requires some additional properties to be set:

Table 2-2 SelectionGroupType settings = UserSet

UserSet	User-defined set of nodes or elements. If this option is chosen, additional properties must be set. These are:	
	UserSetType	FromFile or FromIDs
	SetEntityType	Element or Node
	UserSetIDs	If UserSetType=FromIDs, this property should comprise the list of entities to be processed, e.g., 1,2,4,6-9,11
	UserSetFilename	If UserSetType=FromFile, enter here the name of the file containing the user set definition

2.2.2 MaterialAssignmentGroup

Subgroupings of FE entities are particularly important in the case of a model with different materials or surface conditions, because the FE entities then need to be organized so that properties can be set appropriately for the different parts of the model. The MaterialAssignmentGroup settings control the organization of the FE entities in the material map, which is used to assign different properties to different parts of the model.

Table 2-3 MaterialAssignmentGroup settings

SelectionGroup	The model will be organized using the type of group defined by SelectionGroupType.
Material	The model will be organized by FE material.
Property	The model will be organized by FE property sets.

2.2.3 IgnoreZeroData

If this property is set to **True**, any entities for which the FE results are all zero, or for which data cannot be found, will be excluded from the translation and any subsequent analysis. This is a useful way of making sure that time is not wasted carrying out analysis at locations where there are no results, for example if FE results have just been recovered for certain element sets.

2.2.4 SolutionLocation

SolutionLocation defines the locations (nodes, elements, etc.) from which the FE results will be recovered and at which they will be processed by nCodeDT. The options are as follows.

Element

Results will be recovered from the element centroid. If element centroid results are not available in the FE results file, nCodeDT will average the stresses from the nodes.

If results from state variables are being used, e.g., for a composite fatigue analysis, this setting is required.

NodeOnElement

Unaveraged, node-on-element results will be used.

AveragedNodeOnElement

Averaged node-on-element results will be used. It is more common for the FE results file to contain unaveraged nodal results. In this case, nCodeDT will average the results to the nodes. Where a node is on the boundary of a selection group, averaging will not include contributions from elements that lie outside the selection group. When a node is on the boundary of more than one selection group, it will be evaluated more than once, each time considering contributions from elements within the relevant selection group only. For a definition of selection group, see "SelectionGroupType" above.

If the raw, unaveraged FE results are not in a common coordinate system (for example, thin shell data is typically provided in element coordinates), they will be transformed to the global coordinate system before averaging. It should be noted that the averaging of shell results at nodes can be problematic because of the differing local coordinate systems between elements and using element centroid results for shells is commonly recommended. nCodeDT will not average results from different types of element, for example, when shells are attached to solids.

Note For LS-DYNA results there are no nodal results in the FE, so the element result is used when selecting a nodal solution location.

SpotWeld

If SolutionLocation is set to SpotWeld, nCode DT will look for the results that can provide the cross-sectional forces and moments through each spot weld and also the thicknesses of the attached sheets. For details, see the section on the Spot Weld Analysis Engine.

SeamWeld

The seam weld analyzer makes life predictions based on the stresses at the weld toe, root, or in the weld throat. The stresses at the weld toe can be determined in four possible ways:

1. Use unaveraged nodal stresses calculated by the FE code.
2. Calculate the strains, and hence the stresses for each node-on-element, based on the displacements and rotations of the nodes.
3. Derive the stresses from grid point forces.
4. Calculate the stresses from solid elements.

If stresses are to be used, ideally these will have been generated using CUBIC stress recovery in NASTRAN. MSC.Nastran, NX Nastran, and Optistruct Version 9 all support CUBIC stress recovery. If the CUBIC stress recovery is not available (for example, if you are using ANSYS or ABAQUS) reasonable results may be achieved using second order shells for the weld and weld toe elements.

Three alternative stress recovery approaches are now available.

- The first requires that grid point forces are available (e.g., by requesting GPFORCES in NASTRAN or NFORC in ABAQUS). Note that the method requires linear shells, and may give misleading results if the analysis includes large deformations (non-linear geometry).
- The second requires that nodal displacements and rotations are available (e.g., by requesting DISPLACEMENTS in NASTRAN or U in ABAQUS). This method also requires linear shells. It may also give misleading results if the analysis includes large deformations, or for warped elements.

  See the section "Seam Weld Analysis Engine" on page 264 for details.

  If SolutionLocation = SeamWeld, some additional properties need to be set: See the section on the"Seam Weld Analysis Engine" on page 264 for details.
- The third calculates the stress at the point of interest and the associated bending ratio from solid element stresses. See "Calculation of Stresses from Solid Elements" on page 294 in the Seam Weld Analysis Engine section.

Table 2-4 Solution = Seamweld, additional properties

EntityDataType	Stress, ForceMoment
DistanceUnits	m, ft, in, mm, km, miles, um

StressUnits	MPa, Pa, ksi, psi, ksf, psf (for EntityDataType = Stress only)
ForceUnits	kN, N, lbf (for EntityDataType = ForceMoment only)
MomentUnits	N.m, kN.m, N.cm, N.mm, ft.lbf, in.lbf (for EntityDataType = ForceMoment only)
SeamWeldType	FilletOverlap, LaserOverlap, LaserEdgeOverlap, SolidWeld, CombinedFilletOverlap, or Generic
WeldEndElements	Include, Exclude
WeldResultLocation	NodeOnElement, MidElementEdge

If SeamWeldType = SolidWeld, some additional properties need to be set, as described in Table 2-5. See "Seam Weld Analysis Engine" on page 264 for details.

Table 2-5 Solution = Solidweld, additional properties

WeldDefinitionFilename	Name of the weld definition XML file
MaxWeldDepth	Maximum weld depth in mm
NumWeldLayers	Number of layers through the thickness to use in the calculation

StrainGauge

The strain gauge analyzer allows stress and strain histories to be generated for virtual strain gauges. The position and type of the gauges must defined either in the FE Input glyph, or in an external file. If SolutionLocation = StrainGauge, two additional properties appear:

Table 2-6 Solution = strain gauge, additional properties

StrainGaugeSource	MetaData or File. If set to MetaData, the gauge information must be present in the metadata. This will be the case if the gauges are defined in the FE Input glyph. Alternatively, the gauge definitions may be imported from a file.
StrainGaugeFilename	Filename for gauges

AdhesiveBond

If SolutionLocation is set to AdhesiveBond, nCodeDT will look for nodal forces and moments in the FE results files (for example, GPFORCE in NASTRAN or NFORC in ABAQUS). These are used to calculate the line forces and moments required by the adhesive bond analysis engine. The adhesive bond analysis engine allows durability predictions in the form of J-integral and safety factor calculations to be made for bonded joints between two steel or aluminium sheets, based on relatively coarse shell models.

If SolutionLocation = AdhesiveBond, some additional properties need to be set: See the section on the Adhesive Bond Analysis Engine for details.

Table 2-7 Solution = adhesive bond, additional properties

EntityDataType	ForceMoment
DistanceUnits	mm or m
ForceUnits	N or kN
MomentUnits	Nm, Nmm or kNm (must match force and distance units)

Note also that the adhesive bond analysis engine requires the sheet thicknesses to be present in the FE results file. These are not present by default in ABAQUS results and must be requested (STH).

WeldHotSpot

If *SolutionLocation* = *WeldHotSpot* nCodeDT will look for stress results from solid elements. The stress tensor at the weld toe will be obtained by extrapolation of the surface stress from 2 or 3 points near to the weld.

If *SolutionLocation* = *WeldHotSpot* some additional properties need to be set: See the section ("Calculation of Weld Hot Spot Stress from Solid Elements" on page 312) on the Seam Weld Analysis Engine for details.

Table 2-8 SolutionLocation = WeldHotSpot additional properties

WeldDefinitionFilename	Name of the weld definition XML file
MaxWeldDepth	Maximum weld depth in mm
OffsetType	Ratio,Distance
OffsetMethod	Default,Custom
ExtrapolationPoints	Two,Three
MeshQuality	Fine,Course
OffsetValues	Comma separated list of 2 or three distances (mm) or ratios

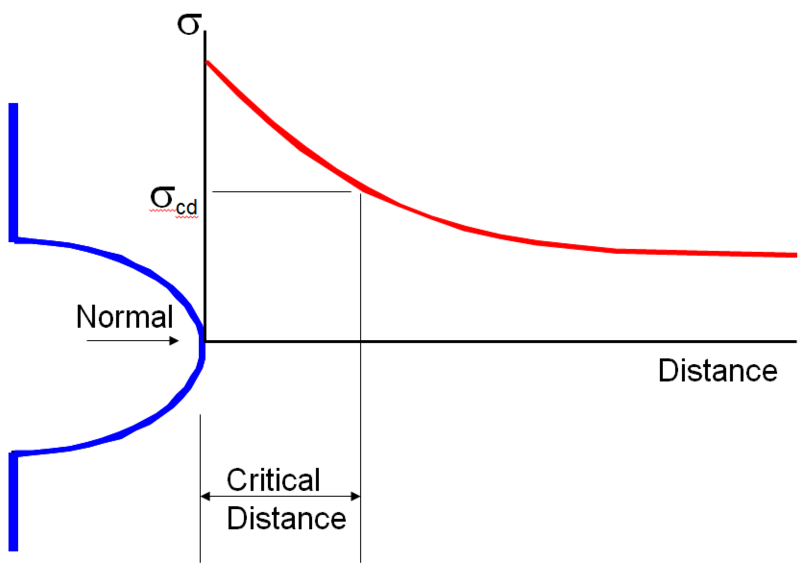
CriticalDistance

This option is used to account for notch sensitivity and uses a stress a distance below the surface to represent the stress that will actually drive the fatigue process. This is applicable for stress, linear strain and stress/strain results and for solid element surface nodes only.

If the *SolutionLocation* is set to *CriticalDistance* for each node the stress will be found beneath the surface at a position defined by the *CriticalDistance* and the node's surface normal direction (Figure 2-4). The surface normal will be the average of the outward surface normal of the free element faces adjacent to the node. These results can be then averaged (*CriticalDistanceType=AveragedNodeOnElement*) or left unaveraged (*CriticalDistanceType=NodeOnElement*).

If *ResolveToLocal* is **False** then the stress returned for analysis will be the 3D stress tensor obtained directly from the critical distance below the surface. If *ResolveToLocal* is **True** then the stress will be transformed to a local coordinate system that has the outward surface normal as the Z-axis (see 2.2.7 ResolveToLocal).

Fig. 2-4 Determination of the stress at the Critical Distance



If the SolutionLocation = CriticalDistance the following additional properties need to be set:

Table 2-9 SolutionLocation=CriticalDistance additional properties

CriticalDistance	This is the distance in FE DistanceUnits defined as the Critical Distance. This is used to obtain the sub-surface stress at this distance below the surface in the node's surface normal direction.
CriticalDistanceType	*NodeOnElement* Results at the interpolation points will remain unaveraged. *AveragedNodeOnElement* Results from interpolation points will be averaged.

2.2.5 EntityDataType

The following types of data can be processed.

Stress

Stresses will be recovered from the FE results file. Stresses can result from a static, transient, or modal analysis. Stresses will normally be from an elastic analysis.

LinearStrain

If this option is selected, nCodeDT will be expecting elastic strain results. Strains can be from static, transient or modal analysis. “Linear” here means elastic. It does not preclude modeling of geometric non-linearity, e.g., contact.

ForceMoment

This option is used for Spot Weld, Seam Weld, or Adhesive Bond analyses only. See the relevant sections on the Spot Weld Analysis Engine, Seam Weld Analysis Engine, or Adhesive Bond Analysis Engine for details. Element or Grid Point forces and moments are used. Results may be from a static, transient, or modal analysis.

ContactForce

This option is used for spot welds only when modelled in LS-DYNA as Hexahedral elements with tied contact between the Hexahedral and the shell elements representing the sheets.

Displacement

This option is used only for Seam Weld analyses. See the section on the Seam Weld Analysis Engine for details. Nodal displacements and rotations are used to determine the strains, and hence the elastic stresses for each node-on-element. Results may be from a static, transient, or modal analysis.

Vibration

Vibration results are required to support the Vibration Load Provider, in order to make fatigue calculations based on PSD or swept-sine loadings. nCodeDT looks in the FE results for a stress frequency response function (FRF). This takes the form of a complex stress response to unit excitation at a number of frequencies. Because the FRF will be combined with a PSD or swept-sine definition in the Vibration Load Provider to generate the local response, the unit excitation and the PSD of swept sine should have consistent units. For example, for a PSD loading, if the PSD is defined in acceleration units of g^2/Hz, then the FE analysis should provide the frequency response to an excitation of amplitude 1g across the frequency range.

StressAndStrain

This option is valid for subsequent E-N or Multiaxial E-N analysis, and will normally be used when the user does not wish to rely on notch methods (Neuber, Hoffmann-Seeger, etc.) to estimate the level of local plasticity, but rather to use elastic-plastic FE to calculate the local elastic-plastic strains and stresses. Results will normally be time-step. This option may also be used with the Virtual Strain Gauge (strain gauge analysis engine).

2.2.6 SurfaceNodesOnly

This True/False option is very useful when analysis is to be carried out at nodes in solid elements. Normally, fatigue cracks initiate at the surface of components, and if it is assumed that this is so, it makes sense to reduce the size of the problem by automatically excluding all but the surface nodes from the analysis by setting this option to True. This setting makes no difference to the handling of shell elements, or if the SolutionLocation is not NodeOnElement or AveragedNodeOnElement.

2.2.7 ResolveToLocal

Most fatigue cracks initiate at the free surface of components. At a free surface, due to the lack of traction (shear) or significant pressure, there exists a state of plane stress, with all non-zero principal stresses lying in the plane of the surface. If we look at these stresses in a local coordinate system having an outward surface normal in the z direction, the stresses can be treated as 2-D in the x-y plane.

There are a number of advantages of having 2-D stresses, chiefly that we can extract the principal stresses/strains by solving quadratic rather than cubic equations, and that we can carry out Multiaxial Assessment (see the section on the Standard SN Analysis Engine for details of MultiaxialAssessment methods). FE results from thin shell elements will normally be 2-D (in the element coordinate system) by default. For solid elements, however, the stresses are normally in a global coordinate system.

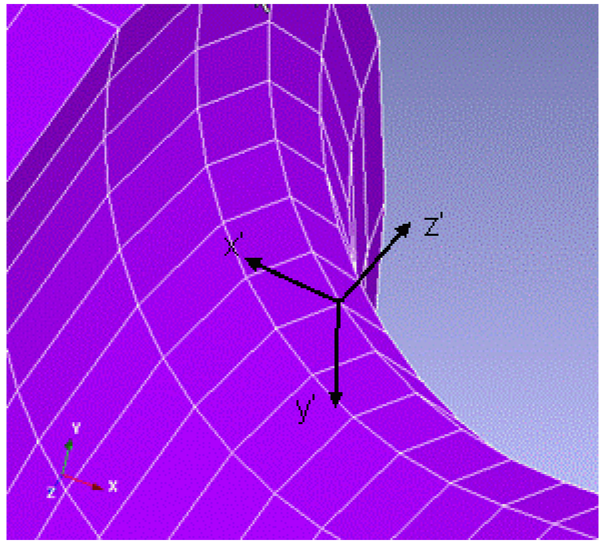
Setting ResolveToLocal to **True** (which should be combined with the SurfaceNodesOnly option) will result in the transformation of the stresses to a local coordinate system that has the outward surface normal as the z-axis—see Figure 2-5. The transformed stresses will be stored in the intermediate (FEI) file and flagged as 2-D.

For solids, the surface normal will be the average of the outward surface normals of the free element faces adjacent to the node. For shells, the surface normal will be the average of the element normals at the node.

Note that shell results already in element coordinates do not need to be resolved. However, if using the AveragedNodeOnElement option based on unaveraged nodal FE shell results in element coordinates, use of the ResolveToLocal option is appropriate. In this case, nCodeDT transforms the stresses to a common (global) coordinate system before averaging them, and then transforming to the surface resolved coordinate system based on the averaged normal at the node.

Note that if the local z-axis is parallel to the global z-axis, the local x-axis will be in the same direction as the global x-axis. Otherwise the direction of the local x-axis (x′ in Figure 2-5) is in the direction of the cross product of the global and local z-axes.

Fig. 2-5 Stresses at surface nodes on solid models are best transformed to local coordinates.



2.2.8 Stress Gradients

This option is applicable for stress or strain results, and can be used in conjunction with shell elements, or with nodal results on solid elements (surface nodes only). If True, this option recovers the gradient of the stress (or strain) tensor in the surface normal direction. The purpose of this is to support the stress gradient correction (see the section on the Standard SN Analysis Engine for details). In the case of shell elements, the stress gradient is calculated from the difference between the top and bottom surface stresses divided by the sheet thickness. In the case of solid elements, the stress gradient is determined from the stresses in the layer of elements adjacent to the surface, averaged to the node. In both cases, the stress gradient is defined as the gradient of stress in the direction of an outward surface normal. Stress gradients are not currently available when a Vibration Fatigue Load Provider is being used or composite elements are being analyzed, and are not applicable for Multi-axial EN, creep, spot weld, or seam weld analysis.

2.2.9 ShellLayer

This property applies to shell elements, and is normally used to determine whether Top, Bottom or TopAndBottom surface stresses are recovered and used for analysis. For example, in NASTRAN thin shell elements, Top refers to layer Z2 and Bottom to Z1. The additional option All is intended primarily for use with composite shell elements. In this case, FE results will be recovered for all layers and section points. For example, if there are 5 layers and 3 section points per layer, results will be recovered at 15 locations per element.

2.2.10 MaterialOrientationTensor

This property applies to composite fatigue analyses only, otherwise it should be set to None. Composites in general have anisotropic structures and properties. The material orientation tensor describes the microstructure in terms of the distribution of fibre orientations at each calculation point (element, layer, section point) in the structure, and this information is required if nCodeDT is to take into account the anisotropy of fatigue properties in the analysis. The material orientation tensor will normally be derived from a simulation of the manufacturing process.

For composite fatigue analyses, the applicable options are StateVariables or ASCII file. If the FE analysis has been run in a strongly coupled analysis with Digimat, the orientation tensors should be available as state variables in the FE results file (ANSYS .rst or ABAQUS .odb). Otherwise, the ASCII file option should be selected, and an additional property OrientationTensorFile will appear. The file specified in OrientationTensorFile must be in the following Glyphworks-compatible CSV format:

```
#HEADER
#CHANTITLE
Orientation Tensors
#TITLES
Element,Layer Number,Section Point,a11,a22,a33,a12,a23,a13
#KEYWORDS
ElementID,LayerNumber,SectionPoint,a11,a22,a33,a12,a23,a13
#DATATYPES
LONG,LONG,LONG,FLOAT,FLOAT,FLOAT,FLOAT,FLOAT,FLOAT
#DATA
1,1,1,0.9483,0.04747,0.004257,-0.003031,2.619E-5,-0.005304
1,2,1,0.7847,0.209,0.006283,-0.002284,-3.154E-5,-0.003902
1,3,1,0.5095,0.4853,0.005148,-7.342E-4,-2.02E-5,-0.003999
1,4,1,0.2087,0.7901,0.001193,0.007548,2.444E-5,-0.002749
```

Optionally, a default orientation tensor may be defined. This orientation tensor will be used if an orientation tensor for the current calculation point is not found in the #DATA section. If a default orientation tensor is required, lines like the following should be included in the file before the #DATA section.

```
#TESTMETADATA
"OrientationTensor.Default,1,0,0,0,0,0"
```

In this example the default value of a11 is 1.0 and the other components of the default orientation tensor are zero.

2.2.11 StateVariableKey

This property is used only if the FE results are from state variables, when the FE analysis has been run strongly coupled with Digimat. Since this process can generate not only Macro (composite) stresses, but also averaged Matrix or fibre stresses, this property should be used to define which will be used for the fatigue analysis. Typically, SN curves are determined for the composite and so the default setting is Macro.

2.2.12 IncludeSpotWeldNuggets

This option applies to spot weld analyses only, and, if True, FE results will be translated for the spot weld nugget. Normally, if a spot weld is adequately dimensioned, i.e., the diameter is sufficiently large compared to the sheet thickness, it will not fail through the nugget. If this is the case, processing time can be reduced by excluding the spot weld nuggets from the calculation.

3 Loading

Loading events and duty cycles are defined in nCodeDT by types of load provider.

A load provider is an object that provides loading information to the different types of analysis engine. This in turn allows the analysis engines to use the FE analysis results to calculate the fluctuating stresses (or strains) required for fatigue analysis.

The following types of load provider are available:

- Time series
- Constant amplitude
- Time step
- Vibration
- Temperature
- Duty cycle—including aero spectra
- Hybrid

The Time Series, Constant Amplitude, Hybrid and Time Step Load Providers generate a stress or strain history. The Hybrid Load Provider can also include temperature information. The Vibration Load Provider is a little different in that it incorporates rainflow counting, and so delivers a rainflow count to the analysis engine. The Duty Cycle Load Provider allows complex duty cycles to be built up from other load providers. Load provider types can be mixed within a duty cycle, and duty cycles can be nested (duty cycles within duty cycles), with the exception that Vibration Load Providers cannot be mixed with other types of event.

The different types of load provider are described in more detail below.

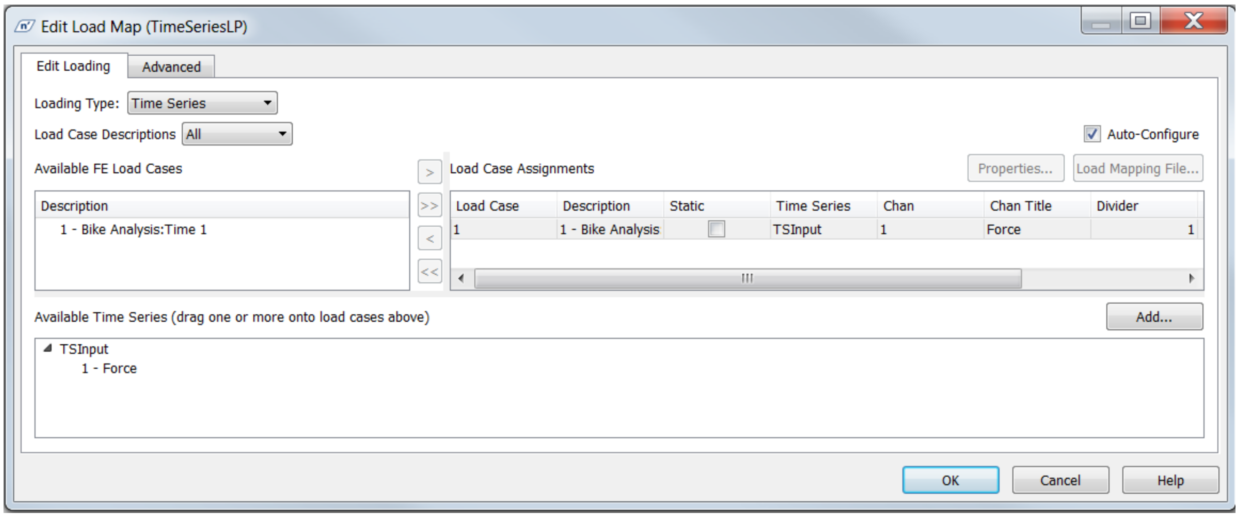
3.1 Time Series Load Provider

The Time Series Load Provider provides the information that an analysis engine needs to create a stress or strain history by scaling and linear superposition of FE results. The most obvious and common application is to combine a number of static loadcases to generate a stress or strain history. In this case (static superposition), the stress tensor history as a function of time $\sigma_{ij}(t)$ is calculated as follows:

$$\sigma_{ij}(t) = \sum_k \frac{(P_k(t) \cdot ScaleFactor_k + Offset_k) \cdot \sigma_{ij,k,static}}{Divider_k}$$

...where $P_k(t)$ is the load time history data, $\sigma_{ij,k,static}$ is the static FE stress result, and k is the loadcase ID. The correspondence between the static loadcases and available time history channels, together with scale factors, dividers and offsets, is set up on the load mapping interface illustrated in Figure 3-1.

Fig. 3-1 Time series load mapping interface



The loading history channels must have a common sample rate and number of data points. If Auto-Configure is set, the load provider will be configured automatically to process all FE load cases and time series channels in order.

The user must ensure that the loadcase scaling is applied correctly to obtain the stress history. By default, the divider and scale factor are set to 1 and the offset to 0. It is common practice to either use unit loadcases and loading histories in consistent units, in which case the default values can be accepted, or, where non-unit loads are used in the FE (e.g., 1000 N), to set this value as the Divider.

At least 1 loadcase must vary, but with multiple simultaneous loadcases, some of them can be static. Typically this would be used for FE loadcases that represent static weight or assembly stresses. It would give the same effect as mapping a time-series channel with a flat line = 1. The divider, scale factor and offset functions can still be used in the same way with or without a static value. A static stress will influence the fatigue damage due to mean stress effects. It may also avoid or contribute to static failure.

Please note the following:

1. Where bodies have significant accelerations, the accuracy of fluctuating stress results can be improved through the use in the FE calculation of inertia relief.
2. Where the behavior of the structure is more dynamic (but still linear), linear superposition can still be used, but using modal results (modal superposition). $P_k(t)$ are now the modal participation factors, e.g., from a modal transient analysis, or multi-body simulation with flexible bodies, and $\sigma_{ij,k,static}$ is replaced by $\sigma_{ij,k,modal}$, the modal stress tensor, where k is the mode ID. Residual vectors or Craig-Bampton modes will normally be included.

3. The same principles can be applied to linear superposition of strains, or in the case of spot welds, of the line forces and moments adjacent to the bond to the cross-sectional forces and moments, or in the case of adhesive bonds, to the line forces and moments adjacent to the bond.

3.2 Constant Amplitude Load Provider

The constant amplitude load provider also uses scaling and linear superposition to create, in the case of static stresses, a simple two-point stress history.

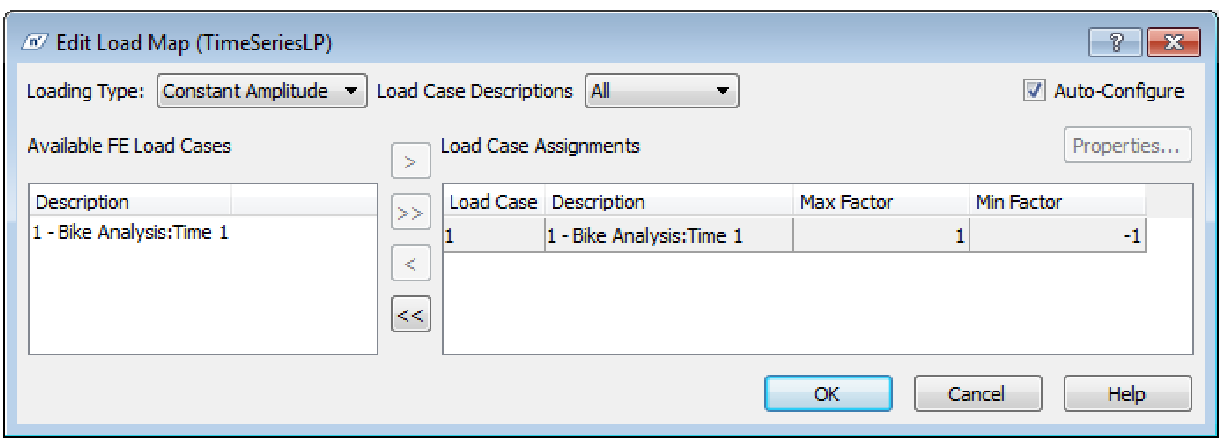
First point: $\sigma_{ij} = \sum_{k} (MaxFactor)_k \cdot \sigma_{ij,k,static}$

Second point: $\sigma_{ij} = \sum_{k} (MinFactor)_k \cdot \sigma_{ij,k,static}$

...where k is the loadcase ID.

Using a single loadcase and taking the default Max and Min factors (1 and -1) results in the simulation of a fully reversed constant amplitude loading.

Fig. 3-2 Constant amplitude load mapping interface



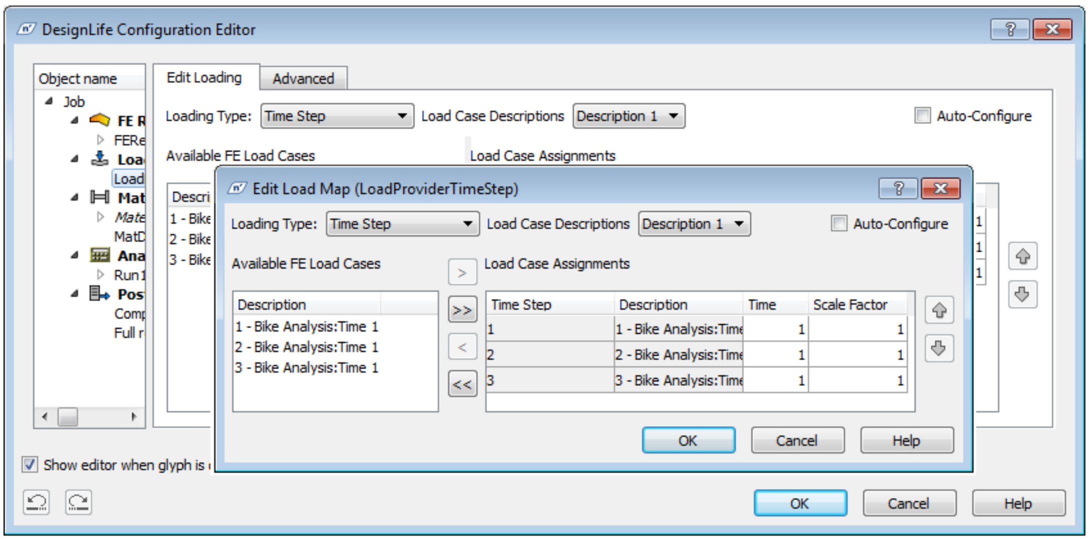
As before, the same principles apply to strain and force/moment. Seam welds use stresses.

3.3 Time Step Load Provider

The Time Step Load Provider allows the user to create a time history by simply selecting a series of time steps (or static loadcases) from the available list in the desired sequence.

The load map interface for the Time Step Load Provider is illustrated in Figure 3-3:

Fig. 3-3 Time step load mapping interface



The **Time Step** column on the interface will be filled in automatically if the time information is available in the FE results file. The time values may also be entered or edited manually.

Scale Factor allows scale factors to be applied to individual time steps.

Note Time information is not normally required for fatigue analysis, except when the Time Step Load Provider is a child of a Hybrid Load Provider. In this case, the time information is required in order to obtain the correct sequence and superposition of loading.

3.4 Temperature Load Provider

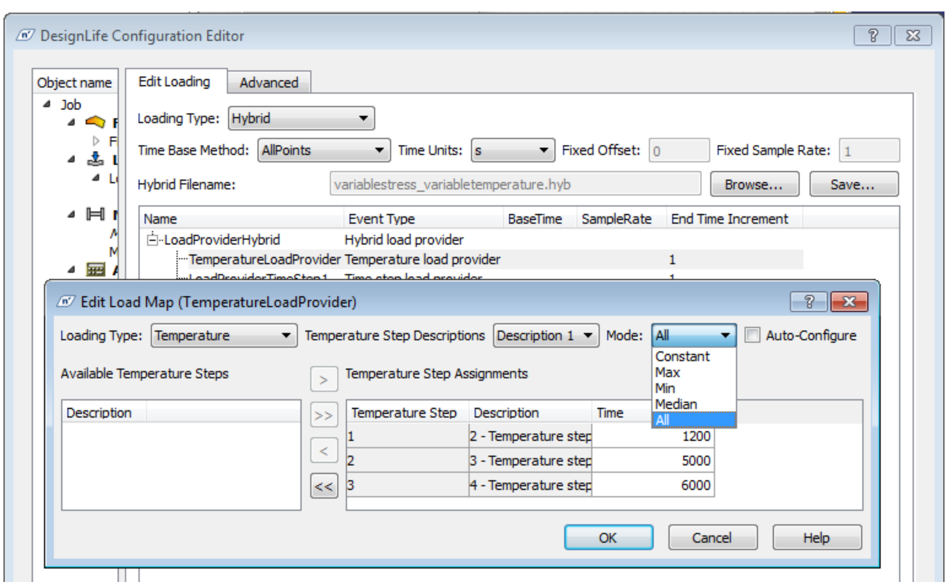
The purpose of the Temperature Load Provider is to provide a temperature-time history that can be used as input for a high temperature fatigue calculation. A Temperature Load Provider must be a child of a Hybrid Load Provider, because the temperatures are used in conjunction with another loading that defines the stress (or strains). The load mapping interface for the Temperature Load Provider is very similar to that of the Time Step Load Provider, requiring the selection from a list of available temperature-time steps.

It has one additional setting—the Mode (Constant, Max, Min or Median)—that determines which temperature is passed on to the analysis engine.

- **Constant** assumes that the temperature distribution in the model is not time-varying; one temperature case should be selected.
- **Max**, **Min** or **Median**, as the names suggest, pass on the maximum, minimum or median (average of maximum and minimum) temperature for each FE entity to the analysis engine.
- **All** generates a temperature time history, where the time value of each temperature step is also used. This mode can be used to perform cycle-by-cycle temperature-corrected calculations.

The load mapper for the Temperature Load Provider is shown below.

Fig. 3-4 Load map interface for temperature load provider



3.5 Hybrid Load Provider

The Hybrid Load Provider provides an alternative way of creating a stress (or strain) time history through a combination of different types of load provider. A

Hybrid Load Provider must have at least one child *Load Provider*, which must be one of the following types:

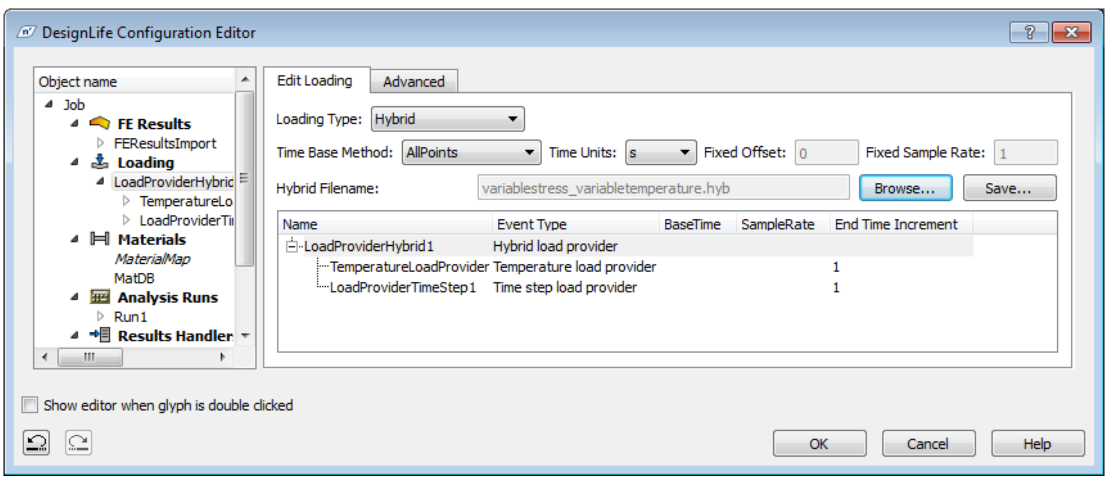
- Time Series
- Time Step
- Constant Amplitude

It may also contain a *Temperature* load provider.

The load mapping interface for the Hybrid Load Provider looks very similar to that of the Duty Cycle Load Provider (see Figure 3-5), the difference being that in the Hybrid Load Provider the stress histories from the child load providers are superimposed rather than considered in sequence.

This provides a very flexible way of creating stress histories, and allows, for example, a cycle simulating a mechanical load to be superimposed on a slow thermal transient.

Fig. 3-5 Hybrid Load Provider interface



The stress history from the above arrangement would be:

$$\sigma_{ij,total}(t) = \sigma_{ij,Time\ Series}(t) + \sigma_{ij,Time\ Step}(t) + \sigma_{ij,Const\ Amplitude}(t)$$

If the component load providers all create stresses at the same time values (t), then the function is obvious. If the time points differ between the child load providers, the following steps are taken.

1. The set of times (t) for which the total stress history is determined is the set of all times from the individual load providers.
2. If the stress for a particular load provider is not available at any required time value, it is interpolated from the stress values from adjacent times generated by that load provider.
3. The start point for determining the stress history is the earliest start time from any of the child load providers.

4. The end point for determining the stress history is the latest end time from any of the child load providers.
5. Where the time history starts before the beginning of a given load provider, data from that load provider will be inserted to fill the gap, repeating as necessary.
6. Where the time history ends after the end of a given load provider, data from that load provider will be added to fill the gap, repeating as necessary.
7. Some addition properties have to be set on the child load providers to set the timing and “padding” correctly.
   - The **Time Step** Load Provider requires the property EndTimeIncrement to be set, which defines the time interval between the end of the time step load provider and any repetition of data required by steps 5 and 6 above
   - The **Constant Amplitude** Load Provider requires the BaseTime and Frequency values to be set.

3.6 Vibration Load Provider

The Vibration Load Provider is different from the other load providers in that it does not provide the analysis engines with the information required to generate a time history of stress, but rather determines the rainflow count directly. The primary purpose of the Vibration Load Provider is to support the simulation of vibration shaker table tests, driven by PSD (power spectral density) multiple PSDs, swept-sine, sine dwell or sine on random loading.

The inputs to the Vibration Load Provider are:

- A Frequency Response Function (FRF) from finite element analysis. This provides the complex stress response $X_{ij}(f)$ to a unit excitation across the frequency range of interest.
  The complex stress response can be accepted as real and imaginary, or magnitude and phase.
  In the case of magnitude and phase, a conversion to real and imaginary is performed.
- Multiple FRF's in the case of the multi-PSD analysis

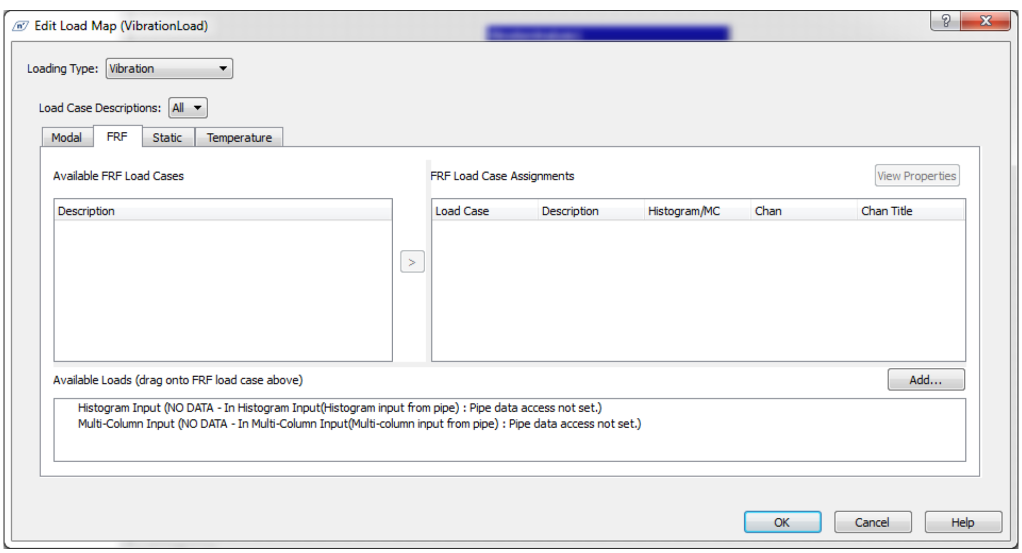
Note DesignLife checks that the input file contains results that are real/imaginary, or magnitude/phase. If they are not then they are not recognised as vibration results.

- A modal stress result. The stress at each mode (real) when combined with a set of modal participation factors (complex) can be used to calculate the FRF (complex). This modal stress result will typically be a much smaller results file having results only for each mode of the system.
- A definition of the loading, either:

- A PSD. This can be equally spaced data in time-series or histogram format, or unequally spaced multi-column data (PSD vs. Frequency).
- Multiple PSDs and cross spectra. The PSD as above and cross spectra defined by multicolumn data of magnitude and phase values.
- A swept-sine loading definition, which can be equally or unequally spaced (Amplitude vs. Frequency).
- A sine dwell definition of a single frequency and amplitude.
- A sine on random definition. An input PSD as above and a number of sine tones defined by their frequency and amplitude.
- Optionally:
    - A number of static FE loadcases can be included, so that for example assembly stresses may be superimposed on the vibration stresses.
    - A number of temperature load cases. The selected temperature is used for each entity in the subsequent isothermal SN analysis.

These inputs are defined via the load map interface, which is illustrated in Figure 3-6.

Fig. 3-6 Load mapping interface for the Vibration Load Provider



The Vibration Fatigue Load Provider can be used with the Standard SN, Standard EN (but Gray Iron analysis is not supported), Short Fibre Composite, Seam Weld, Spot Weld and Custom analysis engines. Note also that, although the Vibration Load Provider can be used in a duty cycle, it cannot be mixed with other types of load provider, and it can only be used with “Independent” event processing. The Vibration Load Provider predicts rainflow counts:

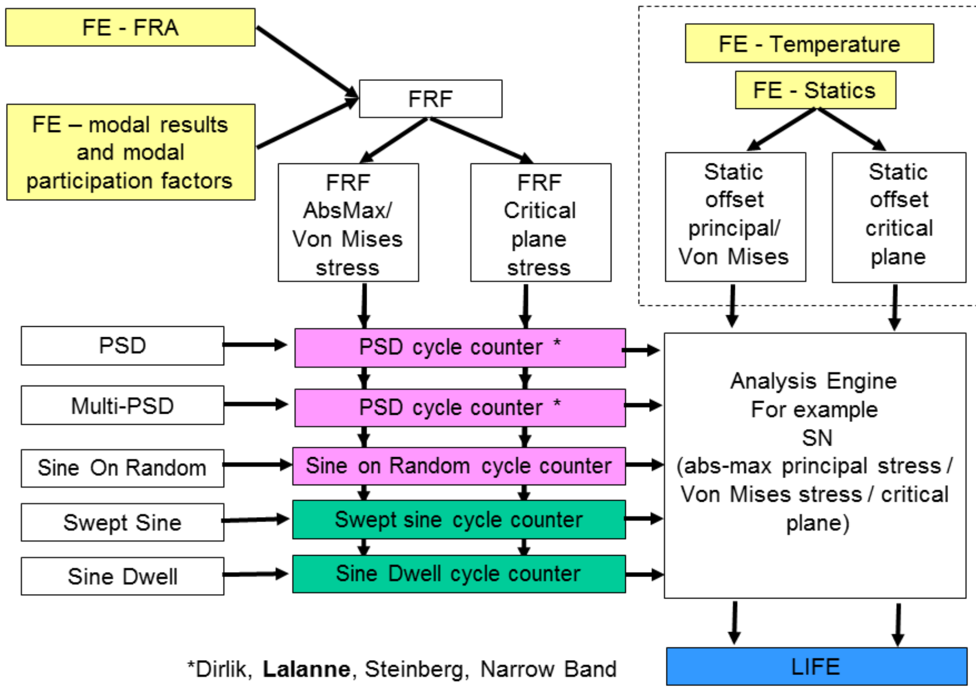
- Based on various stress combination options, depending on which analysis engine is being used.

    - For the S-N analysis engine only the AbsMaxPrincipal, SignedVonMises, and CriticalPlane options are allowed.
    - For the E-N analysis engine only the AbsMaxPrincipal, SignedVonMises, SignedShear, CriticalPlane and TypeBCriticalPlane options are allowed.
    - For the Seam Weld analysis only the AbsMaxPrincipal and CriticalPlane options are allowed.
    - For the Spot Weld analysis only the Standard and Custom options are allowed.
        - NASTRAN and Optistruct results are not currently supported for spot weld with vibration loading.
    - For the Short Fibre Composite analysis engine only the AbsMaxPrincipal and CriticalPlane options are allowed.
- From PSD loadings, a local response PSD is calculated as an intermediate step, and statistical methods are used to predict the probability density function (PDF) of stress range, and hence the rainflow count. In effect, the PDF of stress range is the probable rainflow count per second.
- From swept-sine loadings, a deterministic method is used to predict the rainflow count directly.
- The predicted rainflow count includes fractional cycles. For sine on random loading, this is truncated at very high stress and extremely low probability, see section 3.6.5.
- Other loading statistics are also calculated.

Note	LS-Dyna is not currently supported for vibration loading.

The fatigue analysis process options using the Vibration Fatigue Load Provider are illustrated in Figure 3-7.

Fig. 3-7 Summary of fatigue analysis process using Vibration Load Provider



3.6.1 Resolution of FRF to Abs Max, Signed Von Mises, or Critical Plane

Principal stresses are normally determined from the *eigenvalues* of the stress tensor (a real symmetric matrix). Determination of the *Absolute Maximum Principal* (or *Signed Von Mises*) from the FE-derived FRF is difficult, because this requires a solution for the eigenvalues of a complex matrix. Up until nCode 11, nCodeDT worked around this issue by finding the eigenvalues of the *Real* and *Imaginary* parts of the FRF $X_{ij}(f)$ separately: This is the *Fast* method defined by the *Compex-Algorithm* property.

The Abs Max Principal FRF is then taken to be composed of the Real and Imaginary eigenvalues with the largest magnitude.

The *Exact* method solves the full complex matrix, but typically takes twice the time to complete. In general, the exact algorithm will take around 40 iterations to achieve convergence; however it may take significantly more for poorly conditioned systems. The maximum number of iterations (*ComplexMaxIterations*) should be increased when the desired convergence is not being reached in the number of iterations currently performed. Increasing this *ComplexMaxIterations* may sometimes be necessary, but will increase calculation time.

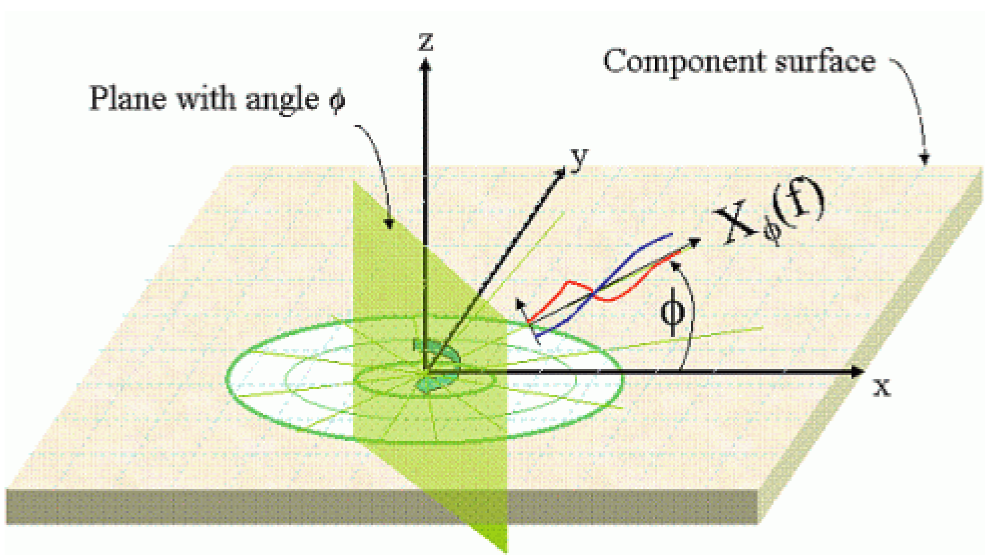
The Von Mises FRF is only possible to calculate using the exact method. It also uses the standard Von Mises formula based on the complex eigenvalues that the exact algorithm returns.

Another more theoretically correct method is to use a *critical plane approach*, where the FRF is resolved onto critical planes. This takes longer because it needs more computation. It needs 2-D stresses (see the section on FE Results). The FRF is resolved onto multiple planes (the critical plane is the plane with the most predicted fatigue damage). The planes on which the FRF is determined have normals that lie in the plane of the physical surface, i.e., in the x-y plane of the 2-D stress results coordinate system. The orientation of each plane is defined by the angle ϕ made with the local x-axis.

When resolving in preparation for a multiPSD analysis with cross spectra, we require the answers to remain complex. This is so that we can maintain the phase information for use in the sum over the matrix elements (see section 3.6.3). This restricts the possible combinations to AbsMaxPrinciple or CriticalPlane, both only using the complex exact algorithm.

For Seam Weld, Spot Weld, Standard EN, Short Fibre Composite and Custom analysis engines, and for LoadingMethod = MultiplePSD, the *Fast* method is not appropriate and therefore the *Exact* is forced. If the *Fast* method is selected this will be set back to *Exact* when the analysis is run.

Fig. 3-8 Resolution of FRF for critical plane analysis



The FRF on each plane is calculated from:

$$X_\phi = \frac{X_{xx} + X_{yy}}{2} + \frac{X_{xx} - X_{yy}}{2} \cdot \cos\phi + X_{xy} \sin\phi$$

...where ϕ can take the values 0, 10, 20, 30, ...170 degrees.

3.6.2 Combination of Load PSD and FRF to Get Local PSD (PSD Cycle Counter)

When the loading is a PSD, [call this P(f)], the next step is the calculation of a local PSD, i.e., the predicted local stress response. Call this G(f).

When using the Abs Max Principal:

$$G_{AMP}(f) = P(f) \cdot \left|\mathrm{X}_{AMP}(f)\right|^2$$

When using Critical Plane:

$$G_{\phi}(f) = P(f) \cdot \left|\mathrm{X}_{\phi}\right|^2$$

It is quite likely that the PSD of the loading and the FRF are not available at the same frequencies f. The values of both may be interpolated to any frequency at which the response PSD needs to be calculated.

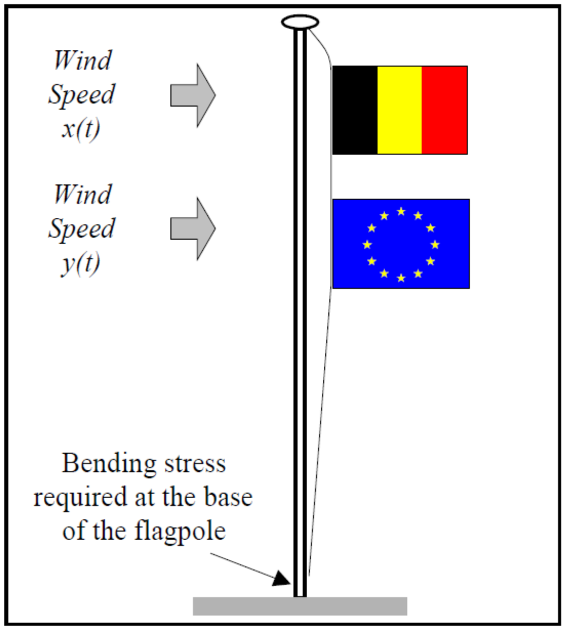
It is important that the FRF and the loading PSD be represented in consistent units. For example, if the PSD is in units of g^2/Hz, the FRF should represent the frequency response to a 1 g loading across the frequency range.

3.6.3 Multiple Random Inputs (Multi-PSD Analysis)

Note Multi-PSD analysis is only supported for the Standard SN analysis engine at the present time.

The standard PSD analysis described above in "Combination of Load PSD and FRF to Get Local PSD (PSD Cycle Counter)" only considered the response on an element resulting from a single random input. This section introduces the analysis of multiple random inputs. Figure 3-9 shows a simple example of a flagpole with two flags flying at different heights. A typical application would be to determine the bending stress at the base of the flagpole as a result of the two random wind speeds seen by the flags.

Fig. 3-9 Example Multiple PSD input



The time history of wind speed at the location of the flags can be recorded using anemometers. PSDs of wind speed can be calculated from these; however, the PSDs alone do not provide information on the phase relationships between the two measured time histories.

With multiple random processes we also require the sequential relationship between the two time histories. If the two flags are far enough apart then the wind speed witnessed by one will be completely independent of that on the other. As they are moved closer together then a correlation between the two time histories will be seen. The two time histories are correlated because the random wind turbulence incident on one flag has a sufficiently large range of influence to also affect the response at the other.

To calculate the bending stress at the base of the flagpole it is insufficient to simply sum the reactions from the two input PSDs, instead we must sum the reactions from the input and cross-power spectra. The cross-power spectra contain information on the joint statistics of the two processes. If the two processes are correlated then the sequencing effects may act to increase or decrease the base bending stress depending on whether the forces are in or out-of-phase. For a mathematical explanation of this see Newland [2]. The single-sided PSD function of stress at the base of the flagpole, $G_{zz}(f)$, is therefore determined by:

$$G_{zz}(f) = \sum_{a=1}^{2} \sum_{b=1}^{2} H_a(f) \cdot H_b(f)^* \cdot W_{ab}(f)$$

$H_1(f)$ and $H_2(f)$ are the transfer functions relating stress at the base of the flagpole to wind load incident on flags *x* and *y*, respectively. The asterisk indicates the complex conjugate. $W_{11}(f)$ and $W_{22}(f)$ are the PSD functions of wind speed at flags *x* and *y*, respectively, and $W_{12}(f)$ and $W_{21}(f)$ are the cross-power spectral density functions. For a general loading with *n* simultaneous forcing functions the

PSD of stress can be obtained from:

$$G_{zz}(f) = \sum_{a=1}^{n} \sum_{b=1}^{n} H_a(f) \cdot H_b(f)^* \cdot W_{ab}(f)$$

The analysis route is to:

- Resolve the input FRFs to the critical planes (or Abs, Max, Principal) in complex form. This ignores the setting of Complex Algorithm and always uses the exact one.
- Take each resolved FRF result and interpolate to the required frequencies.
- Use equation above and take the modulus resulting in the power spectrum response for each resolved angle (or Abs. Max. Principal).
- Pass the calculated response to the standard PSD cycle counter.

References

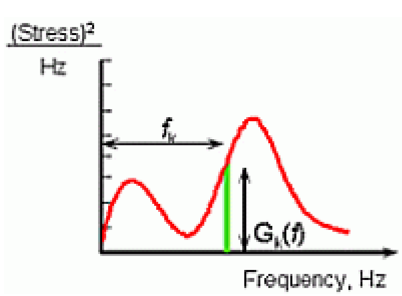
1. HalfPenny A. (1999). "A frequency domain approach for fatigue life estimation from Finite Element Analysis". International Conference on Damage Assessment of Structures (DAMAS 99) Dublin.
2. Newland DE. (1984). "An introduction to random vibrations and spectral analysis (2nd edition), Longman Inc., New York.

3.6.4 PSD-rainflow Prediction Using Lalanne, Dirlik, Narrow Band, Steinberg (PSD Cycle Counter)

When the loading is a PSD, the next step in the calculation is the prediction of the rainflow cycle count (or more correctly the PDF of rainflow range) from the local PSD stress response.

Now the statistics, including the rainflow count, of the local stress response can be predicted based on the shape of the PSD G(f) and specifically in terms of the spectral moments as defined in Figure 3-10:

Fig. 3-10 Definition of spectral moments



n^{th} Moment of area under PSD

$$m_n = \sum f^n \cdot G(f) \cdot \delta f$$

For example, according to the theory of S. O. Rice (1954), we can predict:

$$E[0] = \sqrt{\frac{m_2}{m_0}}$$

The expected number of zero crossings:

$$E[P] = \sqrt{\frac{m_4}{m_2}}$$

The expected number of peaks:

The irregularity factor: $\gamma = \frac{E[0]}{E[P]} = \frac{m_2}{\sqrt{m_0 \cdot m_4}}$

The rms (root mean square) stress is simply the square root of m0.

There are a number of different theories for the prediction of the PDF of stress range. All of these assume that the process (stress history) represented by the PSD of stress is stationary, random, Gaussian and ergodic.

Lalanne

Probably the best solution currently is the Lalanne theory, whereby the probability density function N(S) of stress range S is given by:

$$N(S) = E[P] \cdot p(S)$$

...where N(S) is the number of stress cycles of range S N/mm2 expected per second, E[P] is the expected number of peaks (as defined above), and p(S) is defined as follows:

$$p(S) = \frac{1}{2rms}\left\{\frac{\sqrt{1-\gamma^2}}{\sqrt{2\pi}} e^{\frac{-S^2}{8rms^2(1-\gamma^2)}} + \frac{S \cdot \gamma}{4rms} e^{\frac{-S^2}{8rms^2}}\left[1 + erf\left(\frac{S \cdot \gamma}{2rms\sqrt{2(1-\gamma^2)}}\right)\right]\right\}$$

...where:

$$\mathrm{erf}(x) = \frac{2}{\sqrt{\pi}} \int_0^x e^{-t^2} dt$$

The probability density function of stress range is in effect the predicted rainflow count per second.

Dirlik

Dirlik proposed an empirical closed form solution to estimate the pdf of stress range N(S), following extensive computer simulations using the Monte Carlo technique and different signals. It looks somewhat complicated but it is still only a function of the four moments of area of the PSD, m_0, m_1, m_2 and m_4.

It can be summarized in the following expression:

$$N(S) = E[P] \cdot p(S)$$

...where (*S*) is the number of stress cycles of range *S N/mm*^{2} expected per second, *E*[*P*] is the expected number of peaks (as defined above), and p(S) is defined as follows:

$$p(S) = \frac{\frac{D_1}{Q} \cdot e^{\frac{-Z}{Q}} + \frac{D_2 \cdot Z}{R^2} \cdot e^{\frac{-Z^2}{2 \cdot R^2}} + D_3 \cdot Z \cdot e^{\frac{-Z^2}{2}}}{2 \cdot \sqrt{m_0}}$$

...where:

$$D_1 = \frac{2 \cdot (x_m - \gamma^2)}{1 + \gamma^2} \quad D_2 = \frac{1 - \gamma - D_1 + D_1^2}{1 - R} \quad D_3 = 1 - D_1 - D_2$$

$$Z = \frac{S}{2 \cdot \sqrt{m_0}} \quad Q = \frac{1.25 \cdot (\gamma - D_3 - D_2 \cdot R)}{D_1} \quad R = \frac{\gamma - x_m - D_1^2}{1 - \gamma - D_1 + D_1^2}$$

...and: $x_m = \frac{m_1}{m_0} \cdot \sqrt{\frac{m_2}{m_4}}$

The Dirlik equation is based on the weighted sum of the Rayleigh, Gaussian and exponential probability distributions.

Narrow Band

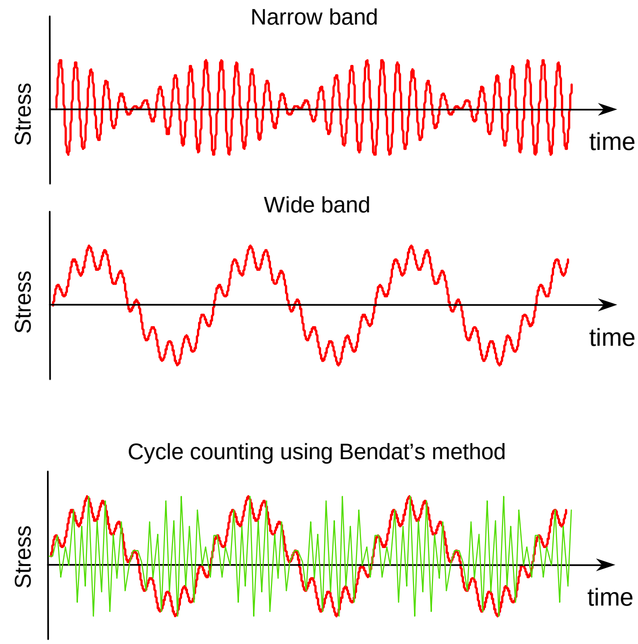
In 1964, Bendat, based on the work of Rice, proposed the following solution:

$$N(S) = E[P] \cdot \frac{S}{4 \cdot m_0} \cdot e^{\frac{S^2}{8 \cdot m_0}}$$

This assumes that the probability of a stress peak is estimated by the Rayleigh distribution, and that, since the signal is narrow band, for every peak, there is a corresponding trough.

Bendat's narrow-band solution tends to be conservative for broad-band signals. The reason for this lies in the assumption that peaks are matched with corresponding troughs of similar magnitude to form stress cycles. This effect is illustrated in Figure 3-11. A narrow-band time signal is characterized by each peak having a corresponding valley of similar magnitude. In comparison, a broad-band time signal is characterized by smaller waves riding on a low frequency carrier. Because Bendat assumes that all positive peaks are matched with corresponding valleys of similar magnitude, the damage is grossly exaggerated for broad-band signals.

Fig. 3-11 Why Bendat's method is conservative



Steinberg

Rice found that for narrow-banded signals, the pdf of peaks tended towards a Rayleigh distribution while broad-band signals tend to a Gaussian distribution. Steinberg assumed that the pdf of stress range would also tend to a Gaussian distribution and proposed a solution based on discrete cycles at multiples of the RMS (Root Mean Square) amplitude. This approach is used extensively in the field of electronics testing. The Steinberg equation can be represented as follows:

$$N(S) = E[P] \times \left| \begin{array}{l} 0.683 \; at \; 2 \times RMS \\ 0.271 \; at \; 4 \times RMS \\ 0.043 \; at \; 6 \times RMS \end{array} \right.$$

3.6.5 Sine-on-random-rainflow Prediction (Sine-on-random Cycle Counter)

When the loading is a SineOnRandom, the response SineOnRandom spectrum is first obtained. The next step in the calculation is the prediction of the rainflow cycle count (or more correctly the PDF of rainflow range) from the local SineOn-Random stress response.

Similarly as in section 3.6.4, statistics based on spectral moments are calculated first.

The formulations for E[0] and E[P] remain the same but use the following modification to the spectral moments:

$$m_n = \sum f^n . G(f) . \delta f + \frac{1}{2} \sum_1^N f_i^{\,n} {b_i}^2$$

With f_i and b_i the frequency and amplitudes of the *N* sine tones added to the noise.

The sine-to-random power ratio a_0^2 is also calculated. It is a measure of the relative importance of the harmonic part versus the random part in terms of power. It is, defined as:

$${a_0}^2 = \frac{{\sigma_s}^2}{{\sigma_r}^2} = \frac{\sum_{i=1}^N {b_i}^2}{2{\sigma_r}^2}$$

With σ_r the RMS value of the noise and σ_s the RMS value of the sine waves.

The prediction of the PDF of stress range is based on the theory by S.O. RICE. It assumes that the process (stress history) represented by the PSD of stress is stationary, random, Gaussian, ergodic and narrowband.

Such as in the case of the NarrowBand method for the PSD cycle counter (see previous section 3.6.4), this solution tends to be conservative for broadband signals (i.e. where more than one mode dominates).
However, this conservatism is reduced when ${a_0}^2 > 1$.

Another assumption is that the frequencies of sine tones are within the frequency range of the noise. The violation of this assumption will lead to only slightly conservative results.

Note that the original method assumes that the frequencies of the sine tones are not multiples of each other. Should this be the case, the results may potentially show a high level of conservatism, especially for ${a_0}^2 > 10$.

The probability density function *N(S)* of stress range *S* is given by:

$$N(S) = E[P] \cdot p_S(S)$$

...where *N(S)* is the number of stress cycles of range *S* expected per second, *E[P]* is the expected number of peaks (as defined in 3.6.4), and *p(S)* is defined as follows:

$$p(S) = \frac{S}{2} . \int_0^\infty \frac{x}{2} . e^{-\left(\frac{\sigma_r^2 x^2}{8}\right)} J_0\left(\frac{xS}{4}\right) \prod_{i=1}^N J_0\left(\frac{x b_i}{2}\right) dx$$

with J_0 the Bessel function of the first kind of order zero and σ_r the RMS of the random PSD.

The equation giving the PDF of stress range $p(S)$ is integrated numerically.

Although numerical integration is very efficient to provide accurate results, it may fail to describe the rainflow cycle distribution at very high stress ranges (where probabilities of occurrence are extremely low). For this reason we truncate the rainflow cycles array at very high stress ranges, when the numerical integration breaks down. This has no significant impact on Damage.

If the sine tone amplitudes become too small after being scaled by the local response, we ignore them and filter them out before doing the calculation. Here, 'too small' is defined as below the noise floor, which is set in a property on the load provider. This can sometimes result in not having any sine tones left. In this situation we fall back on a narrowband RandomPSD calculation which is the calculation sine on random is based on and gives the same results faster.

3.6.6 Combination of SineDwell Definition with FRF to Get Local Rainflow Count

The Sine-Dwell definition provides a relationship between the amplitude of excitation *A*, its frequency *f* and how many cycles will be repeated *N*.

Assuming a steady-state response, the amplitude of the local stress response (we will call this S_{AMP}) is given at frequency *f* by:

$S_{AMP} = A.|X_{AMP}(f)|$

...or:

$S_{\varphi} = A.|X_{\varphi}(f)|$

The rainflow cycle count is obtained from the local stress response and is given by:

$$N(S) = \begin{cases} N & if\ S = 2.S_{AMP} \\ 0 & otherwise \end{cases}$$

None of the methods for creating a rainflow count (Lalanne, Dirlik, Narrow Band, Steinberg, Swept-Sine, Sine on Random, and Sine Dwell) predict mean stress values.

3.6.7 Combination of Swept-sine Definition with FRF to Get Local Rainflow Count

The swept-sine definition provides a relationship between the amplitude of excitation and frequency. Let us call this A(f).

Assuming a steady-state response, the amplitude of the local response (we will call this B) is given as a function of frequency by:

$$B_{AMP}(f) = A(f) \cdot \left| X_{AMP}(f) \right|$$

...or:

$$B_{\phi}(f) = A(f) \cdot \left| X_{\phi}(f) \right|$$

The number of cycles at each frequency and amplitude is predicted based on the sweep rate, and the resulting cycles classified into a rainflow cycle histogram.

3.6.8 Superposition of Static Load Case

None of the methods for creating a rainflow count (Lalanne, Dirlik, Narrow Band, Steinberg, Swept-Sine, Sine on Random, and Sine Dwell) predict mean stress values. The rainflow cycle count that these methods produce has zero mean stress for all cycles. Although vibration in a linear system cannot generate mean stresses, mean stresses may exist and may be modeled due to a variety of reasons, for example, gravity loading or assembly stresses. For this reason, nCodeDT allows a mean stress to be included from a static offset loadcase. The mean stress applied is either the Abs Max Principal or Critical Plane stress extracted from the static offset loadcase stress tensor, and is used to shift the mean of all cycles in the predicted rainflow count.

3.6.9 Inclusion of a Temperature Load Case

The Vibration Load Provider can include temperatures on the material using a multi-temperature SN material curve. These temperatures can be set by *material mapping*, in which case they are *constant* across the whole material.

If *varying* temperatures are required, these can be loaded from the finite element model as *temperature load cases*. The temperature load cases must be included in the static offset results set. This is because they are a constant background property on top of which the vibration is applied.

To be able to load temperatures, the *include temperatures* flag must also be set to *true* on the static offset results set.

3.6.10 Vibration Load Provider properties

The properties of the Vibration Load Provider are accessed from the **Property Editor** (right click on the load provider). The properties vary slightly, depending on whether the loading is a PSD , Multi-PSD, Sine Sweep, Sine Dwell or Sine on Random.

Fig. 3-12 Vibration Fatigue Load Provider properties—LoadingMethod=PSD

Object Name: VibrationLoad (Vibration load provider)

Name	Value	Description
General		
ResultsSetName		Name of the FE results set this loading applies to
ModalLoading	False	Whether to use modal decomposition to calculate the Frequency Response Function
LoadingConfigSource	Custom	Specifies where the configuration of the loading should come from, metadata or custom.
ComplexAlgorithm	Fast	Algorithm used to calculate complex eigenvalues and eigenvectors
CycleCountBins	2048	Number of bins to use when cycle counting
FrequencySelectionMethod	LoadingAndFRFFrequencies	Method used for selecting frequency points
ScaleFactors		
ScaleFactorAllStaticLoadCases		The scale factor to apply to all static loadcases
DividerAllStaticLoadCases		The divider to apply to all static loadcases
Custom		
LoadingMethod	PSD	Vibration loading method
InterpolationMethod	LogLog	Method to use when interpolating the loading spectra
PSD		
PSDCycleCountMethod	Lalanne	PSD cycle count method
ExposureDuration	1	Exposure duration in seconds
NoiseFloor	-120	Noise floor expressed as decibel reduction from peak

Fig. 3-13 Vibration Fatigue Load Provider properties—LoadingMethod=SineSweep

Object Name: VibrationLoad (Vibration load provider)

Name	Value	Description
General		
ResultsSetName		Name of the FE results set this loading applies to
ModalLoading	False	Whether to use modal decomposition to calculate the Frequency Response Function
LoadingConfigSource	Custom	Specifies where the configuration of the loading should come from, metadata or custom.
ComplexAlgorithm	Fast	Algorithm used to calculate complex eigenvalues and eigenvectors
CycleCountBins	2048	Number of bins to use when cycle counting
FrequencySelectionMethod	LoadingAndFRFFrequencies	Method used for selecting frequency points
ScaleFactors		
ScaleFactorAllStaticLoadCases		The scale factor to apply to all static loadcases
DividerAllStaticLoadCases		The divider to apply to all static loadcases
Custom		
LoadingMethod	SineSweep	Vibration loading method
InterpolationMethod	LogLog	Method to use when interpolating the loading spectra
SineSweep		
SweepType	LinearHzPerSec	Sweep type using sweep rate provided
SweepRate	1	Sweep rate applied in terms of the sweep type
NumberOfSweeps	1	Number of sweeps

FrequencySelectionMethod

This property determines the frequencies at which the local stress response will be calculated. The options are:

- **LoadingFrequencies**. The frequencies at which the local response is calculated will be the same as the frequencies included in the load PSD or Sine Sweep.

- **FRFFrequencies**. The frequencies at which the local response is calculated will be the same as the frequencies at which the FRF was calculated.
- **EquallySpacedFrequencies**. The local response will be calculated at a set of frequencies to be specified by the user, using the properties MinFrequency, MaxFrequency, Frequency Interval described below.
- **LoadingAndFRFFrequencies**. The frequencies at which the local response is calculated will include all the frequencies included in the load PSD or Sine Sweep, and the FRF. This is a good default, as further refinement will not improve the precision of the calculation.

Note that increasing the resolution of the local PSD response will increase the computation time.

MinFrequency

If FrequencySelectionMethod=EquallySpacedFrequencies, this setting specifies the minimum frequency at which the local response is to be calculated.

MaxFrequency

If FrequencySelectionMethod=EquallySpacedFrequencies, this setting specifies the maximum frequency at which the local response is to be calculated.

FrequencyInterval

If FrequencySelectionMethod=EquallySpacedFrequencies, this setting specifies the frequency interval at which the local response is to be calculated.

InterpolationMethod

When the local response needs to be calculated at frequencies where the FRF or loading (PSD or SineSweep) is not defined, the necessary values are interpolated. Interpolation can be LogLog (Log-Log) or LinLin (Linear-Linear). Note that if data is required beyond the ends of the data, the last data point is used.

CycleCountBins

The rainflow cycle count (based on the continuous PDF) is discretized into a number of histogram bins. Increasing the number of bins increases precision at the expense of computation time. 2048 seems to be a reasonable compromise for most purposes.

LoadingMethod

PSD or SineSweep

If LoadingMethod = PSD, the next three properties are:

PSDCycleCountMethod

This defines the method used to predict the rainflow count based on the local PSD stress response. The options Lalanne, Dirlik, Steinberg and NarrowBand are described above.

ExposureDuration

By default, this is set to 1 second. This means that the rainflow count represents the likely rainflow count per second, and when we carry out a fatigue calculation, the result is the damage per second.

NoiseFloor

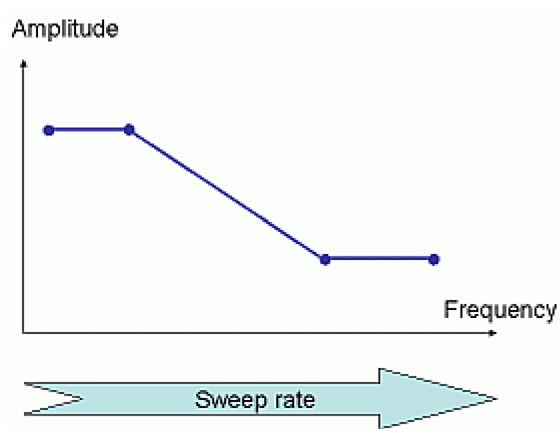
This defines the level in db below the peaks in the PSD which will be considered noise. The default value -120 db corresponds to six orders of magnitude below the peak in the PSD. Values below this limit are set to zero.

If LoadingMethod = SineSweep, the next three properties are:

SweepType

Each sine sweep definition defines a relationship between amplitude of excitation and frequency. The SweepType and SweepRate determine the way and the rate at which the calculation (or corresponding physical test) sweeps through the frequencies.

Fig. 3-14 Sine sweep definition



SweepType can have the following settings:

- **LinearHzPerSec**. The sine sweep will be linear through the frequencies at a rate defined by the property SweepRate in Hz per second.
- **LogOctavesPerMin**. The sine sweep will be logarithmic—at a constant number of octaves per minuted defined by the property SweepRate.

- **LogSecsPerDecade**. The sine sweep will be logarithmic, with a constant number of seconds per decade defined by the property SweepRate.

SweepRate

See the details of SweepType above.

NumberOfSweeps

Defines the number of complete sweeps (low to high frequency) to be carried out. Damage predicted will be for this number of sweeps.

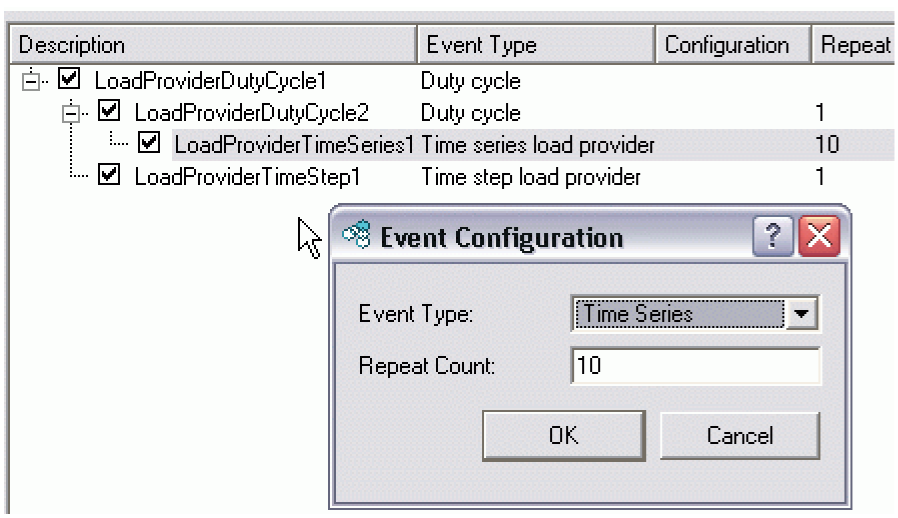
3.7 Duty Cycle Load Provider

The Duty Cycle Load Provider is a special type of load provider that allows a duty cycle to be constructed from sequences and repeats of the other load providers.

A duty cycle consists of a series of duty cycle items. Each duty cycle item consists of an event (as defined by a load provider) and a repeat count. The load providers can be of any type, except that Vibration Load Providers cannot currently be mixed with the other types that create time histories. Duty cycles can be nested, i.e., it is possible to have duty cycles within duty cycles. Events can appear multiple times within a duty cycle.

A simple duty cycle is illustrated (using the Duty Cycle Editor) in Figure 3-15.

Fig. 3-15 A simple duty cycle

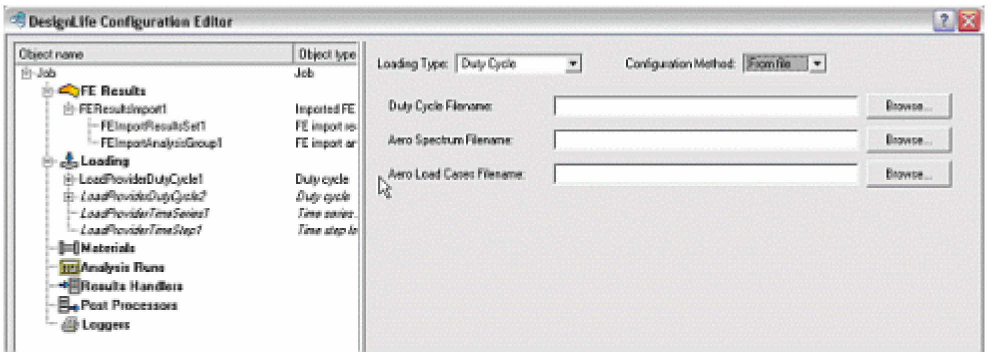


A duty cycle can also be defined by:

- **A duty cycle (.dcy) file**. The format of a duty cycle file is described in File Formats.

- **A GlyphWorks schedule (.sch) file**, although this permits only the definition of simple duty cycles consisting of time series events. This file may be created using the ScheduleCreate module. Insert the schedule file name in the **Duty Cycle Filename** field. (See Figure 3-16.) Note also that if a schedule file is piped into the CAE Fatigue glyph by means of a Time Series Input glyph, it will be interpreted to give a single time series consisting of a fully expanded schedule file. This may take a long time to process.

Fig. 3-16 Duty cycle configuration from file



3.8 Aero Spectrum Handling and Aero Spectrum Load Providers

In addition, a loading history may be defined by specifying Aero Spectrum (`.spe`) and Aero Load Case (`.lcs`) files. The Load Cases file describes a number of combinations of FE load cases that can be used to build up a load spectrum as commonly used in aerospace loading conditions. The Spectrum file defines the sequence of these loadcase combinations that will make up the loading spectrum. The formats of these files are described in detail in File Formats. nCodeDT interprets Aero Spectrum and Loadcases files as a duty cycle. Each flight block from the spectrum file is interpreted as a subsidiary duty cycle, and each loadcase combination of the flight is interpreted as an event of the subsidiary duty cycle.

4 Materials

The materials category in an nCodeDT configuration contains the definitions of material databases and material maps, where fatigue-related material properties can be assigned to different parts of the structural model and other relevant properties such as surface condition can be defined. This section deals briefly with the material properties required for the different analysis types, and with certain aspects of the material map.

4.1 Material Properties and Analysis Types

Every fatigue analysis requires Material properties. In nCodeDT, Material properties for analysis are retrieved from a material database, normally an nCode MXD database or the nCodeDT memory-based material database. (The memory-based material database allows material properties to be embedded in a process flow.) There is also partial support for the nSoft MDB database, but this does not support all the options available in nCodeDT. There is an additional type of material database object which is used only for short fibre composite fatigue analyses using the Digimat fatigue model.

nCodeDT has a number of different fatigue analysis engines, and some of those engines support a number of different methods. There are a number of different material data types, and each type is associated with a particular analysis engine/method. The different analysis engines, methods and material types are summarized in the following table.

Table 4-1 Summary of Different Analysis Engines, Methods, and Material Types

Analysis Engine	Method/Data Type	Description
Standard SN analysis engine	Standard SN	Parameterized (log-log) S-N curve
	SN mean multi-curve	Set of digitized S-N curves at different mean stress levels
	SN R-ratio multi-curve	Set of digitized S-N curves at different R-ratios
	SN Haigh multi-curve	Digitized Haigh (constant life) diagram
	Bastenaire SN	Parameterized dataset for S-N calculations using the Bastenaire formulation
	SN temperature multi-curve	Set of digitized S-N curves at different temperatures

Table 4-1 Summary of Different Analysis Engines, Methods, and Material Types

Analysis Engine	Method/Data Type	Description
Standard EN analysis engine	Standard EN	Parameterized stress-strain and strain-life curves to support the local strain approach using Coffin-Manson-Basquin formulation
	EN mean multi-curve	Parameterized stress-strain curve with a set of digitized strain-life curves at different mean stress levels
	EN R-ratio multi-curve	Parameterized stress-strain curve with a set of digitized strain-life curves at different R-ratios
	EN temperature multi-curve	A set of standard parameterized EN curves at different temperatures
	Gray Iron	Parameterized stress-strain and strain-life curves to support the specific Gray Iron local strain approach.
Standard Dang Van analysis engine	Dang Van	Parameters to support the Dang Van multi-axial safety factor method
Spot Weld analysis engine	Spot Weld	Parameterized S-N curve for prediction of sheet metal or nugget failure at spot welds using the Rupp method
Seam Weld analysis engine	Seam Weld	Individual or paired parameterized S-N curves (flexible and stiff) for use with the "Volvo" seam weld method
Standard Multiaxial EN analysis engine	Standard EN	Parameterized stress-strain and strain life curves to support the local strain approach using Coffin-Manson-Basquin formulation. Some additional parameters may be set for multiaxial analyses, but otherwise defaults will be used.
Strain Gauge analysis engine	Standard SN or EN	Only the modulus and Poisson ratio are required.
Adhesive Bond analysis engine	Adhesive Bond	Simple data set including the elastic modulus, Poisson ratio and threshold strain energy release rate. Any data type that includes modulus and Poisson ratio may be used to define the sheets.
Standard Assessment analysis engine	—	No material properties required.
Short Fibre Composite analysis engine	Short Fibre Composite Basquin Curves	Set of SN curves with Basquin formulation for different fibre shares (fibre orientations)
	Digimat SN	The SN curve is determined using a call to Digimat, based on the stress state or direction and the local fibre orientations.

Table 4-1 Summary of Different Analysis Engines, Methods, and Material Types

Analysis Engine	Method/Data Type	Description
	DigimatHaigh	A set of Haigh curves is obtained using a call to Digimat, based on the stress state or direction and the local fibre orientations.
Larson-Miller (creep)	Larson-Miller Curves	Collapses the results from the creep tests onto a single curve that is independent of temperature. See "The Larson-Miller Method" on page 316.
	Larson-Miller Polynomial	Fits a polynomial through the data. See "The Larson-Miller Method" on page 316.
Chaboche (creep)	Chaboche Creep	Based on a creep damage evolution equation. See "The Chaboche Method" on page 322 and "Chaboche" on page 315.
	Chaboche Fatigue	Allows for the use of a set of Chaboche temperature curves to calculate life for given stress cycles together with a temperature. See "Chaboche" on page 107.
	Chaboche Transient Fatigue	Allows for the use of a Chaboche temperature analysis to calculate life for given stress cycles together with temperature. See "Chaboche Transient" on page 110.
Safety Factor Engine (stress)		This performs stress based safety factor analyses. Several different SN based safety factor methods are supported (listed below). Each uses a different method for mapping the relationship between stress amplitude and the corresponding safety factor.
	Standard	Standard nCode method using material SN dataset
	MultiMeanCurve	Multiple mean curve using mean stress
	MultiRRatioCurve	Multiple mean curve using R-ratio
	Haigh	Haigh diagram method
Animation Analysis Engine	Nodal displacements	This is an implementation of the analysis engine that creates animation files. Animation files can be used by the FE display to animate CAE models.

The Standard SN and Spot Weld analysis engines also support custom formulations for the SN curve.

Because the format of the different material data types and associated analysis methods are intimately linked, they are discussed in more detail in the sections describing the individual analysis engines.

Note that the MXD database can store further material types. However, these are not currently supported in nCodeDT and are not discussed here.

4.2 Material Group Parameters

In addition to assigning material data sets to different parts (material groups) of the structural model, the material map also allows certain other parameters to be defined for each group, which affect the way the analysis is carried out.

These are:

- Scale Factor
- Offset
- KTreatment
- KUser
- KRoughness
- Weld diameter
- Default temperature
- ShapeFactor
- Adhesive Thickness
- Bond Line Offset
- Initial Crack Size

The availability of these settings for the different material types/analysis engines is summarized in the following table.

Table 4-2 Availability of Material Group Settings for Different Analysis Types

Group Property	Analysis Engine/Material Type Category							
	S-N	E-N	Dang Van	Spot-Weld	Seam-Weld	Multiaxial E-N	Adhesive Bond	Compos-ite
Scale Factor	✓	✓	✓	✓	✓	✓	✓	✓
Offset	✓	✓		✓	✓	✓		✓
KTreatment	✓	✓†◆				✓		
KUser	✓	✓†◆				✓		
KRoughness	✓	✓†◆				✓		
Weld diameter				✓				
Default temperature	✓*	✓*				✓		
ShapeFactor		✓◆				✓		
Adhesive Thickness							✓	
Bond Line Offset							✓	
Initial Crack Size							✓	

✓ Available

* Available for Multi-temperature curve
† Not available for Multi-mean or Multi R-ratio curves
◆ Not available for gray iron

Table 4-3 Availability of Material Group Settings for Different Analysis Types

Group Property	Analysis Engine/Material Type Category			
	Short Fibre	Creep	Animation [a]	Safety Factor
Scale Factor	✓	✓		✓
Offset	✓	✓		✓
KTreatment				✓
KUser				✓
KRoughness				✓
Weld diameter				
Default temperature		✓		
ShapeFactor				
Adhesive Thickness				
Bond Line Offset				
Initial Crack Size				

a. Not applicable to the Animation Analysis engine.

These settings may all be set via the **Group Properties** button on the Material Map interface.

4.2.1 Scale Factor and Offset

The scale factor and offset settings allow the user to manipulate the stresses or strains before further analysis, for whatever purpose. For example, the offset might be used as a simple way to explore the possible effect of a residual stress.

Note

- The material scale factor and offset are applied to the *stresses*, not the material properties.
  The effect of this is (*FE Stress* x *Scale Factor*) + *Offset* = *Stress* used for fatigue calculation.
- **Scale factor**: A scale factor of >1 can be applied to evaluate the effect of stresses being higher than that which the FE and loadcase combination predicts. Or <1 if they are lower.
- **Offset**: If the effect of residual stresses, e.g. from manufacturing, not predicted by the FE analysis needs to be evaluated, this can be applied as an offset. *Tensile* residual stresses should be added as a +ve offset, e.g. +50MPa. *Compressive* residual stress, e.g. due to shot peening, would normally be a compressive stress, and specified as a -ve value.

In the case of the S-N, Spot Weld, Short Fibre Composite, and Seam Weld engines, the scale factor and offset are applied to the combined stress history (i.e., after extracting the Abs Max Principal, Critical Plane stress, etc.) before cycle counting. (See the sections describing the individual analysis engines for details of the whole process.)

(Stress for cycle counting) = (Combined stress) x (Scale Factor) + (Offset)

In the case of the E-N method, the offset, which is defined as a stress, has to be converted into strain units first, so:

(Strain for cycle counting) = (Combined strain) x (Scale Factor) + (Offset)/E

In the case of the Dang Van approach, because it is a multiaxial safety factor approach, and because it includes an elastic shakedown procedure, definition of an offset of undefined direction makes little sense. For this reason, the Dang Van engine supports only the scale factor, which is applied to the whole stress tensor before the Dang Van calculation.

How scale factors and offsets are applied (equation)

This equation shows the sequence in which the different scale factors and offsets are applied to the stress or strain history, this uses the *TimeSeriesLoadProvider* factors to as an example (other load providers may differ in detail):

$s = SF_{eng} * (OFF_{mat\_id} + SF_{mat\_id} * ((\sigma_{static} / DIV_{load}) * (SF_{load} * P_k(t) + Off_{load}))$

...where:
s = the scale factor applied
SF_{eng} = Engine scale factor
SFmat_id = Material scale factor
OFF_{mat_id} = Material offset (stress)
σ_{static} = FE results (stress)
DIV_{load} = Loadcase divider
SF_{load} = Loadcase scale factor
Off_{load} = Loadcase offset
$P_k(t)$ =Load time history

4.2.2 Surface Finish and Surface Treatment Settings

Surface finish and treatment can have a significant effect on fatigue behavior. Rough surface finishes, e.g., due to machining marks, will in general reduce the fatigue strength, whereas surface treatments are often applied to increase the fatigue strength.

In nCodeDT, surface finish and treatment effects are modeled in the S-N and E-N engines by means of a single **Surface Factor** K_{sur}. This works in a different way from the scale factor described above, with which it should not be confused. The Surface Factor is used to adjust the material curve. The application is slightly different for the S-N and E-N methods, but the basic principal is the same—the surface factor is applied to the fatigue strength of the material in the high cycle (long-life) regime, but the effect reduces in the low cycle (short-life) regime. The details of the application of the surface factor are given in the sections describing the S-N and E-N analysis engines. K_{sur} is the product of three factors, which can be defined via the material map. Each of these has default value 1.

K_{sur} = (KTreatment) x (KUser) x (KRoughness)

KTreatment

A factor used to adjust the fatigue strength to take into account surface treatment. A factor > 1 will result in an improvement in fatigue strength.

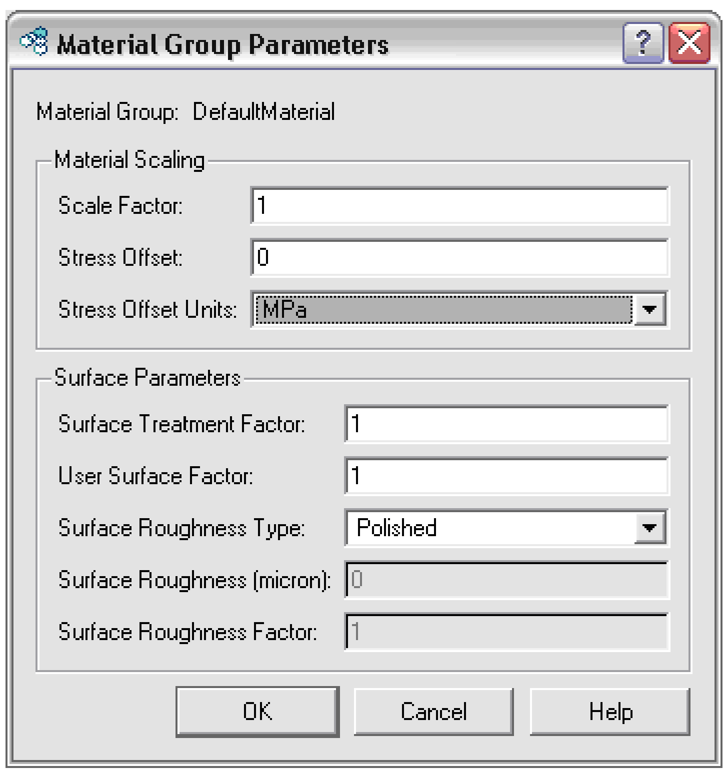
KUser

A factor used to adjust the fatigue strength for any unspecified reason. A factor > 1 will result in an improvement in fatigue strength.

KRoughness

KRoughness can be defined via the **Material Group Parameters (Group Properties...)** dialog in three different ways, depending on the setting of the Surface Roughness Type (see Figure 4-1).

Fig. 4-1 Material Group Parameters dialog



The roughness factor normally provides an indication of the fatigue strength relative to a polished surface condition, but in practice it is relative to whatever the surface condition of the material was on the specimens used to derive the material properties.

The three methods for creating a surface roughness factor (KRoughness) are:

Enter Roughness Factor

If Surface Roughness Type = Enter Roughness Factor, the Surface Roughness Factor KRoughness (K_R) may be set directly, e.g., based on the user's experience, or on experimental evidence.

Enter Surface Roughness

If Surface Roughness Type = Enter Surface Roughness, the user should enter a value for the Surface Roughness R_z in µm. This is the average surface roughness according the German standard DIN 4768. The Surface Roughness Factor K_R will then be calculated, based on the strength and type of material (for example, stronger materials are in general more sensitive to surface finish, and cast materials less so). The method for calculating K_R is based on the FKM Guideline "Analytical strength assessment of components in mechanical engineering".

If $R_z \leq 1$ µm, $K_R = 1$.

Otherwise:

$K_R = 1\text{-}a_R.\log(R_Z).\log(2R_m/R_{m,N,min})$

R_m is the UTS in MPa

$R_{m,N,min}$ and a_R are constants.

Values of the constants are a function of the type of material and the UTS. They are defined in the following table. In order for the calculation to work in nCodeDT, the nCode material type number must be defined in the material dataset, and must be one of the type numbers specified in the table.

Table 4-4 Constants for Derivation of Surface Roughness Factor K_R from Roughness R_z.

Material	Steel	GS	GGG	GT	GG	Wrought Al alloys	Cast Al alloys
nCode material type number	13,14,16-22-25,26-99	9-12,15	5-8	2-4	1	100-105	106
a_R	.22	.20	.16	.12	.06	.22	.20
$R_{m,N,min}$/MPa	400	400	400	350	100	133	133

GS= cast steel and heat treatable cast steel, for general purposes
GG= cast iron with lamellar graphite (gray cast iron)
GGG= nodular cast iron
GT= malleable cast iron

Descriptive

If one of the surface roughness descriptions is selected (Polished, Ground, Machined, etc.) the FKM method described above will be applied, using one of a number of pre-set values for the surface roughness R_Z. The values are given in the following table:

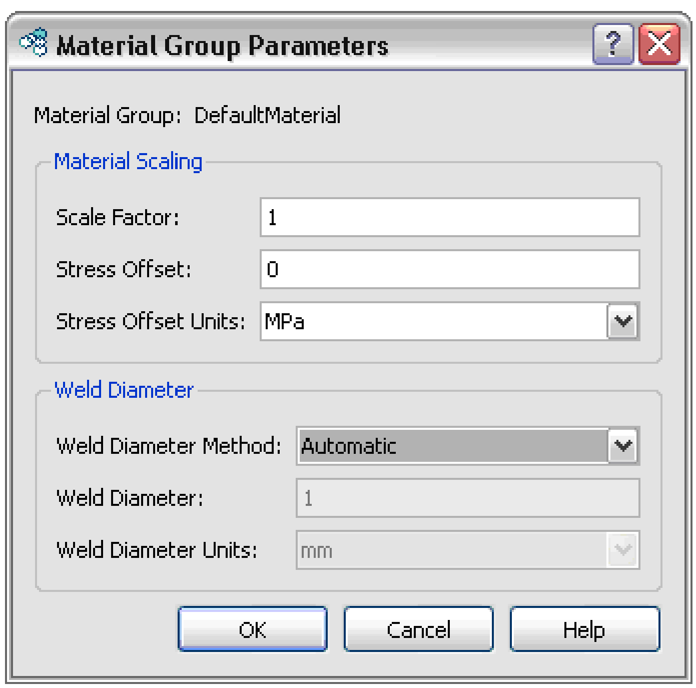
Table 4-5 Surface Roughness (R_Z) Values Corresponding to Surface Finish Descriptors

Condition	R_Z (µm)
Polished	0
Ground	12.5
Machined	100
Poor machined	200
As rolled	200
As cast	200

4.2.3 Weld Diameter

Weld diameter applies to spot weld calculations only. Once again this is defined via the **Material Group Parameters (Group Properties...)** dialog. There are two options.

Fig. 4-2 Material Group Parameters (Group Properties…) dialog



Enter Value

Enter the value of the weld diameter directly. Note that the diameter should represent the actual diameter of the weld, not the nominal diameter of the welding tool. Nominal diameters may be as much as 1 mm less than actual diameters, and using nominal diameters will result in over-conservative predictions.

Automatic

If the Weld Diameter Method is set to Automatic, nCodeDT will set the weld diameter automatically based on the sheet thicknesses. The diameter is set as a function of the thickness of the thinnest sheet joined by the spot weld. It is based on a file, `spotweld.sys`, which can be found in the `nssys` directory of the DesignLife installation.

Note The file delivered with the software is intended as an example of the format, and if the Automatic mode is to be used in practice, this file should be modified to represent actual practice.

nCodeDT does not interpolate between the data points in the `spotweld.sys` file; rather, it works as follows. The thickness of the thinnest sheet from each spot weld is compared to the table, and the value of thickness that is nearest to but less than or equal to the thickness of the sheet is identified. The corresponding diameter from the table is assigned to that spot weld.

Example `spotweld.sys` file:

```
SPOT WELD DEFINITION FILE
NUGGET_DIAMETER=BY_THICKNESS
0.3,3.5
0.8,4.0
1.2,5.0
2.0,5.5
3.0,6.0
```

Note that there may be rounding errors in the reading of sheet thicknesses from FE files, so when defining a `spotweld.sys` file, it may be a good idea to reduce the sheet thickness values by a small tolerance in order to avoid anomalous results.

4.2.4 Default Temperature

The default temperature is optional when using Multi-temperature E-N or S-N curves. If the default temperature is set, and no temperature result is available for a particular node/element, the default temperature is used.

4.2.5 ShapeFactor

This property is used by the EN and Multiaxial EN analysis engines only, and then only when the Seeger-Heuler method is invoked.

Simple notch rules such as the Neuber and Hoffmann-Seeger rules estimate the redistribution of stress and strain around a notch with limited yielding, but with no reference to the geometry of the component. In practice, particularly when a notch is shallow or the loading is high, plasticity may become more widespread, and these simple notch rules may then significantly underestimate the plastic strains.

The Seeger-Heuler method provides a variation on the Neuber or Hoffmann-Seeger methods that increases the amount of plasticity estimated when net section yielding is predicted, based on the Shape Factor. The Shape Factor for any section is defined as the ratio of the plastic limit load to the yield load. If left blank, a default value (typically 3) will be assumed.

The Seeger-Heuler method is described in more detail in the section describing the EN engine properties.

4.2.6 Adhesive Thickness

Adhesive Thickness applies to adhesive fatigue calculations only. Enter here the thickness of the adhesive.

4.2.7 Bond Line Offset

Bond Line Offset applies to adhesive fatigue calculations only. It defines the distance from the position of the beam elements at the periphery of the bonded joint and the actual position of the edge of the adhesive. See the section on the Adhesive Bond Analysis Engine for details.

4.2.8 Initial Crack Size

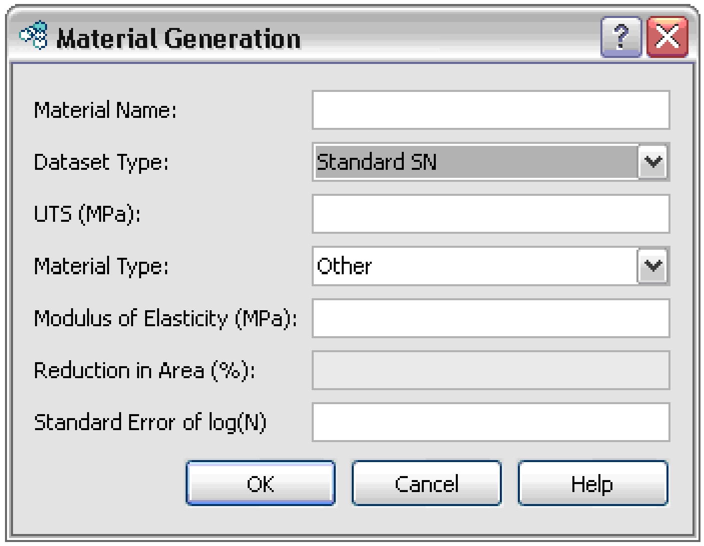
Initial Crack Size applies to adhesive fatigue calculations only. It defines the initial flaw size used to calculate the J-integral. See the section on the Adhesive Bond Analysis Engine for details.

4.2.9 Material Generation

Clicking the **Generate...** button pops up the Material Generation interface. This allows the user to enter information that will allow Standard SN, Standard EN or Dang Van fatigue material properties to be estimated, based on basic information such as the type and strength of the material. The properties thus created are added to a Memory Materials Database.

The estimation techniques used to create these material properties are approximate. They may be better than nothing when a design needs to be assessed, but should not be considered a good substitute for actual test-based properties. The meaning and application of the material properties is discussed later in the sections describing the individual analysis engines.

Fig. 4-3 Material Generation interface



Standard SN Property Generation

Standard SN properties can be generated for 4 different material classes—Ferrous, Aluminium, Titanium and Other. Apart from selecting the material class, the user has to provide the static strength (UTS) and an estimate of the Standard Error of log(N), except for the case of “Other” materials, where the Elastic Modulus is also required. The fatigue properties are then calculated as follows.

For **Ferrous** materials, the generic parameters are as follows.

Parameter Name	Calculation
MaterialType	99
UTS	User entry
YS	
E	210000 MPa
K_1C	
K_1D	
n_monotonic	
K_monotonic	
Me	0.3
Mp	0.5

Parameter Name	Calculation
Comments	
References	

Then the S-N curve is defined by the values of the stress at 1000 cycles (call this S1) and at the transition life Nc1 (call this S2). For ferrous materials:

S1 = 0.9 x UTS

S2 = 0.357 x UTS

Parameter Name	Calculation
SRI1	2 x S2/(Nc1)^{b1}
b1	(log(S2) - log(S1))/(log(Nc1) - 3)
Nc1	1E6
b2	b1/(2 + b1)
SE	User entry
RR	-1
Nfc	
M1	
M2	
M3	
M4	

Note Parameters left blank are not set.

For **Aluminum** alloys:

Parameter Name	Calculation
MaterialType	100
UTS	User entry
YS	
E	73000 MPa
K_1C	
K_1D	
n_monotonic	

K_monotonic	
Me	0.3
Mp	0.5
Comments	
References	

Then the S-N curve is defined by the values of the stress at 1000 cycles (call this S1) and at the transition life Nc1 (call this S2). For Al alloys:

S1 = 0.7 x UTS

S2 = 0.258 x UTS

Parameter Name	Calculation
SRI1	2 x S2/(Nc1)^{b1}
b1	(log(S2) - log(S1))/(log(Nc1) - 3)
Nc1	5E8
b2	b1/(2 + b1)
SE	User entry
RR	-1
Nfc	
M1	
M2	
M3	
M4	

For **Titanium** alloys:

Parameter Name	Calculation
MaterialType	300
UTS	User entry
YS	
E	110000 MPa
K_1C	
K_1D	

Parameter Name	Calculation
n_monotonic	
K_monotonic	
Me	0.3
Mp	0.5
Comments	
References	

Then the S-N curve is defined by the values of the stress at 1000 cycles (let us call this S1) and at the transition life Nc1 (let us call this S2). For Ti alloys:

S1 = 0.8 x UTS

S2 = 0.307 x UTS

Parameter Name	Calculation
SRI1	2 x S2/(Nc1)^{b1}
b1	(log(S2) - log(S1))/(log(Nc1) - 3)
Nc1	1E6
b2	b1/(2 + b1)
SE	User entry
RR	-1
Nfc	
M1	
M2	
M3	
M4	

For **Other** alloys:

Parameter Name	Calculation
MaterialType	0
UTS	User entry
YS	
E	73000 MPa

K_1C	
K_1D	
n_monotonic	
K_monotonic	
Me	0.3
Mp	0.5
Comments	
References	

Then the S-N curve is defined by the values of the stress at 1000 cycles (let us call this S1) and at the transition life Nc1 (let us call this S2).

S1 = 0.8 x UTS

S2 = 0.274 x UTS

Parameter Name	**Calculation**
SRI1	$2 \times S2/(Nc1)^{b1}$
b1	(log(S2) - log(S1))/(log(Nc1) - 3)
Nc1	1E6
b2	b1/(2 + b1)
SE	User entry
RR	-1
Nfc	
M1	
M2	
M3	
M4	

Standard EN Property Generation

Standard EN properties can be generated for 4 different material classes—Ferrous, Aluminium, Titanium and Other. Apart from selecting the material class, the user has to provide the static strength (UTS) and an estimate of the Standard Error of log(e), except for the case of “Other” materials, where the Elastic Modulus and Reduction in Area (%) are also required. The fatigue properties are then calculated as follows.

For **Ferrous** materials, the generic parameters are:

Parameter Name	Calculation
MaterialType	99
UTS	User entry
YS	
E	210000 MPa
me	0.3
mp	0.5
Comments	
References	

...and the fatigue properties are:

Parameter Name	Calculation
Sf'	1.5 x UTS
b	-0.087
c	-0.58
Ef'	0.59 x ϕ
n'	0.15
K'	1.65 x UTS
Nc	2E8
SEe	User entry
SEp	User entry
SEc	User entry
Ne	
FSN	
S	

...where:

if UTS/E ≤ 3E-3, ϕ = 1.0

if 3E-3 < UTS/E < 1E-2, ϕ = 1.375 – (125 x UTS/E) if E-2 ≤ UTS/E, $\phi = 0.1$

Note Parameters left blank are not set.

For **Aluminium** alloys, the generic parameters are:

Parameter Name	Calculation
MaterialType	100
UTS	User entry
YS	
E	73000 MPa
me	0.3
mp	0.5
Comments	
References	

...and the fatigue properties are:

Parameter Name	Calculation
Sf'	1.67 x UTS
b	-0.095
c	-0.69
Ef'	0.35
n'	0.11
K'	1.61 x UTS
Nc	2E8
SEe	User entry
SEp	
SEc	
Ne	
FSN	
S	

For **Titanium** alloys, the generic parameters are:

Parameter Name	Calculation
MaterialType	300
UTS	User entry
YS	
E	110000 MPa
me	0.3
mp	0.5
Comments	
References	

...and the fatigue properties are:

Parameter Name	Calculation
Sf'	1.67 x UTS
b	-0.095
c	-0.69
Ef'	0.35
n'	0.11
K'	1.61 x UTS
Nc	2E8
SEe	User entry
SEp	User entry
SEc	User entry
Ne	
FSN	
S	

Properties for **Other** metallic materials may also be estimated as follows.

First the generic parameters:

Parameter Name	Calculation
MaterialType	0
UTS	User entry
YS	
E	User entry
me	0.3
mp	0.5
Comments	
References	

...and the fatigue properties are:

Parameter Name	Calculation
Sf'	1.9 x UTS
b	-0.12
c	-0.6
Ef'	$0.76 \times D^{0.6}$
n'	0.2
K'	$Sf'/(Ef'^{n'})$
Nc	2E8
SEe	User entry
SEp	User entry
SEc	User entry
Ne	
FSN	
S	

where D = log (1/(1-RA/100)). RA is the Reduction in Area (%) entered by the user.

Dang Van Property Generation

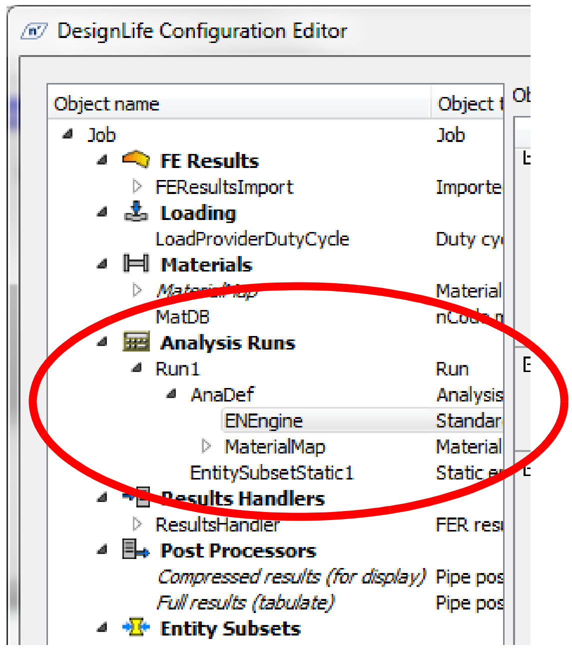
Dang Van properties can be estimated for ferrous materials only, based on the UTS as follows:

Material Parameter	Description
MaterialType	99
UTS	User Entry
E	210000
TAFE	0.28 x UTS if UTS < 1400 MPa 430 MPa if UTS ≥ 1400 MPa
HSS	0.35
Comments	
References	

5 Analysis Runs

The *Analysis Runs* section of the configuration is concerned with the details of how a fatigue analysis is to be run. It consists of one or more *Runs*, each of which includes one or more *Analysis Definition*. Each Analysis Definition in turn requires an *Analysis Engine*, and usually a *Material Map* object, to be attached.

Fig. 5-1 Analysis Run structure



Most of the following sections of this document deal with the properties of the analysis engines, which perform the actual fatigue analyses. These are:

- "Standard Assessment Analysis Engine" on page 82
- "Standard SN Analysis Engine" on page 84
- "Standard EN Analysis Engine" on page 157
- "Multiaxial EN Analysis Engine" on page 203
- "Standard Dang Van Analysis Engine" on page 227
- "Spot Weld Analysis Engine" on page 241
- "Seam Weld Analysis Engine" on page 264
- "Strain Gauge Analysis Engine" on page 314
- "Adhesive Bond Analysis Engine" on page 320
- "Creep Analysis Engine" on page 332
- "Short Fibre Composite Analysis Engine" on page 352
- "Animation Analysis Engine" on page 373
- "Safety Factor Analysis Engine" on page 375
- "Strain Energy Analysis Engine " on page 395

The theoretical background to the methods used by these engines is described in more detail later.

Before that there is one property of the Run itself that needs to be considered, and is described in the next section.

5.1 Time History Compression

The time taken to run a fatigue analysis is a function of the number of entities (nodes and elements), loadcases (for superposition events), the type of analysis (E-N is somewhat slower than S-N) and for time-series loadings, the number of data points in the time series. One way of reducing the time taken to run an analysis is to reduce the number of data points to be processed. nCodeDT provides a means of doing this that applies to time-series load providers only, using peak-valley compression. In effect, this provides the option of reducing the number of points in the time-series inputs before linear superposition.

This feature is controlled by the TimeHistoryCompression property on each analysis run, which can be set to None, PeakValley, or Limits.

Setting the property to **None** means that the entire loading time history will be processed without any compression.

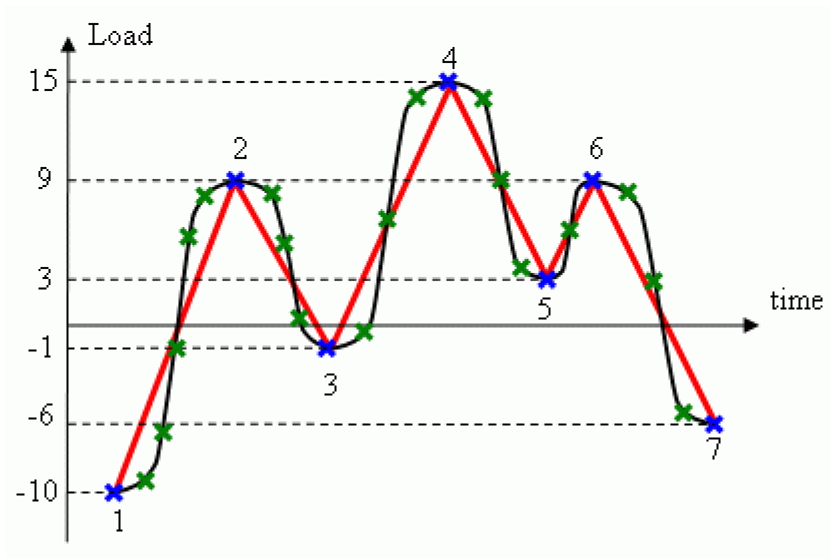
Setting the property to **PeakValley** means that peak-valley slicing will be carried out. Optionally, a rainflow gate may be used by setting the Gate and GateUnits properties. Consider the simple example in Figure 5-2. Peak-valley extraction reduces the time history to seven turning points. At the same time, the time history is rainflow cycle counted, and any cycles that have ranges less than the Gate value are discarded. The Gate value can be expressed either as an absolute value in the same units as the time history, or as a percentage of the maximum range of the time history.

For example, in the time history illustrated in Figure 5-2, we might use a gate value of 25%. First, the points labelled 1 to 7 are identified as turning points, and the rest are discarded. Rainflow counting this history (see later for a more general description of rainflow counting) identifies three cycles, formed by:

- Points 1 and 4—a range of 25 (in whatever units)
- Points 2 and 3—a range of 10
- Points 5 and 6—a range of 6

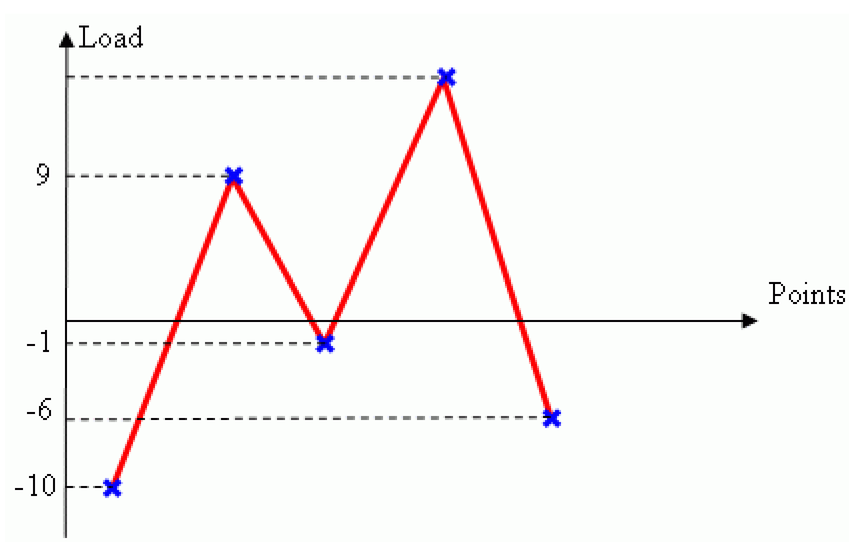
Because they are associated with a cycle that has a range of less than 25% of the maximum range, points 5 and 6 are also discarded.

Fig. 5-2 Peak-valley extraction on a single channel



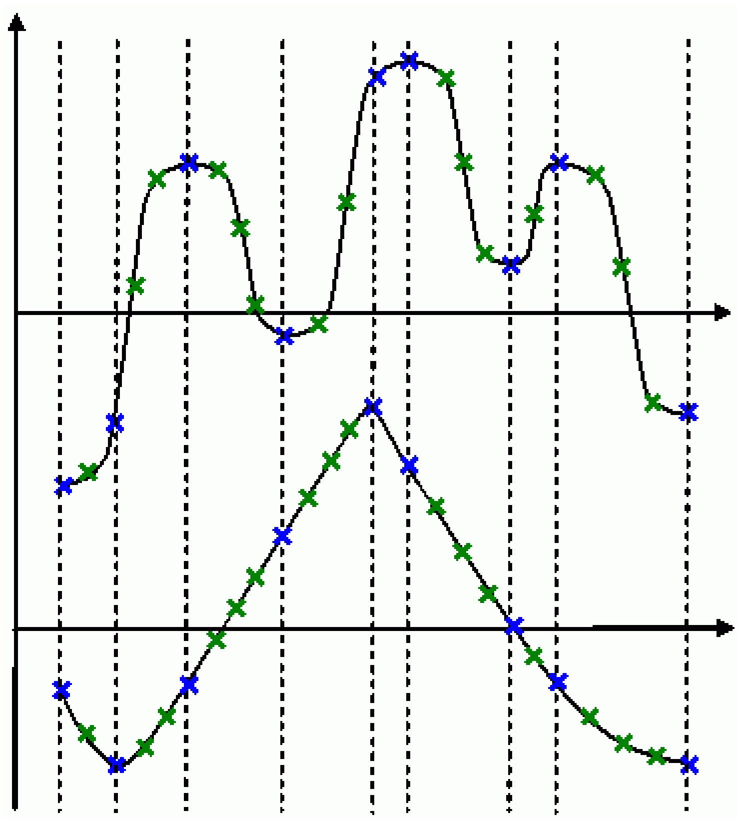
Finally, the list of turning points processed looks like this. Note that the horizontal axis is now points and not time:

Fig. 5-3 Peak-valley sequence



When multiple channels are processed, it is important to maintain the synchronization between channels. For this reason, when multiple channels are processed, after the peak-valley points to be retained in each channel have been identified, the corresponding points in time from all other channels are also retained, whether or not they are turning points. Fewer points will in general be discarded than would be the case for a single channel.

Fig. 5-4 Multi-channel peak-valley slicing



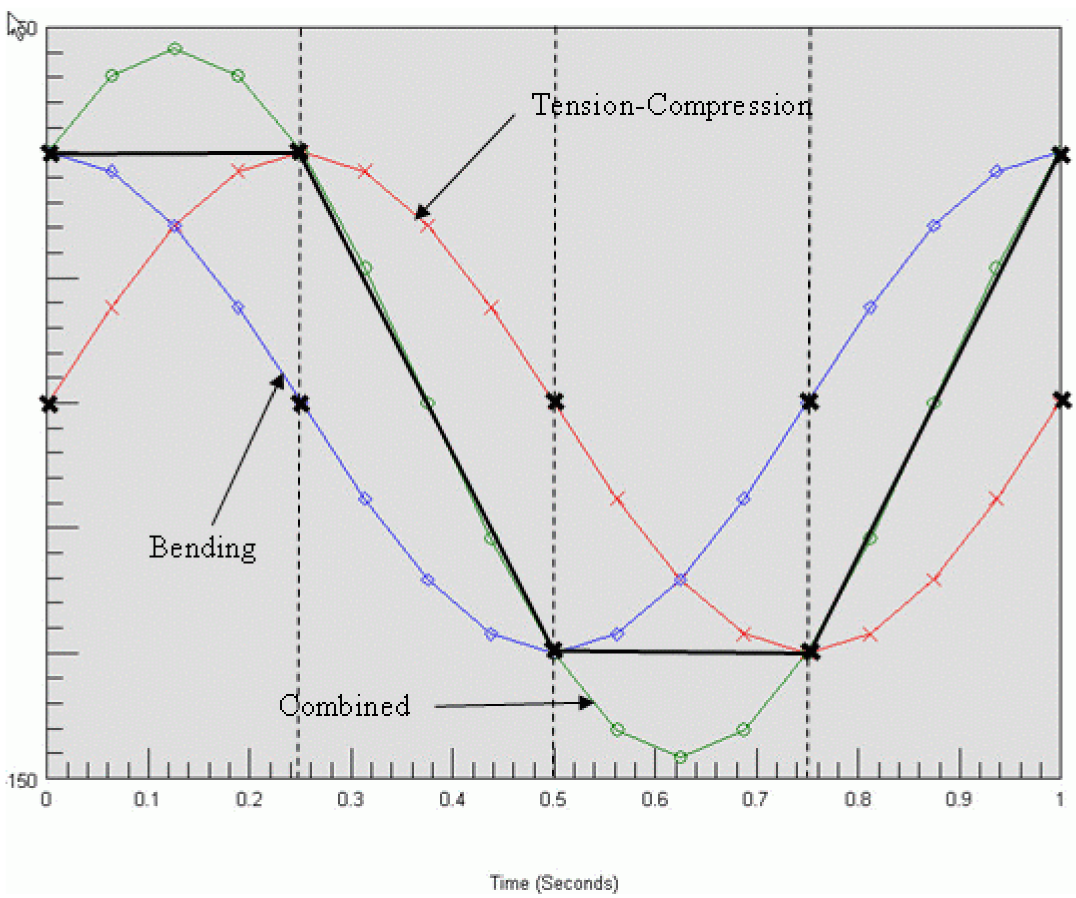
Note that when setting the value of the cycle gate, there is a trade-off between the speed of processing and accuracy. The bigger the gate value, the more potentially damaging cycles may be removed.

Note also that it is possible in certain circumstances that using this feature may eliminate the most damaging loads from the loading history. For example, consider a simple example—a cantilever beam loaded in combined tension-compression and bending, where the tension and bending have similar stress levels at the root of the beam, with sinusoidal loading and a 90-degree phase difference. The result is illustrated in Figure 5-5. This shows the contributions to the stress from bending and tension-compression, and the combined stress. The black crosses indicate the data points that are retained by peak-valley slicing, and the black line represents the combined stress that will be calculated after peak-valley slicing. In this case, the use of peak-valley slicing will result in an attenuation of the peak stress of about 40%.

This example represents a worst case scenario. If loading histories are represented by out-of-phase sine waves (which might be the case in simulating a rotating shaft, for example), it would be advisable to set the time history compression to **None**. However, in the majority of cases with multiple loadcases and variable amplitude loadings, peak-valley slicing provides a very useful way of reducing analysis times without great loss of accuracy. In these circumstances, peak-valley slicing with a cycle gate of 20% or so (for example) will normally give good acceleration without significant loss of accuracy.

If in doubt, configure a second analysis run with no time-history compression, which re-analyzes the most critical entities from the first run.

Fig. 5-5 Attenuation of combined stress from out-of-phase sine loadings due to peak-valley slicing



Setting **TimeHistoryCompression** to **Limits** gives a much greater degree of data reduction, rather like applying peak-valley extraction with a gate of 99.99%. Compression is applied, which retains the points corresponding to the maximum and minimum values of each channel (time history), including the corresponding points from all the other channels. It approximates the worst cycle, but discards many other damaging cycles. This method is really only intended to be used as a way of identifying the most highly stressed regions of a structure for more accurate analysis in a subsequent run.

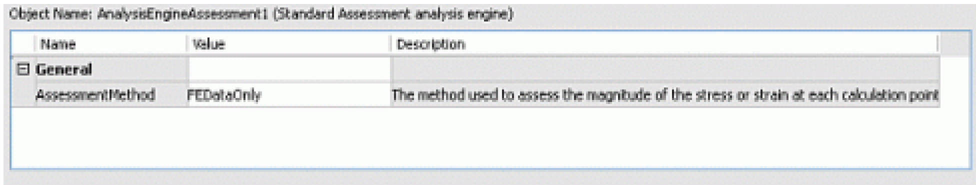
When running a duty cycle, time compression operates on each event in the duty cycle independently.

The following sections describe the details of the operation of the different analysis engines:

- "Standard Assessment Analysis Engine" on page 82
- "Standard SN Analysis Engine" on page 84
- "Standard EN Analysis Engine" on page 157
- "Multiaxial EN Analysis Engine" on page 203
- "Standard Dang Van Analysis Engine" on page 227
- "Spot Weld Analysis Engine" on page 241
- "Seam Weld Analysis Engine" on page 264
- "Strain Gauge Analysis Engine" on page 314
- "Adhesive Bond Analysis Engine" on page 320
- "Creep Analysis Engine" on page 332
- "Short Fibre Composite Analysis Engine" on page 352
- "Animation Analysis Engine" on page 373
- "Safety Factor Analysis Engine" on page 375
- "Strain Energy Analysis Engine " on page 395

5.2 Standard Assessment Analysis Engine

Fig. 5-6 Standard Assessment analysis engine properties



The purpose of the *Standard Assessment Engine* is to provide a rapid method for putting FE entities (node, element or node-on-element) into a ranking order based on stress levels. This feature can be used to reduce processing time on large models by making a multi-pass analysis in which the first run employs a Standard Assessment Engine to eliminate low-stressed elements from the analysis.

- The Standard Assessment Engine works with *Time Series* or *Constant Amplitude* load providers.
- It also works with *Duty Cycle* load providers that contain only *Time Series*, *Constant Amplitude*, and *Duty Cycle* load providers.

It can work with stresses or strains.

Its only property is *AssessmentMethod*, which can take one of two values:
FEDataOnly
SimpleRange

Both these methods are based on the von Mises stress (or strain) calculated from each loadcase. Note that in the case of 2-D stresses or strains (e.g., from thin shell elements, or surface resolved stresses from solids), the calculation of von Mises stress or strain will assume that the out-of-plane stress and shear components are zero.

If **AssessmentMethod** = **FEDataOnly,** then the assessed stress is calculated by summing the von Mises stresses from each loadcase.

$$\sigma_{Assessed} = \sum_k \sigma_{Von\,Mises,k}$$

...where $\sigma_{Von\ Mises,k}$ is the von Mises stress for loadcase k.

If **AssessmentMethod** = **SimpleRange,** then the assessed stress is calculated by summing the von Mises stresses from each loadcase, taking the range of loading, divisor, and scale factor associated with each loadcase into account. For time-series events this can be expressed:

$$\sigma_{Assessed} = \sum_k \frac{\left(Max_k - Min_k\right) \cdot ScaleFactor_k \cdot \sigma_{Von\,Mises,k}}{Divider_k}$$

...where Max_k and Min_k are the maximum and minimum values from each channel of loading history. For constant amplitude events this can be expressed:

$$\sigma_{Assessed} = \sum_k \left(MaxFactor_k - MinFactor_k\right) \cdot \sigma_{Von\,Mises,k}$$

The calculation is similar for strains, except that the output will be in strain units, and that the calculation of von Mises strain requires a value of the Poisson ratio. For the purposes of this simple ranking procedure, the Poisson ratio is assumed to be 0.3.

When the analysis involves a duty cycle, the resulting AssessedStress value will be the largest value from each of the events in the duty cycle.

5.3 Standard SN Analysis Engine

Standard SN analysis engine properties are shown below.

Fig. 5-7 Standard SN analysis engine properties

Object Name: SNEngine (Standard SN analysis engine)

Name	Value	Description
⊟ **General**		
SNMethod	Standard	The method used to calculate damage from a stress cycle
CustomSNMethod	None	Specifies the customised SN method to use
CombinationMethod	AbsMaxPrincipal	The method used to combine component stresses/strains
MeanStressCorrection	None	The method used to correct the damage calculation for mean stress
InterpolationLimit	UseMaxCurve	Multicurve material interpolation limit
MultiAxialAssessment	Standard	Whether to perform assessment of the multi-axial stress state
CertaintyOfSurvival	50	Required certainty of survival (%) on damage results
ScaleFactor	1	The scale factor to apply prior to damage calculation
OutputMaxMin	False	Whether to output max and min stresses
SmallCycleCorrection	None	Adjusts materials data in the high cycle regime.
OutputMaterialNames	False	Whether to output material names to the results
OutputDistributedSource	False	Whether to output details of the distributed process that generated each resu
OutputVibrationStats	False	Whether to output Vibration PSD parameters such as ExpectedZeroUpcrossing
⊟ **BackCalculation**		
BackCalcMode	None	Whether to perform a back-calculation or not
TargetDamage	1E-6	Target damage for back calculation
BackCalcAccuracy	5	The accuracy of the back calculation
BackCalcMaxScale	5	The max scale factor to allow in back calculation
BackCalcMinScale	0.2	The min scale factor to allow in back calculation
⊟ **DutyCycle**		
EventProcessing	Independent	How to process separate events in duty cycles
OutputEventResults	False	Whether to output results per event or not for duty cycle processing
⊟ **Advanced**		
CheckStaticFailure	Warn	The action to take on static failure
DamageFloor	0	The calculated damage value below which the damage is set to zero
MaxDamage	1E30	The maximum damage value
StaticFailureDamage	1E10	The damage value to insert on static failure
⊟ **StressGradients**		
StressGradients	Auto	Controls whether stress gradient corrections are done
StressGradientsUser		The name of the file that contains the stress gradient parameters

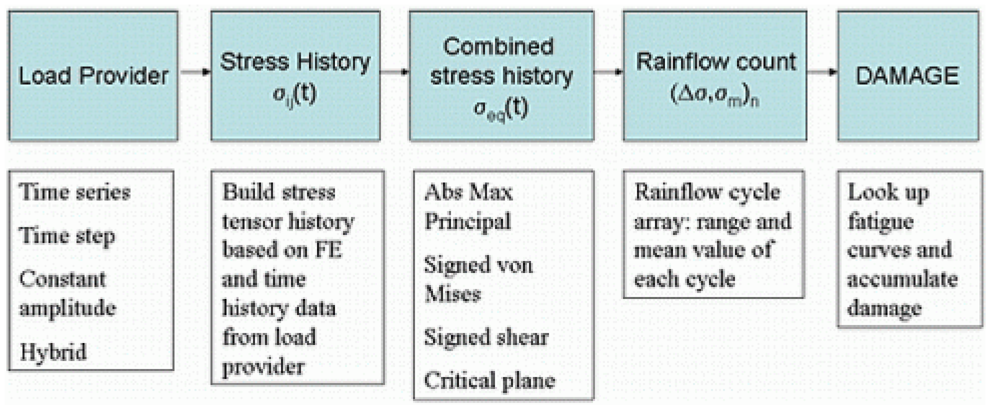
If a temperature-based method is selected, additional properties are offered:

⊟ **Temperature**		
TemperatureSelection	CycleMax	How to process temperature data
TemperatureInterpolatic	UseMaxCurve	Whether to extrapolate above the highest temperature curve

5.3.1 Summary

The basic operation of the standard SN analysis engine can be summarized in the following diagram.

Fig. 5-8 Basic S-N analysis engine steps

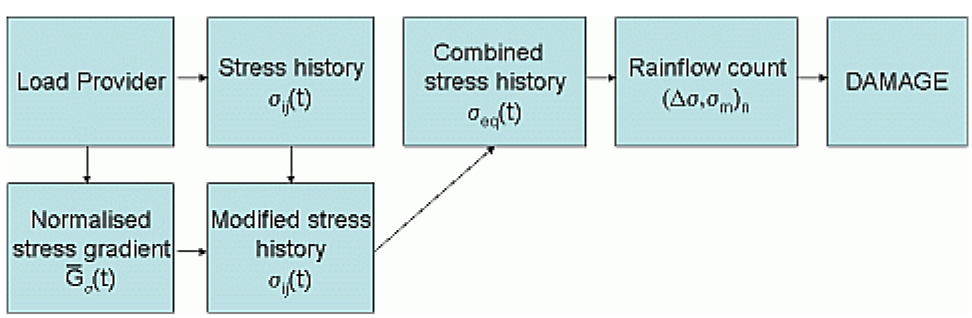


A typical S-N analysis has the following basic steps, which are executed for each analysis entity (Node, Element, Node-on-Element), shell surface and critical plane where appropriate.

1. The stress tensor history, as a function of time or data points, is assembled from the information provided in the load provider. This applies to time-series, time step, constant amplitude and hybrid load providers. See the section on load providers for details. Vibration and duty-cycle (including aero spectrum) load providers work a little differently and are addressed later.
2. From the stress tensor, extract a combined stress parameter (Abs Max Principal, Signed von Mises, Signed Shear or Critical Plane).
3. Rainflow count the combined stress history to obtain a list of rainflow cycles, each with a known range and mean value.
4. Calculate and accumulate damage from S-N curve definitions.

When the stress gradient correction is being used, the process is modified slightly as follows:

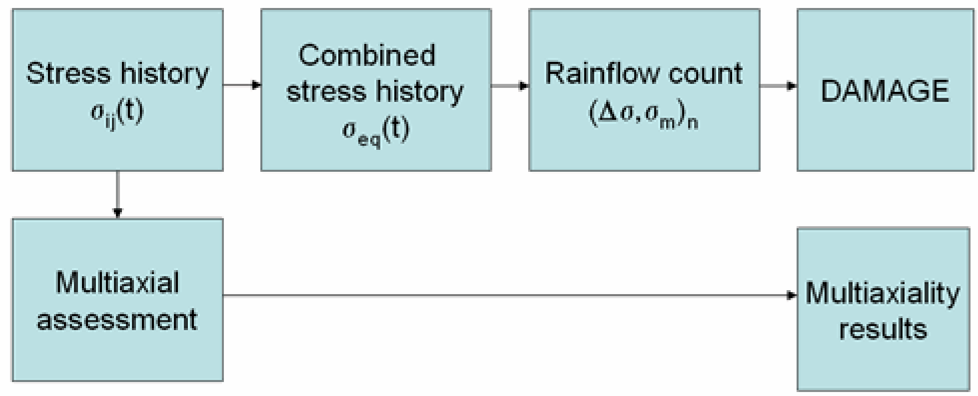
Fig. 5-9 S-N analysis process with stress gradient correction



Now the load provider also provides the information necessary to calculate a normalized von Mises stress gradient, and this information is used to modify the stress tensor history to take into account the stress gradient effect. The stress tensor history is then processed as before.

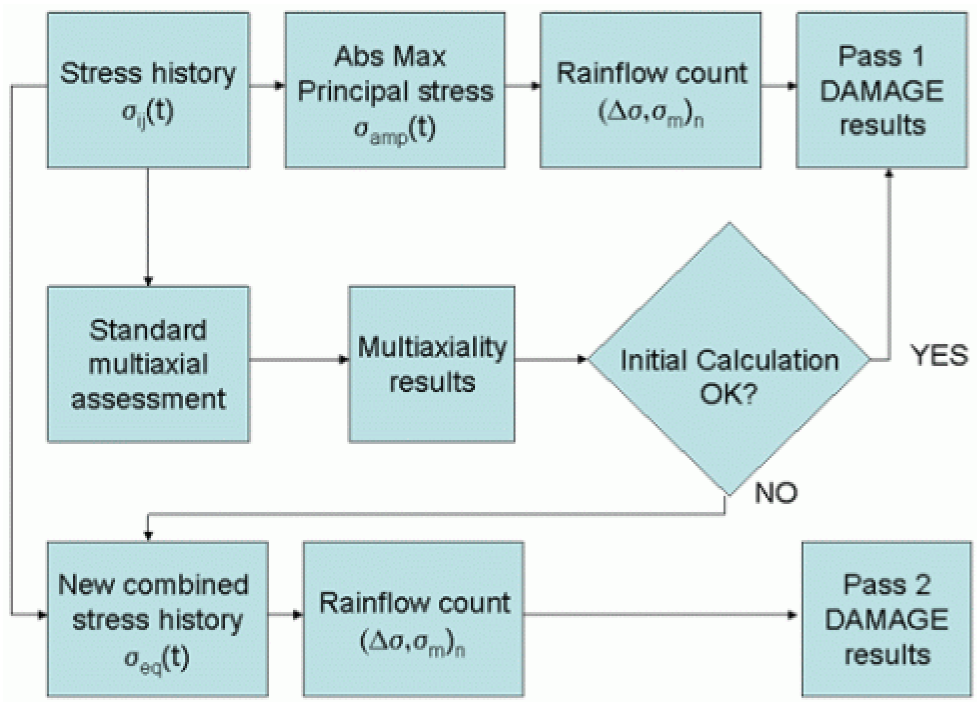
When a multiaxial assessment is requested (Simple Biaxiality or Standard; not the Auto option), the multiaxial assessment is based on the stress tensor, after stress gradient correction is applied, but the assessment doesn't affect the predicted life or damage.

Fig. 5-10 S-N analysis with Multiaxial Assessment



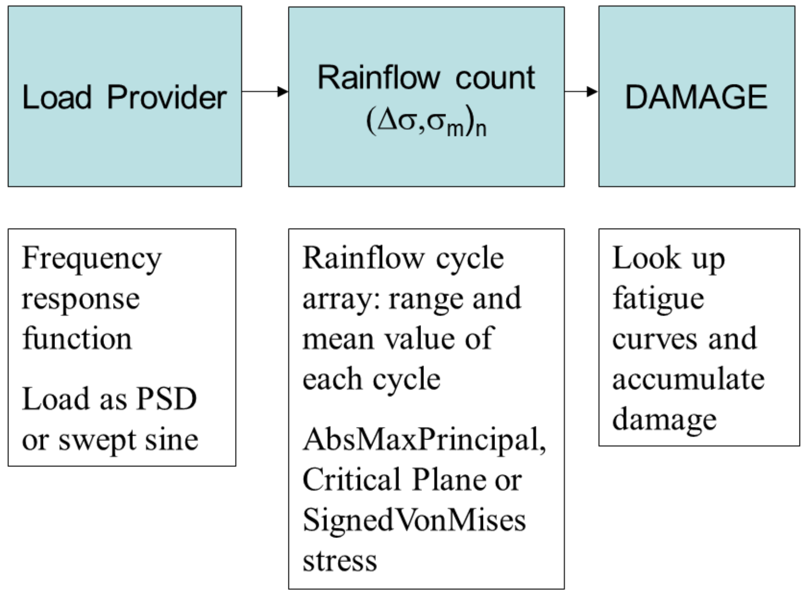
The Auto mode employs a two-pass approach. In the first pass, a calculation is made using the Absolute Maximum Principal Stress, and a Standard multiaxial assessment is also carried out. Based on the multiaxial assessment results, a second pass calculation may be made using a different combined stress option (e.g., critical plane). Where a second pass calculation is made, its results overwrite the first.

Fig. 5-11 S-N calculation process with Auto multiaxial assessment



When the load provider is a Vibration Load Provider, the process is rather different, in that a time history is not generated. The rainflow count is generated directly, based on the PSD, Multi-PSD, swept-sine, sine dwell or sine-on-random loading and the Frequency Response Function (FRF). Actually, because what is really being predicted is the probability of a rainflow cycle occurring (probable rainflow count per second), fractional cycles are possible. This is discussed in more detail in the section describing the Vibration Load Provider.

Fig. 5-12 S-N fatigue analysis process with Vibration Load Providers



The handling of duty cycles is discussed later.

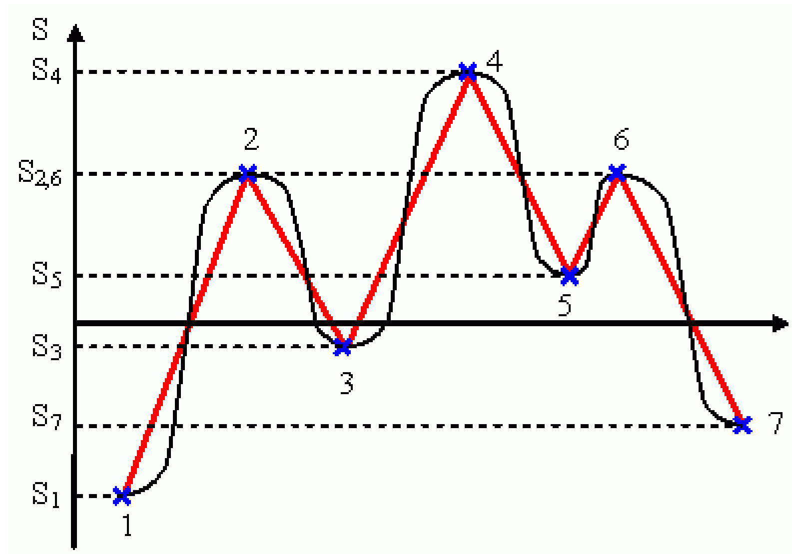
5.3.2 Note on Rainflow Cycle Counting

Rainflow cycle counting is a means of taking a variable amplitude stress or strain history and from it identifying a number of cycles with different range and mean values (or maximum and minimum). In fatigue analysis, these cycles are identified as potentially damaging events. There are a number of different possible cycle counting algorithms. The basic method used in nCodeDT is outlined here. Certain variations discussed elsewhere are possible.

The basic method has the following steps (assume we are looking at a stress history):

1. Reduce the stress time history to a series of turning points (maxima and minima):

Fig. 5-13 Reduction of stress history to turning points



2. Cycles are identified by considering four points at a time. The logic is as follows:

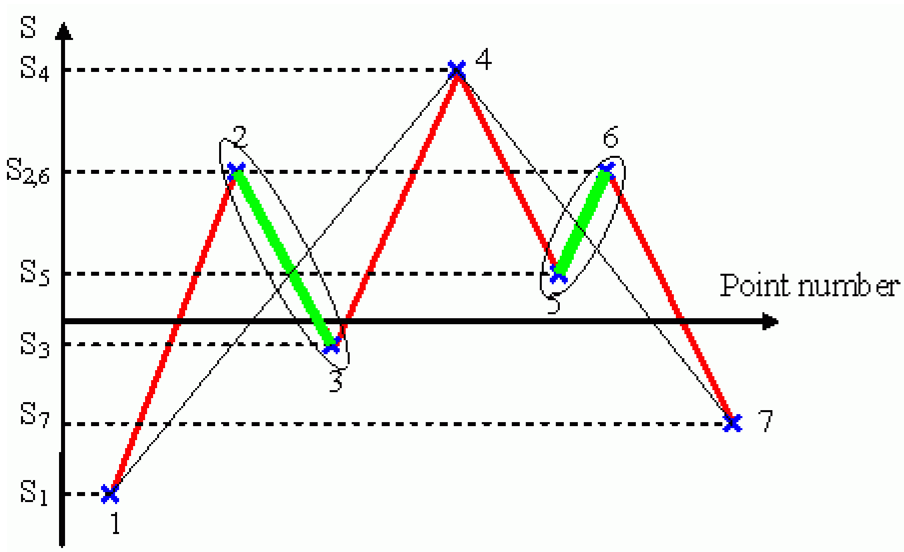
If point n is a maximum, IF $S_n \geq S_{n-2}$ AND $S_{n-1} \geq S_{n-3}$ then S_{n-2} and S_{n-1} make a cycle that can be extracted from the sequence.

If point n is a minimum, IF $S_n \leq S_{n-2}$ AND $S_{n-1} \leq S_{n-3}$ then S_{n-2} and S_{n-1} make a cycle that can be extracted from the sequence.

After this logic has been applied to the whole sequence, any remaining points are known as the residual.

This is illustrated below:

Fig. 5-14 Extraction of rainflow cycles

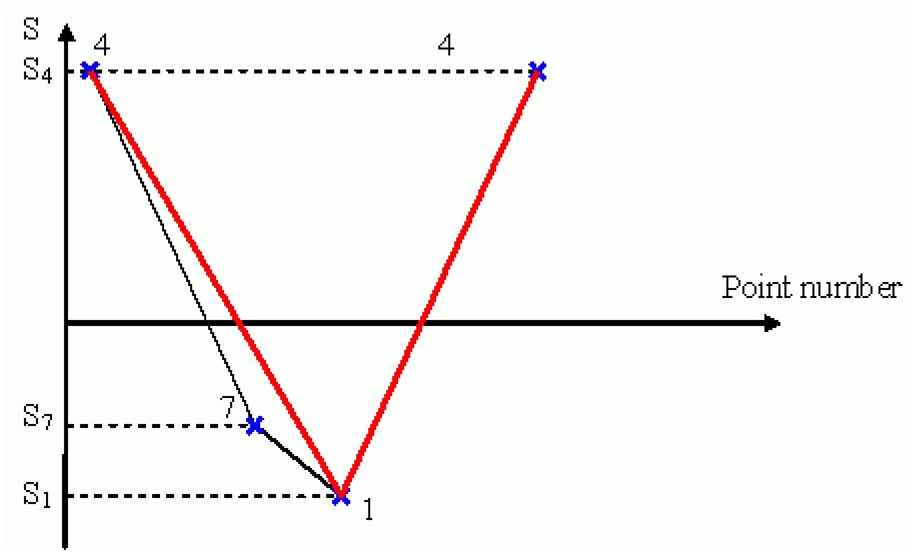


Two cycles are extracted from the sequence, represented by points 2 and 3 and by points 5 and 6. The remaining turning points, 1, 4, and 7, form the residual. (In the trivial case where the history contains only two or three points, there will be only one obvious cycle and no residual.)

3. Finally, the residual may be closed down. This is done by re-ordering the points to start from the point with the largest absolute value, and repeating this point at the end. Steps 1 and 2 are then repeated. In this

case, point 7 is no longer a turning point, so we are left with the trivial case with only three points.

Fig. 5-15 Closing the residual



The end result of cycle counting is a list of cycles. For example, the case illustrated gives:

Cycle number	Max value	Min value
1	S_2	S_3
2	S_6	S_5
3	S_4	S_1

Note that if you start cycle counting a stress time history starting from the absolute maximum value, all cycles will close; that is, there will be no residual.

The following values associated with each cycle may be used in subsequent fatigue calculations:

- Max—the maximum value of the cycle
- Min—the minimum value of the cycle
- The range of stress (Max – Min)
- The amplitude—half the range
- The mean stress (Max + Min)/2
- The load ratio or stress ratio R, Min/Max

5.3.3 Note on Damage Calculation and Accumulation

Damage for each cycle is defined as $1/N_f$ where N_f is the calculated number of cycles to failure at that stress level, taking into account all effects including mean stress, surface condition and certainty of survival.

nCodeDT basically assumes linear damage accumulation; that is, the total damage D for an event or duty cycle is the sum of the individual damage values of each cycle in the event or duty cycle, as if they happened separately. The damage reported in the results is always this value.

Miner's rule predicts that failure will occur when the accumulated damage reaches 1, i.e., the life in repeats of the event or duty cycle is 1/D. The life value reported in the fatigue analysis results from an S-N analysis engine may be modified by specifying a value of the Miner's sum that is other than 1.0, or by specifying equivalent units for the event or duty cycle. This functionality is controlled by the properties of the result postprocessor objects.

The following sections describe the properties of the S-N analysis engine in detail, with reference to the general description just given. The properties of the engine are as follows:

5.3.4 SNMethod

The SNMethod property defines the format of the S-N data to be used for the analysis, and the method for calculating damage. The options are:

- Standard
- MultiMeanCurve
- MultiRRatioCurve
- Haigh
- Bastenaire
- MultiTemperatureCurve
- Chaboche
- ChabocheTransient
- Custom

The format of the different S-N data types, the meaning of the parameters and their use is now discussed in more detail. The parameters are listed as they appear in the MXD database.

Standard

The following are the generic material parameters. The highlighted parameters are generally required for S-N fatigue analysis:

Parameter Name	Description
MaterialType	Numeric code defining the type of material. The material type is required for correct application of surface finish and stress gradient corrections.
UTS	Ultimate tensile strength. This is required for the correct definition of the upper part of the S-N curve, and to apply the static failure criterion. It is also required for surface corrections and stress gradient corrections.
YS	Yield stress. Optional (not used).
E	Modulus of elasticity. Required for S-N when the FE results are elastic strain.
K_1C	Plain strain fracture toughness
K_1D	Plain stress fracture toughness
n_monotonic	Strain hardening exponent
K_monotonic	Strain hardening coefficient
Me	Elastic Poisson ratio (defaults to 0.3 if undefined)
Mp	Plastic Poisson ratio (defaults to 0.5 if undefined)
Comments	
References	

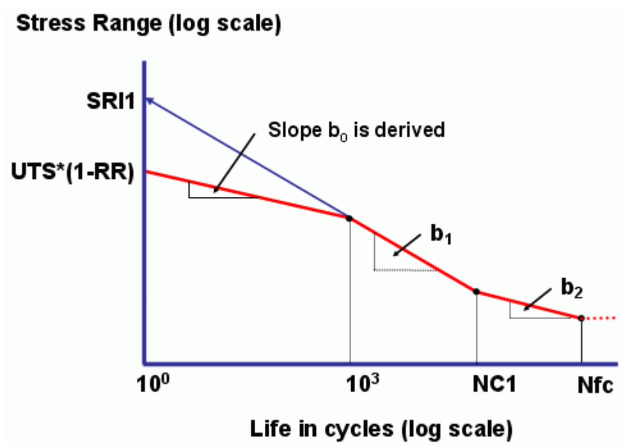
The following are the specific S-N curve parameters. The highlighted parameters are generally required. M1 to M4 are required only if the FKM mean stress correction method is being used, and even then they can often be estimated based on the material type:

Parameter Name	Description
SRI1	Stress range intercept
b1	First fatigue strength exponent
Nc1	Transition life
b2	Second fatigue strength exponent
SE	Standard error of $\log_{10}(N)$. This is used to calculate the life adjusted to a certain probability of failure/survival.
RR	R-ratio (ratio of minimum to maximum load) of the tests used to define the S-N curve
Nfc	Numerical fatigue cutoff life. Beyond this life, damage will be assumed to be zero.

Parameter Name	Description
M1	Mean stress sensitivity when R > 1 for FKM mean stress correction method
M2	Mean stress sensitivity when -∞ < R < 0 for FKM mean stress correction method
M3	Mean stress sensitivity when 0 ≤ R < 0.5 for FKM mean stress correction method
M4	Mean stress sensitivity when 0.5 ≤ R < 1 for FKM mean stress correction method

The standard S-N curve consists of 3 linear segments on a log-log plot. The central section has the formula $\Delta\sigma = SRI1(N_f)^{b1}$ where N_f is the number of cycles to failure.

Fig. 5-16 S-N curve (R-ratio = -1)



NC1 defines the point on the curve where it transitions to the second slope b2. If b2 is set to zero, this acts as a fatigue limit.

At lives of less than 1000 cycles, the slope of the curve may be modified to take into account the limitation in fatigue strength imposed by the static strength of the material, so that, at a life of 1 cycle, the maximum stress is equal to the UTS. Note that if the property CheckStaticFailure is set to False, this modification to the low cycle portion of the S-N curve is not made.

The fatigue cutoff is a numerical limit, normally set at around 1E30 cycles. It has no physical interpretation.

RR is the R-ratio or load ratio ($\sigma_{min}/\sigma_{max}$) of the tests used to determine the S-N curve. It is important when a mean stress correction is to be applied.

SE, the standard error of $\log_{10}(N)$, is used to adjust the life/damage predicted to any given probability of survival. Fatigue life always includes some scatter, and at any given level of stress range, the distribution of fatigue lives is assumed to be a log-normal distribution, that is, a Normal or Gauss distribution of the logarithm of the fatigue life.

The Gauss distribution defines the probability density function of a value x as:

$$\phi(x) = \frac{1}{\sigma\sqrt{2\pi}} e^{-\frac{(x-\mu)^2}{2\sigma^2}}$$

and the cumulative probability of x as:

$$\Phi(x) = \frac{1}{2}\left(1 + erf\left(\frac{x-\mu}{\sigma\sqrt{2}}\right)\right)$$

When applying this to fatigue calculations:

- x is replaced by $\log_{10}(N)$ where N is the fatigue life in cycles.
- μ is replaced by $\log_{10}(N_{50})$ where N_{50} is the number of cycles at which 50% of tested specimens are predicted to fail.
- σ is the standard error of $\log_{10}(N)$ associated with the S-N curve (obtained from linear regression of the original test data —SE in the material database).

In practice, if we want to make a life or damage prediction based on a particular percentage probability of survival, we use a lookup table to determine the deviation from the mean (50%) life in terms of the number of standard errors.

Table 5-1 Lookup Table for Certainty of Survival

Number of SD's from mean	% Certainty of survival
-5.00	99.99997
-4.75	99.9999
-4.50	99.9997
-4.25	99.9989
-4.00	99.997
-3.75	99.99
-3.50	99.98
-3.25	99.94
-3.00	99.87
-2.75	99.70
-2.50	99.38
-2.25	98.78
-2.00	97.72
-1.75	95.99
-1.50	93.32
-1.25	89.44
-1.00	84.13

Table 5-1 Lookup Table for Certainty of Survival

-0.75	77.34
-0.50	69.15
-0.25	59.87
0.00	50.00
0.25	40.13
0.50	30.85
0.75	22.66
1.00	15.87
1.25	10.56
1.50	6.68
1.75	4.01
2.00	2.28
2.25	1.22
2.50	0.62
2.75	0.30
3.00	0.13
3.25	0.06
3.50	0.02
3.75	0.01
4.00	0.003
4.25	0.001
4.50	0.0003
4.75	0.0001
5.00	0.00003

For example, if the standard error is 0.1, creating a calculation with 97.7% certainty of survival corresponds to -2 standard errors. Therefore:

$$\log_{10} N = \log_{10} N_{50} - 0.2$$

$$N = N_{50} \cdot 10^{-0.2}$$

MultiMeanCurve

When MultiMeanCurve is selected, the S-N analysis engine expects the S-N data in the form of a family of S-N curves representing the fatigue strength of the material at different mean stress levels. This type of data is found only in the MXD database (not supported in the nSoft MDB database) under the category nCode SN Mean Stress Curves. The data consists of a set of generic data and then a number of children representing the individual S-N curves. The generic data consists of the following parameters:

Table 5-2 Generic data - Multi Mean Curve

Parameter Name	Description
MaterialType	Numeric code defining the type of material. The material type is required for correct application of surface finish and stress gradient corrections.
UTS	Ultimate tensile strength. This is required to apply the static failure criterion.
E	Modulus of elasticity. Required for S-N when the FE results are elastic strain.
Nfc	Numerical fatigue cutoff life. Beyond this life, damage will be assumed to be zero. In cycles.
Ne	Endurance limit. This is a specified life in cycles. The main function of this is to define the point on the S-N curves where surface finish corrections are applied.
SEls	Standard error of $\log_{10}$(stress)
StressType	Type of stress used to define each cycle—Range, Amplitude, or Maximum
Comments	
References	

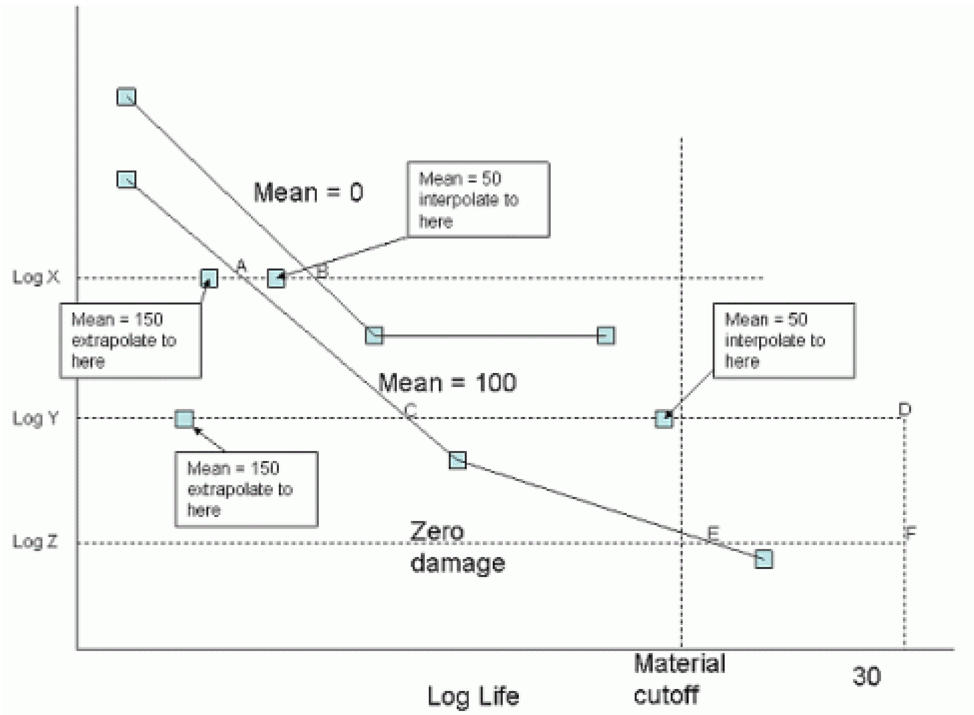
In addition, each child S-N curve (one for each mean stress value) has the following properties:

Table 5-3 Generic data - Multi Mean Curve

Parameter Name	Description
MeanStress	Value of mean stress
StressValues	Comma-separated list of stress values (descending order)
LifeValues	Corresponding comma-separated list of life values (ascending order)

The normal way to use such a curve set is to interpolate between the curves to determine the life and corresponding damage for each cycle. This process is illustrated in the following diagram:

Fig. 5-17 Interpolation scheme for MultiMeanStressCurve data



The material S-N behavior is described in this case by two S-N curves representing the material at mean stress levels of 0 and 100.

Consider a cycle with Range X. First we must identify which two curves to use for interpolation by finding the pair with mean values either side of that of our cycle.

If the cycle has a mean of 50, this lies between our two curves. In this case, we look up the log(Life) values on the two curves corresponding to LogX, and locate points A and B. If necessary, we extrapolate the curves beyond their end points. We then linearly interpolate between these two log(Life) values based on the mean stress of our cycle to determine the log(Life) of our cycle.

If the cycle has a mean less than or equal to the minimum mean of all the curves (in this case 0), the life is determined using the curve with the minimum mean value. The life corresponds to Point B.

If the cycle has a mean greater than the maximum mean in the curve set, in this case 100, either the two curves with the highest mean values can be used to determine the log(life) by extrapolation, or the curve with the highest mean may be used, i.e., in this case the life corresponds to point A. This is controlled by the InterpolationLimit property.

Consider a cycle with range Y. This corresponds to point C on the Mean = 100 curve, but it does not intersect the mean = 0 curve, or that intersection point is beyond the Fatigue Cutoff Life. In this case, the second point D used for interpolation or extrapolation is set to 30 (corresponding to life of 1E30).

Consider now a cycle with Range Z. This cycle does not intersect any of the S-N curves at a value less than the material cutoff, so the damage value will be set to zero.

Any cycle for which the resulting life is beyond the material cutoff will have its damage set to zero.

The variability in material strength is taken into account by the standard error of $\log_{10}$Stress (SEls). For example, if SEls is 0.1 and a calculation is to be made with 97.7% certainty of survival, the fatigue strength (all the stress values in each curve) would be reduced by 2 standard errors:

$$\log(\mathit{Fatigue\ Strength}) = \log(\mathit{Mean\ Fatigue\ Strength}) - 2 \times 0.1$$

$$\mathit{Fatigue\ Strength} = (\mathit{Mean\ Fatigue\ Strength}) \times 10^{-0.2}$$

Treatment of Large Stress Cycles

Where the low-cycle end of the material curve is exactly a flat line (e.g. two life values for the same stress) and a cycle stress range exceeds this flat line stress value, it is not possible to extrapolate a valid damage. If a cycle stress range exceeds this flat line section, the damage value for this cycle will correspond to the lower life value.

For example, for a flat line section:

Life = 0.01 cycles, Stress = 500 MPa

Life = 10 cycles, Stress = 500 MPa

If the stress cycle is 600 MPa, the corresponding life will be the lower life value of 0.01 cycles, and the returned damage will be 100 (1/0.01). In this way, the user can control the returned damage value by setting an appropriate minimum life value.

If this large stress range exceeds the UTS static failure check, this is correctly flagged as a static failure.

MultiRRatioCurve

The handling of MultiRRatioCurve data is very similar to that of MultiMeanCurve data.

When MultiRRatioCurve data is selected, the S-N analysis engine expects the S-N data in the form of a family of S-N curves representing the fatigue strength of the material as tested at different R-ratios. This type of data is found only in the MXD database (not supported in the nSoft MDB database) under the category nCode SN R-Ratio Curves. The data consists of a set of generic data and then a number of children representing the individual S-N curves. The generic data consists of the following parameters:

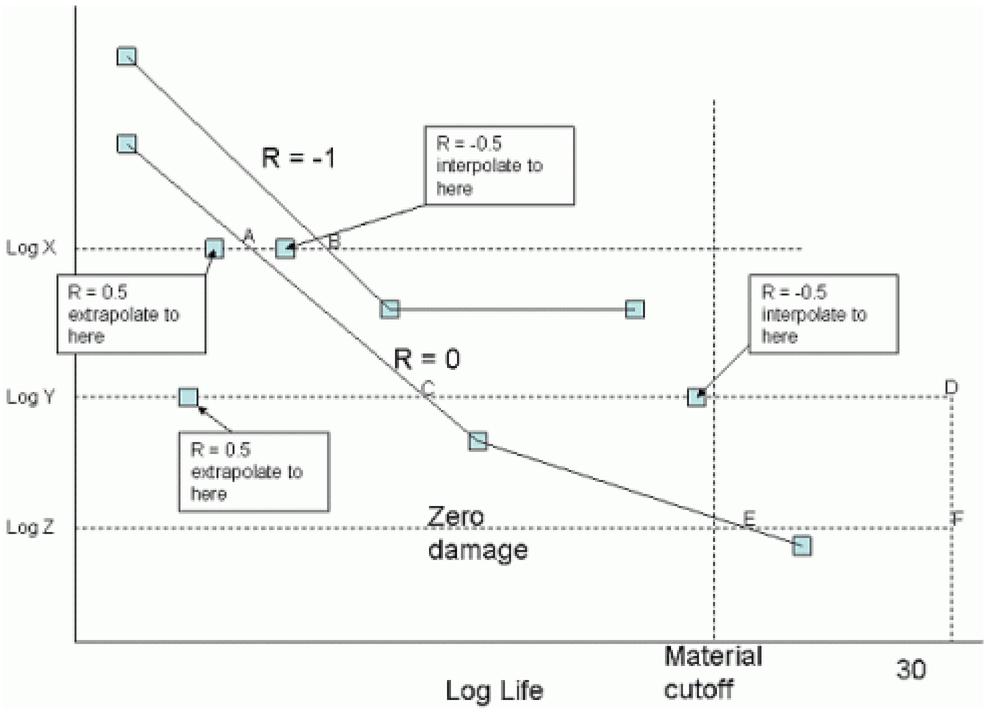
Parameter Name	Description
MaterialType	Numeric code defining the type of material. The material type is required for correct application of surface finish and stress gradient corrections.
UTS	Ultimate tensile strength. This is required to apply the static failure criterion.
E	Modulus of elasticity. Required for S-N when the FE results are elastic strain.
Nfc	Numerical fatigue cutoff life. Beyond this life, damage will be assumed to be zero. In cycles.
Ne	Endurance limit. This is a specified life in cycles. The main function of this is to define the point on the S-N curves where surface finish corrections are applied.
SEls	Standard error of $\log_{10}$(stress)
StressType	Type of stress used to define each cycle – Range, Amplitude or Maximum
Comments	
References	

In addition, each child S-N curve (one for each R-ratio value) has the following properties:

Parameter Name	Description
R-Ratio	R-ratio corresponding to curve
StressValues	Comma-separated list of stress values (descending order)
LifeValues	Corresponding comma-separated list of life values (ascending order)

The normal way to use such a curve set is to interpolate between the curves to determine the life and corresponding damage for each cycle. This process is illustrated in the following diagram:

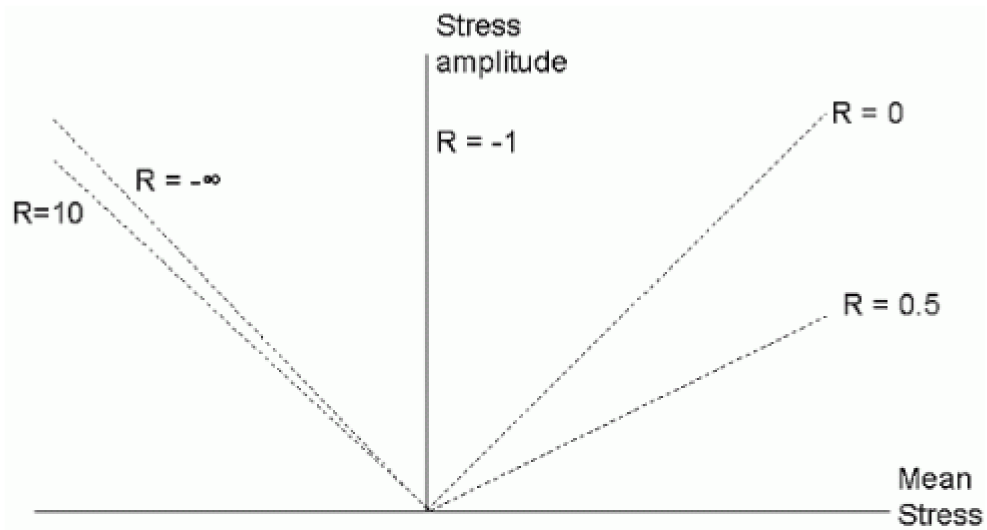
Fig. 5-18 Interpolation scheme for MultiRRatioCurve data



The material S-N behavior is described in this case by two S-N curves representing the material at R-ratios of -1 and 0.

Consider a cycle with range X. First we must identify which two curves to use for interpolation by finding the pair with mean values either side of that of our cycle. We have to be a little careful because the R-ratio has two distinct regimes bounded by the condition where maximum stress = 0, at which point R is undefined.

Fig. 5-19 Lines of constant R-ratio



If the cycle has an R-ratio of -0.5, this lies between our two curves. In this case, we look up the log(Life) values on the two curves corresponding to LogX, and locate points A and B. If necessary, we extrapolate the curves beyond their end points. We then calculate the mean stress corresponding to points A and B and linearly interpolate between these two log(Life) values based on the mean stress of our cycle to determine the log(Life) of our cycle. We don't use the R-ratio for interpolation because of the non-linear and discontinuous behavior illustrated in Figure 5-19.

Assuming that there are no curves present with R > 1 (i.e., totally in the compressive regime, which is unlikely):

- If the cycle has an R-ratio less than or equal to the minimum curve R-ratio, or which is greater than 1, the life is determined using the curve with the minimum R-ratio value.
- If the cycle has an R-ratio greater than the maximum R-ratio in the curve set, but less than 1, either the two curves with the highest R-ratio values can be used to determine the log(life) by extrapolation, or the curve with the highest R-ratio value may be used, i.e., in this case, the life would correspond to point A. This is controlled by the InterpolationLimit property.

Consider a cycle with Range Y. This corresponds to point C on the R = 0 curve in Figure 5-18, but it does not intersect the R = -1 curve, or that intersection point is beyond the Fatigue Cutoff Life. In this case, the second point D used for interpolation or extrapolation is set to 30 (corresponding to a life of 1E30).

Consider now a cycle with Range Z. This cycle does not intersect any of the S-N curves at a value less than the material cutoff, so the damage value will be set to zero.

Any cycle for which the resulting life is beyond the material cutoff will have its damage set to zero.

See "Treatment of Large Stress Cycles" on page 99.

The variability in material strength is taken into account by the standard error of $\log_{10}$Stress (SEls). For example, if SEls is 0.1 and a calculation is to be made with 97.7% certainty of survival, the fatigue strength (all the stress values in each curve) is reduced by two standard errors:

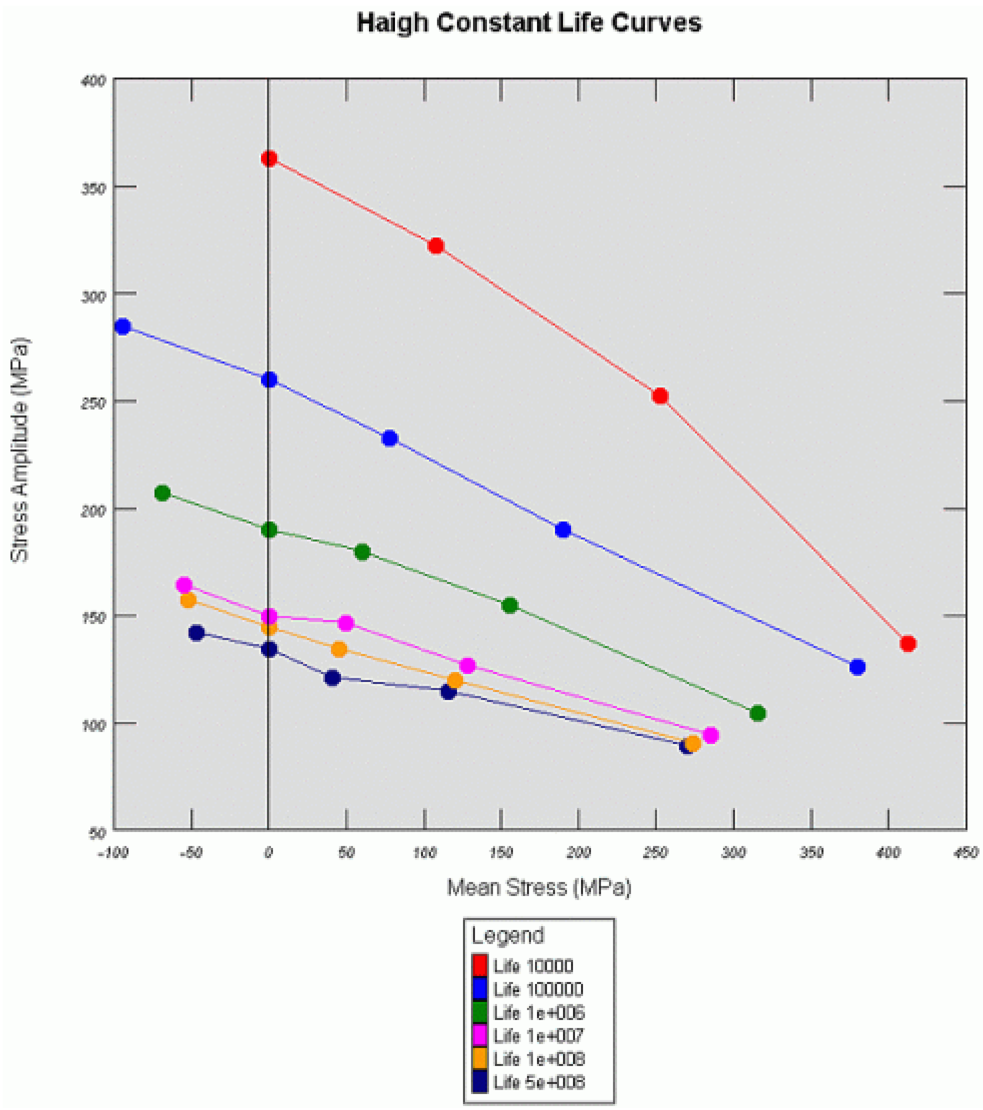
$$\log(\textit{Fatigue Strength}) = \log(\textit{Mean Fatigue Strength}) - 2 \times 0.1$$

$$\textit{Fatigue Strength} = (\textit{Mean Fatigue Strength}) \times 10^{-0.2}$$

Haigh

Another way of defining stress-life data is in terms of a constant life or Haigh diagram. The material behavior is described by a number of curves on a plot of stress amplitude vs. mean stress, each of which represents test failures with the same fatigue life.

Fig. 5-20 Haigh constant life curves diagram



Once again, the dataset consists of a set of generic parameters, and a number of children containing the constant life curves.

Parameter Name	Description
MaterialType	Numeric code defining the type of material. The material type is required for correct application of surface finish and stress gradient corrections.

Parameter Name	Description
UTS	Ultimate tensile strength. This is required to apply the static failure criterion.
E	Modulus of elasticity. Required for S-N when the FE results are elastic strain.
Nfc	Numerical fatigue cutoff life. Beyond this life, damage will be assumed to be zero. In cycles.
Ne	Endurance limit. This is a specified life in cycles. The main function of this is to define the point on the S-N curves where surface finish corrections are applied.
SEls	Standard error of $\log_{10}$(stress)
StressType	Type of stress used to define each cycle—Range, Amplitude, or Maximum
Comments	
References	

Each child constant life curve (there will be one of these for each mean stress value) has the following properties:

Parameter Name	Description
Life	Life in cycles for constant life curve
MeanStressValues	Comma-separated list of mean stress values
StressValues	Corresponding comma-separated list of stress values (range, amplitude or maximum)

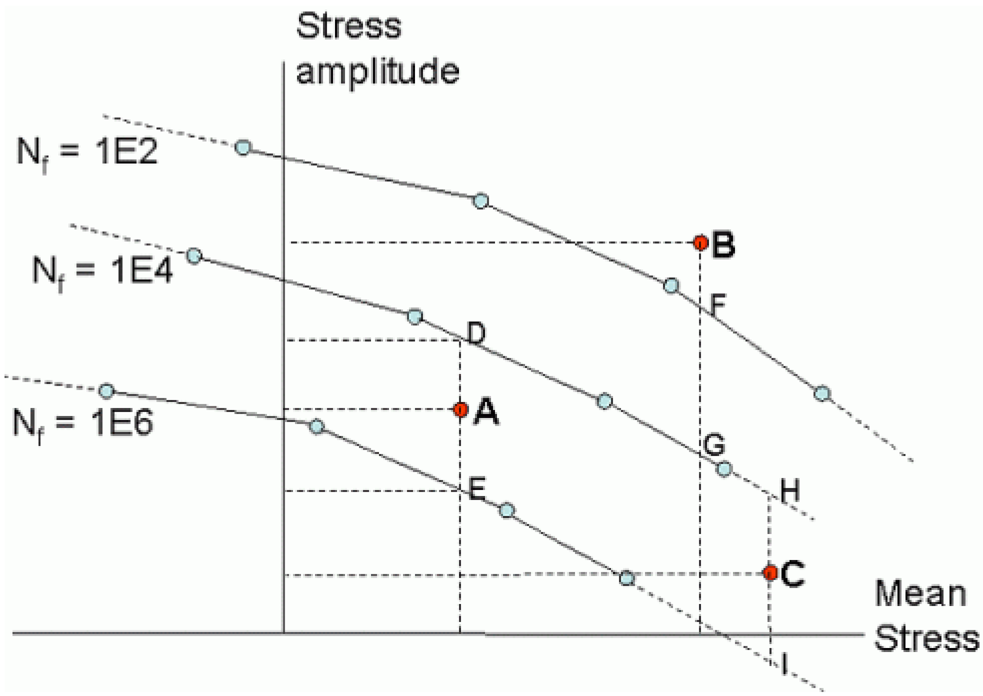
The life for a given cycle is determined by interpolation between the curves of the Haigh diagram. For example, in Figure 5-21 below, consider the cycle represented by point A. This lies between the curves for N_f = 1E4 and N_f = 1E6. Based on the mean stress of this cycle, we look up the stress amplitudes corresponding to points D and E. The life for this cycle is determined by linear interpolation of the log(Life) between these points. For example, if A lies halfway between D and E, the predicted life for that cycle will be 1E5 (damage 1E-5).

The stress amplitude points for interpolation may be determined by extrapolation if necessary, as for point C.

For a cycle that lies above or below all the curves such as that represented by point B, the life and damage is determined by extrapolation (subject to limitations imposed by the MaxDamage and Fatigue cutoff properties).

Results based on cycles that use extrapolation should be treated with some caution.

Fig. 5-21 Fatigue life by interpolation using the Haigh diagram

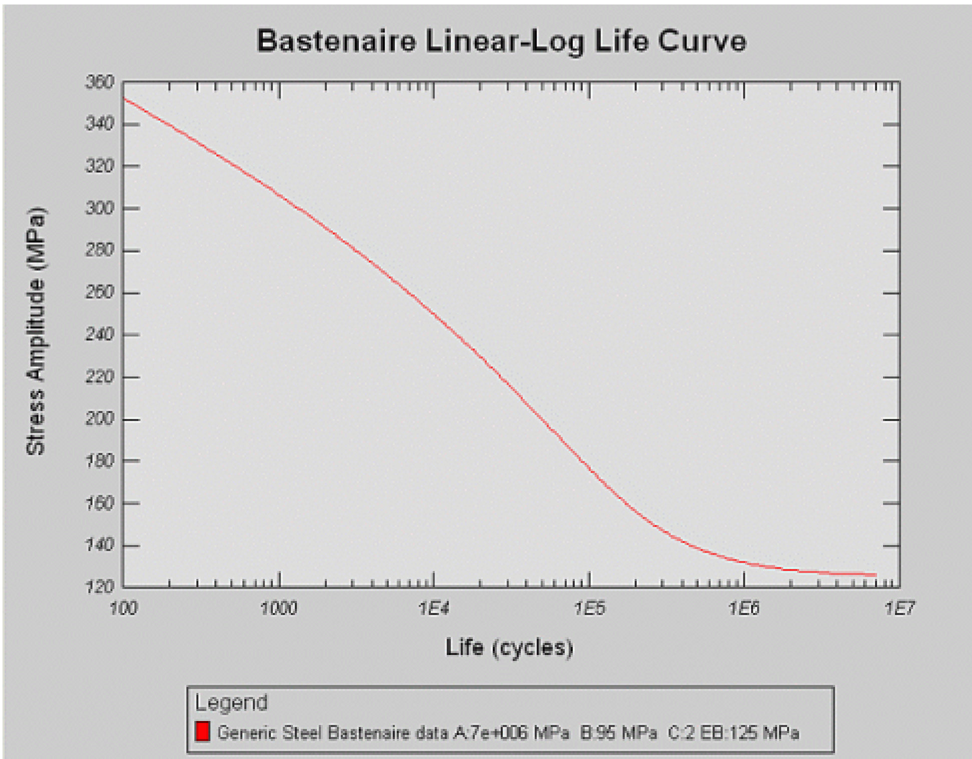


Note that the InterpolationLimit property is not used in the Haigh method.

The stress amplitude values are adjusted to consider the certainty of survival in the same way as for the multi-curve formulation. That is, the log of the fatigue strength is reduced by an appropriate number of standard errors based on the lookup table.

Bastenaire

Fig. 5-22 Bastenaire S-N curve



The material dataset for the Bastenaire method is as follows:

Parameter Name	Description
MaterialType	Numeric code defining the type of material. The material type is required for correct application of surface finish and stress gradient corrections.
UTS	Ultimate tensile strength. This is required to apply the static failure criterion.
E	Modulus of elasticity. Required for S-N when the FE results are elastic strain.
A	Bastenaire coefficient—a parameter positioning the curve along the life axis
B	Scale factor parameter
C	Bastenaire exponent
EB	Bastenaire fatigue limit
RR	R-ratio of test
StressType	Amplitude, Range or Maximum

Parameter Name	Description
Sd	Bastenaire scatter factor
Comments	
References	

Based on the analysis of thousands of tests carried out on different steels, Bastenaire proposed in 1974 a general formulation of the Stress Life curve:

$$N = \frac{A}{S - E_B} \exp\left[-\left(\frac{S - E_B}{B} \right)^C \right]$$

The four parameters A, B, C and E_B are derived from material raw data. If C > 1, the SN curve has an inflexion point. If C = 0, the model simplifies to the Stromeyer formulation N = A/(S-E) (with a factor of e).

The Bastenaire formula aims to correctly describe the whole endurance domain and the parameter calculation takes into account all tested specimens, including the run-outs.

Bastenaire curves can also be modified to calculate lives at any certainty of survival, using a scatter factor and the normal distribution law (the assumption is made that the stresses are normally distributed for a specified life):

$$N_{P\%} = \frac{A}{(S \pm m * sd - E_B)} \exp\left[-\left(\frac{S \pm m * sd - E_B}{B} \right)^C \right]$$

- sd: scatter factor, estimation of the standard deviation on stress
- m: number of standard deviations from the mean value, defines the required probability and is given by the normal law tables .

Below are given example formulas for 84%, 50% and 16% certainty of survival:

$$N_{84\%} = \frac{A}{(S + sd - E_B)} \exp\left[-\left(\frac{S + sd - E_B}{B} \right)^C \right]$$

$$N_{50\%} = \frac{A}{(S - E_B)} \exp\left[-\left(\frac{S - E_B}{B} \right)^C \right]$$

$$N_{16\%} = \frac{A}{(S - sd - E_B)} \exp\left[-\left(\frac{S - sd - E_B}{B} \right)^C \right]$$

As usual, linear damage accumulation applies, and damage per cycle is assumed to be 1/N.

MultiTemperatureCurve

MultiTemperatureCurve datasets provide a set of S-N curves at different temperatures. The fatigue curves for intervening temperatures are determined by interpolation. Each dataset consists of a generic parent and a number of children containing the temperature sensitive data. The generic data is as follows:

Parameter Name	Description
MaterialType	Numeric code defining the type of material. The material type is required for correct application of surface finish and stress gradient corrections.
StressType	Type of stress used to define each cycle—Range, Amplitude, or Maximum
R-Ratio	R-Ratio, this applies to all curves for this material
Comments	
References	

Note For older material data sets which do not contain R-Ratio a value of -1 is assumed.

There can be a number of child datasets at different temperatures, including parameters that may be temperature sensitive as well as the actual S-N values. The additional parameters in each are as follows:

Parameter Name	Description
Temperature	Temperature
UTS	Ultimate tensile strength. This is required to apply the static failure criterion.
E	Modulus of elasticity. Required for S-N when the FE results are elastic strain.
me	Elastic Poisson ratio (default 0.3)
mp	Plastic Poisson ratio (default 0.5)
Nfc	Numerical fatigue cutoff life. Beyond this life, damage will be assumed to be zero. In cycles.
Ne	Endurance limit. This is a specified life in cycles. The main function of this is to define the point on the S-N curves where surface finish corrections are applied.
SEls	Standard Error of $\log_{10}$(stress)
StressValues	Comma-separated list of stress values (descending order)
LifeValues	Corresponding comma-separated list of life values (ascending order)

Interpolation between the curves uses the same method as for MultiMeanCurve data. We determine the life/damage for each cycle by linear interpolation of log(life) based on temperature. When the temperature is below the lowest temperature for which a material is available, the nearest curve is used, that is there is no extrapolation. When the temperature is above the highest temperature, the behavior is controlled by the SNEngine property "TemperatureInterpolationLimit". If set to UseMaxCurve, the nearest curve is used. If set to Extrapolate, extrapolation is performed.

The temperature to be used for the interpolation can be constant for the whole time history, or can vary cycle-by-cycle. Constant temperatures can be obtained from a temperature load provider (as part of a hybrid provider with stress loading) or, if that is not available, by using the DefaultTemperature property of the material map groups. From the SN engine, the user can select constant temperature options OverallMax, OverallMin and OverallMedian, which will take the max, min and median of the temperatures provided on a node/element by the hybrid load provider.

Varying temperatures can also be provided by the hybrid load provider, where they are synchronized in time with the stresses. As the stresses are cycle counted, the temperature within that cycle can be found and this temperature is used for the cycle. Either the maximum temperature over the cycle, or the minimum temperature, can be used. This is covered in "Extracting Temperatures Cycle by Cycle" on page 153.

When duty cycles are analyzed using multi-temperature curves where the events have constant temperatures, Independent mode can use a different constant temperature on each event.

However, for combined full and combined fast methods, only one temperature can be used for the whole duty cycle. The temperature used will depend on the TemperatureSelection property, and can be the maximum temperature across all events, the minimum temperature or the median of the maximum and minimum (i.e., (max+min)/2).

If the temperature selection is CycleMax or CycleMin, cycles that close within an event are analyzed according to the (possibly) varying temperature on that event. Cycles that cross events are analyzed using either the maximum temperature across all events or the minimum temperature across all events, depending on whether CycleMax or CycleMin is chosen.

Chaboche

The Chaboche option allows for the use of a set of Chaboche temperature curves to calculate life for given stress cycles together with a temperature. The temperature is provided either by the hybrid load provider (see "Hybrid Load Provider" on page 33) or by defining the temperature for each group on the material map. See "Default Temperature" on page 64.

If the hybrid load provider is used to supply the temperature, the SN engine property TemperatureSelection can be used to determine whether a constant temperature is used from the data provided by the hybrid load provider (max, min or median of the supplied temperatures) or whether the temperature varies cycle by cycle (using either the maximum temperature over the cycle or the minimum temperature over the cycle). The TemperatureSelection option “AllAvailable” is not supported by this method.

The Chaboche equation for Stress-Life is as follows:

$$N_f = \frac{\sigma_u - \sigma_{max}}{a\big((\sigma_{max} - \bar{\sigma}) - \sigma_{lo}(1 - b\bar{\sigma})\big)} \cdot \left[\frac{\sigma_{max} - \bar{\sigma}}{C_0(1 - b\bar{\sigma})}\right]^{-\beta}$$

...where N_f is the number of cycles to failure, $\bar{\sigma}$ is the mean stress of the cycle, σ_{max} is the maximum stress in the cycle, and the other variables are materials properties defined in the Chaboche material property set:

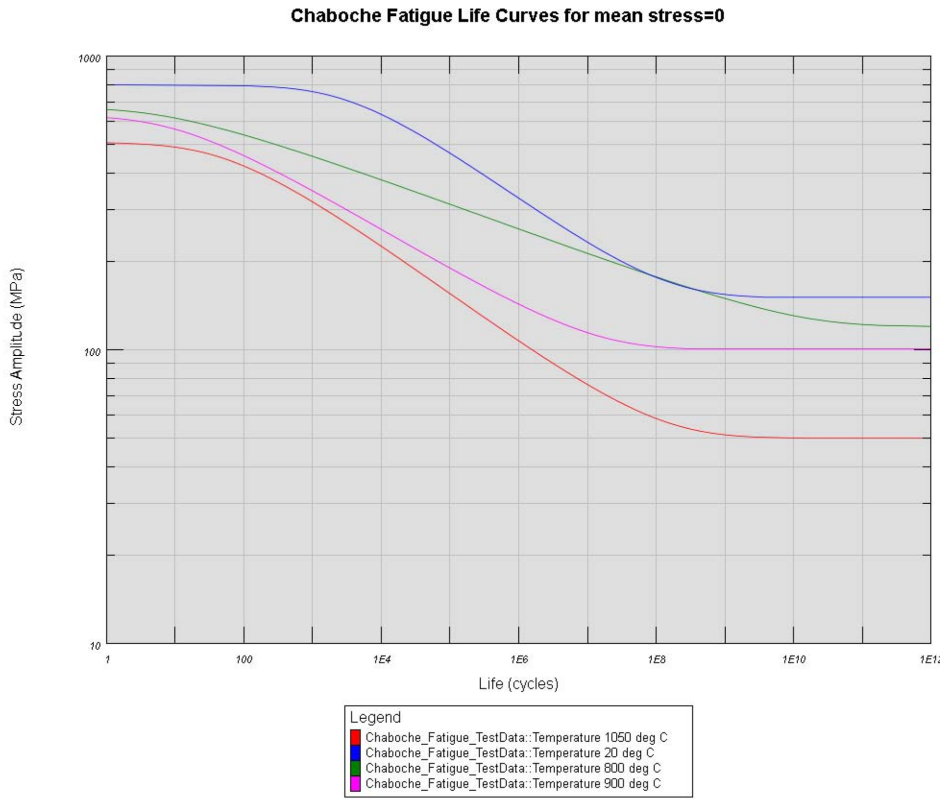
Parameter Name	**Description**
Temperature	Temperature for this curve
Sigma_u	The upper limit stress
Sigma_lo	The lower limit stress
C0	The Chaboche equation coefficient
Beta	The Chaboche equation exponent
b	Mean stress correction factor
a	Non-linear damage sensitivity

Each Chaboche Curve has a unique temperature and these are grouped together as children of a parent data set with the following properties:

Parameter Name	**Description**
MaterialType	Numeric code defining the type of material
MaximumTemperature	The maximum allowable temperature
Comments	
References	

A typical set of Chaboche curves looks like this:

Fig. 5-23 Chaboche curves for mean stresses



Because the Chaboche equation incorporates its own mean stress correction, the curves vary cycle-by-cycle according to the value of $\bar{\sigma}$.

If the temperature for a given cycle is not one of the temperatures on the curve (so that the curve data cannot be used), the following rules apply:

1. If the temperature is below the lowest curve temperature (in the plot above, this is 20C), then that curve is used; there is no extrapolation.
2. If the temperature is in between two curves, interpolation of properties is performed according to the scheme outlined below.
3. If the temperature is above the highest curve temperature, then the property *TemperatureInterpolationLimit* determines whether the properties are extrapolated or whether the Highest Curve is used. Using extrapolation is more conservative.
4. If the temperature is above the maximum allowed, the damage is set to the value specified by the *StaticFailureDamage* property.

For property interpolation, a log-log interpolation is used on all properties except the non-linear damage sensitivity property "a", which is interpolated linearly. To avoid negative temperatures in log equations, the temperatures are converted to Kelvin for the interpolation.

The equation forces asymptotes at *sigma_u* and *sigma_lo*.

- Any cycle amplitude (after any mean stress correction) that is greater than or equal to *sigma_u* will cause static failure.
- Any cycle amplitude that is less than or equal to *sigma_lo* will cause zero damage.

To use the Chaboche mean stress correction, the *MeanStressCorrection* property must be set to "*Chaboche*". This allows other mean stress corrections to be used including None, Goodman, Gerber, GoodmanTensionOnly and GerberTension-Only. FKM and Interpolate mean stress options are not allowed.

When None, Goodman or Gerber are used, the effect of the material property "b" is removed by forcing the mean stress of the cycle to be 0 in the equation.

For Goodman and Gerber, sigma_u is used as the UTS.

SN Engine properties InterpolationLimit, SmallCycleCorrection and CertaintyOf-Survival are not supported in the Chaboche method. No surface finish corrections are performed when using the Chaboche method.

When duty cycles are analyzed using Chaboche where the events have constant temperatures, Independent mode can use a different constant temperature on each event.

However for combined full and combined fast methods, only one temperature can be used for the whole duty cycle. The temperature used will depend on the TemperatureSelection property, and can be the maximum temperature across all events, the minimum temperature or the median of the maximum and minimum (i.e., (max+min)/2).

If the temperature selection is CycleMax or CycleMin, cycles that close within an event are analyzed according to the, possibly, varying temperature on that event. Cycles that cross events are analyzed using either the maximum temperature across all events or the minimum temperature across all events, depending on whether CycleMax or CycleMin is chosen.

See "Extracting Temperatures Cycle by Cycle" on page 153 for details on how temperatures are extracted cycle-by-cycle.

Chaboche Transient

The Chaboche Transient option allows for the use of a Chaboche temperature analysis to calculate life for given stress cycles together with temperature. The temperature is provided either by the hybrid load provider (see "Hybrid Load Provider" on page 33) or by defining the temperature for each group on the material map (see "Default Temperature" on page 64).

If the hybrid load provider is used to supply the temperature, the SN engine property TemperatureSelection can be used to determine whether a constant temperature is used from the data provided by the hybrid load provider (max, min or median of the supplied temperatures) or whether the whole temperature time series is used ("AllAvailable" option).

The TemperatureSelection options “CycleMax” and “CycleMin” are not supported by this method.

The Chaboche equation for Stress-Life is as follows:

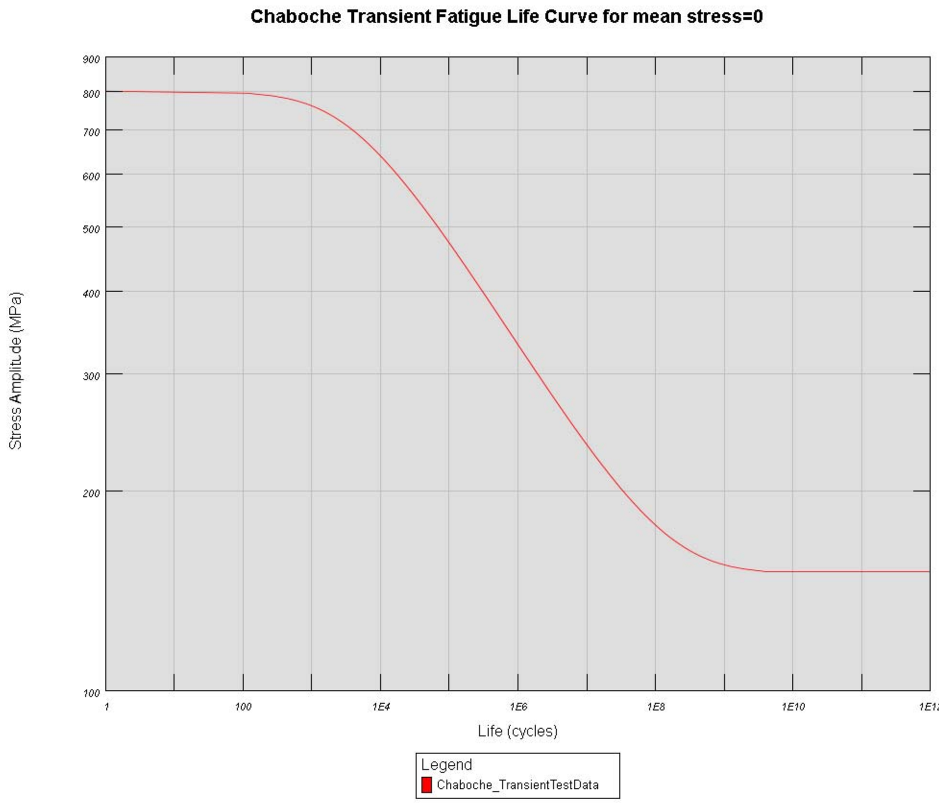
$$N_f = \frac{\sigma_u - \sigma_{max}}{a\left((\sigma_{max} - \bar{\sigma}) - \sigma_{lo}(1 - b\bar{\sigma})\right)} \cdot \left[\frac{\sigma_{max} - \bar{\sigma}}{C_0(1 - b\bar{\sigma})}\right]^{-\beta}$$

...where N_f is the number of cycles to failure, $\bar{\sigma}$ is the mean stress of the cycle, σ_{max} is the maximum stress in the cycle, and the other variables are materials properties defined in the Chaboche material property set:

Parameter Name	**Description**
Material Type	Integer value determining material type
Maximum Temperature	Maximum allowable temperature
Sigma_u	The upper limit stress
Sigma_lo	The lower limit stress
C0	The Chaboche equation co-efficient
Beta	The Chaboche equation exponent
b	Mean stress correction factor
a	Non-linear damage sensitivity
NormalisationTemperatures	Temperature values for the normalization table
NormalisationParameters	Parameter values for the normalization table
Comments	
References	

A typical Chaboche Transient curve looks like this:

Fig. 5-24 Chaboche transient curve for mean stresses



Unlike the multi-curve Chaboche method, there is only one curve in the Transient method, as the results from all temperatures (Figure 5-25) have been collapsed onto a single curve (Figure 5-26).

Fig. 5-25 Collapsing the Chaboche curves (a)

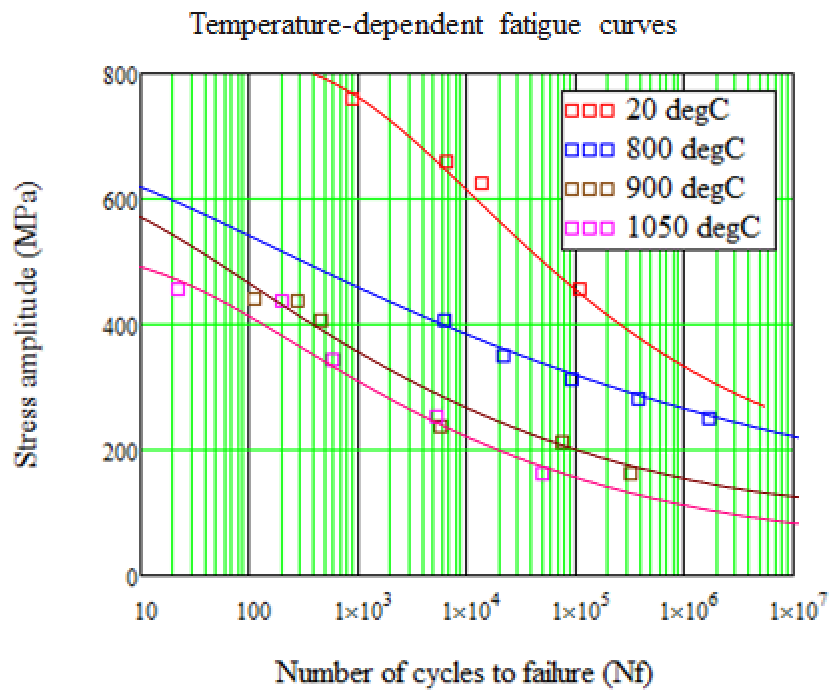
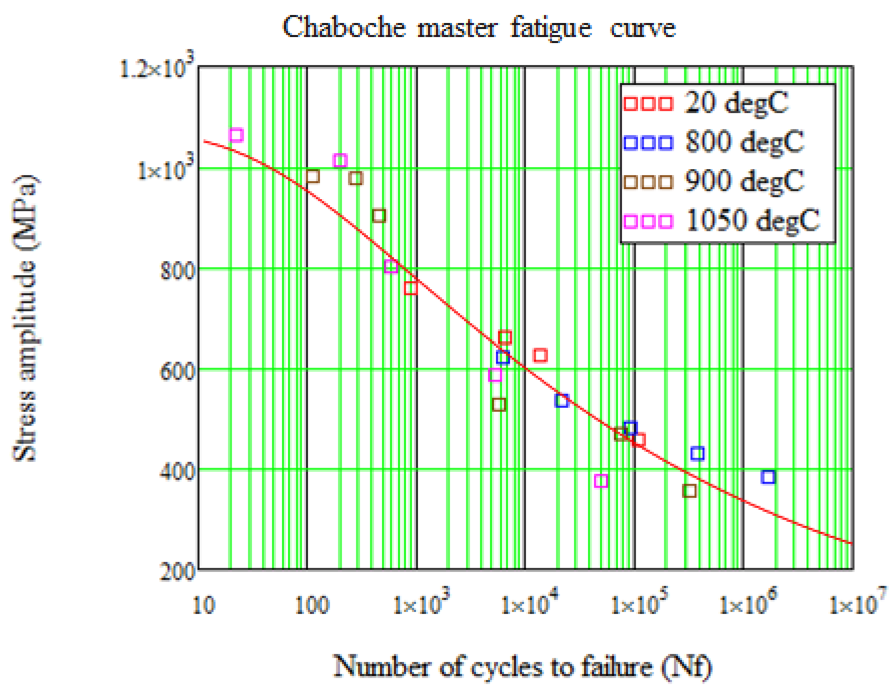


Fig. 5-26 Collapsing the Chaboche curves (b)



Instead, the temperature is accounted for by normalizing the input stress history prior to cycle counting using the normalisation table defined in the materials data.

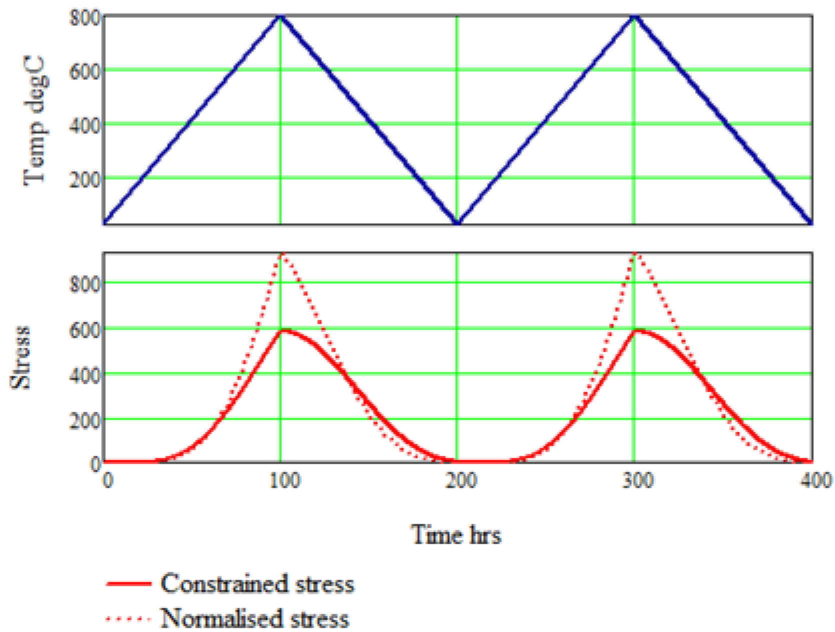
Here is an example of a normalization table:

Temperature (deg C)	Normalization parameter
20	1
800	0.65
900	0.45
1050	0.43

For each point in the stress history, the stress is divided by the normalization parameter for the equivalent temperature at that point in time.

If the temperature lies between the temperatures in the table, the normalization parameter is linearly interpolated. If the temperature is below the minimum temperature in the table, the normalization parameter is set to the value for the minimum. Extrapolation is used if the temperature exceeds the highest value in the table.

Fig. 5-27 Temperature vs constrained and normalised stress



The temperature-normalized time signal is rainflow cycle counted in the usual way in order to obtain the normalized fatigue cycles. The fatigue damage is calculated using a modified version of the algorithm used for the regular Chaboche fatigue model; however, the fatigue parameters now relate to the single collapsed curve and require no temperature interpolation.

If the temperature is above the maximum allowed, the damage is set to the value specified by the StaticFailureDamage property.

The equation forces asymptotes at sigma_u and sigma_lo. Any cycle amplitude (after any mean stress correction) that is greater than or equal to sigma_u will cause static failure. Any cycle amplitude that is less than or equal to sigma_lo will cause zero damage.

Using Chaboche mean stress correction

To use the Chaboche mean stress correction, the MeanStressCorrection property must be set to “Chaboche”. This allows other mean stress corrections to be used including None, Goodman, Gerber, GoodmanTensionOnly and GerberTension-Only. FKM and Interpolate mean stress options are not allowed.

Note If *Chaboche Transient* is chosen as mean stress correction method during a back calculation, then a static failure check is carried out (it cannot be switched off).

When None, Goodman or Gerber are used, the effect of the material property “b” is removed by forcing the mean stress of the cycle to be 0 in the equation.

For Goodman and Gerber, sigma_u is used as the UTS.

SN Engine properties InterpolationLimit, SmallCycleCorrection and CertaintyOf-Survival are not supported in the ChabocheTransient method. No surface finish corrections are performed when using the ChabocheTransient method.

When duty cycles are analyzed using Chaboche Transient where the events have constant temperatures, Independent mode will use a different constant temperature on each event.

However, for combined full and combined fast methods, only one temperature can be used for the whole duty cycle. The temperature used will depend on the TemperatureSelection property, and can be the maximum temperature across all events, the minimum temperature or the median of the maximum and minimum (i.e., (max+min)/2).

If we are using duty cycles and the temperature selection is AllAvailable, all data is analyzed using the correct temperature, since the temperature correction is applied prior to cycle counting and thus there is no cross event cycle issue to resolve.

Strain-based FE data cannot be used for this method, as the conversion from strain to stress is temperature dependent. This can be handled by performing a visco-elastic-plastic FE analysis to get the correct stress values from the FE solver.

Custom

The “Custom” option allows for a user-defined S-N method. This option allows the user to customise the software using a Python script. The script is used to define the relationship between the rainflow count and the damage, and may involve the use of custom material properties and a user-defined mean stress correction.

An example of the custom SN method in use is in the DesignLife Worked Examples, *Example 11 Using Python to Define a Custom SN Method*.
Syntax and properties of the custom SN method are described in detail in the DesignLife User Guide.

Also, if a hybrid load provider is used, the temperature can vary cycle-by-cycle. See “Extracting Temperatures Cycle by Cycle” on page 153.

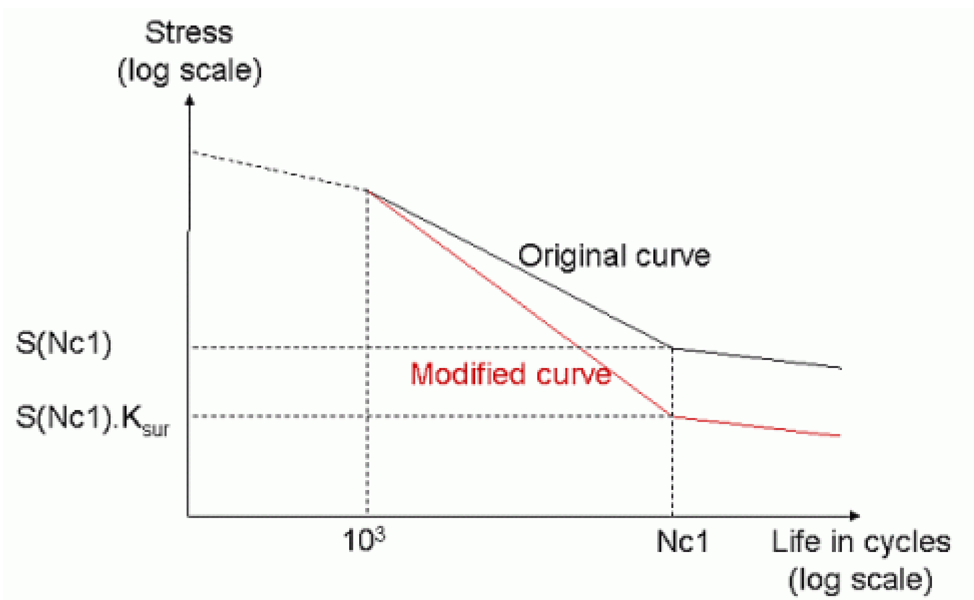
5.3.5 Note on Application of Surface Correction Factors

The fatigue strength of manufactured components and structures is also affected by the surface condition—the level of finish and any subsequent surface treatment. These effects may be taken into account in nCodeDT by a surface factor K_{sur}, derived from information provided on the Material Map, which is then used to adjust the fatigue properties of the material before fatigue calculation. The derivation of the surface factors is described in the section on Material Mapping. This note concerns how the surface factor is applied to modify the S-N data. The mode of application depends upon the type of data (SNMethod). See "Surface Finish and Surface Treatment Settings" on page 60. However, in each case, the basic principle is the same in that the surface condition has the greatest effect in the high-cycle regime and progressively less as the life reduces, with little or no effect in the low-cycle regime.

Standard

Standard S-N curves are modified by changing the slope and intercept of the central portion of the S-N curve so that the fatigue strength is reduced by a factor K_{sur} at life ≥ Nc1 (the transition life), but remains the same for life ≤ 1000 cycles. See Figure 5-28 below:

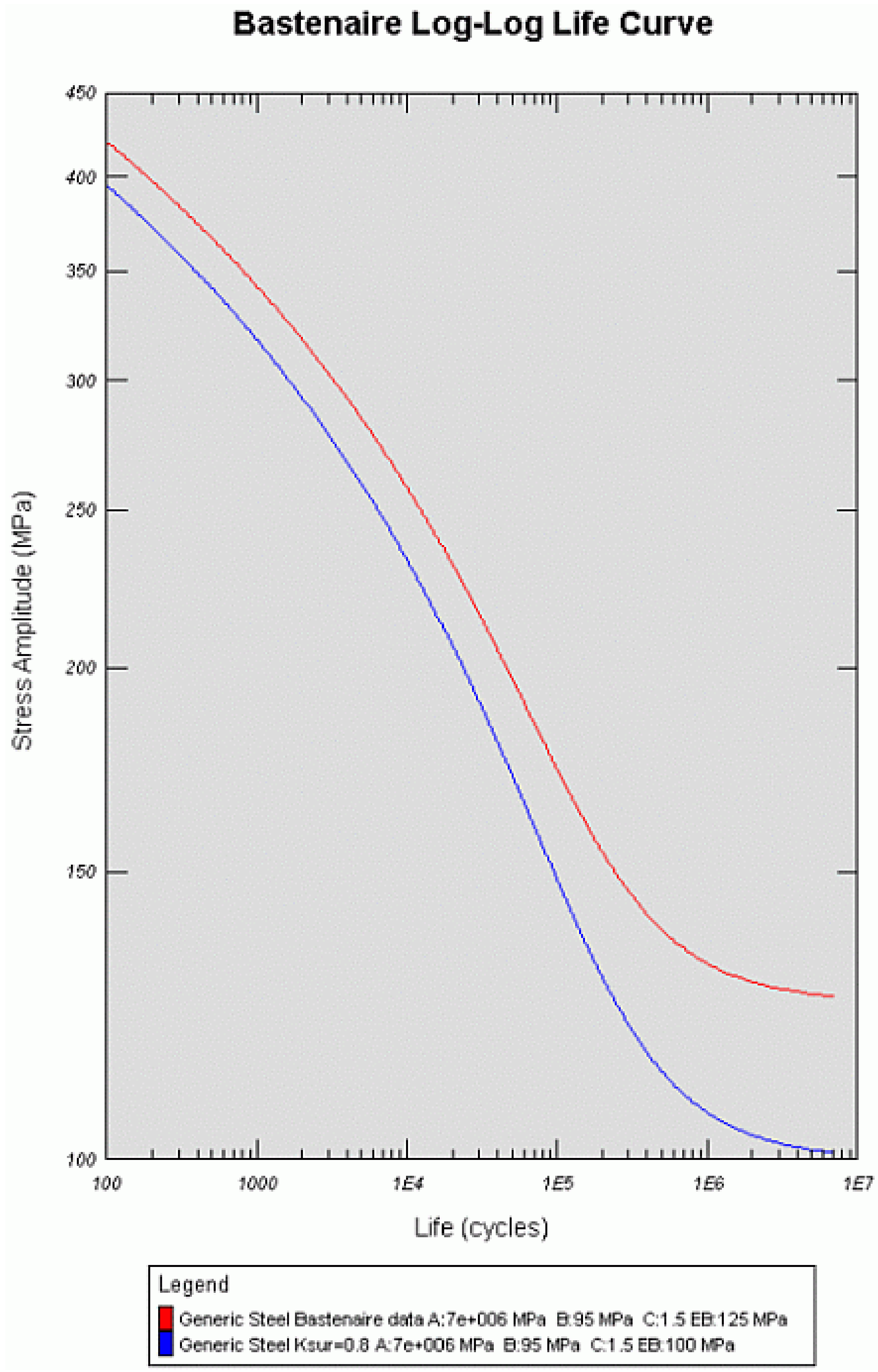
Fig. 5-28 Modification of Standard S-N curves to take into account surface condition



Bastenaire

Bastenaire data is modified by multiplying the Bastenaire fatigue limit parameter EB by K_{sur} resulting in shifting the curve as illustrated in Figure 5-29:

Fig. 5-29 Effect of surface finish correction on Bastenaire S-N curves



Multi-curve Options

For all the other S-N options, that is MultiMeanCurve, MultiRRatioCurve, Haigh and MultiTemperatureCurve, all the stress values (Amplitude, Range or Max stress) are multiplied by a factor, say K_{sur}' which is a function of K_{sur} and the life value N_f corresponding to the stress value.

For $N_f \le$ 1E3, $K_{sur}' = 1.0$

For $N_f \ge Ne$, $K_{sur}' = K_{sur}$

For 1E3 < N_f < Ne, K_{sur}' is interpolated logarithmically as follows:

$$K_{sur}' = 10^{\log K_{sur} \cdot (3 - \log N_f)/(3 - \log N_e)}$$

5.3.6 CombinationMethod

For all load providers apart from the Vibration Load Provider, the analysis engine creates a stress tensor history, $\sigma_{ij}(t)$. In order to make a fatigue calculation, we need to reduce this stress tensor to a scalar value, so that we can cycle count it and compare the resulting cycles to an S-N curve or curves. This process is called stress combination. The available options for combined stress damage parameters are:

- AbsMaxPrincipal
- SignedVonMises
- SignedShear
- CriticalPlane
- MaxPrincipal
- VonMises
- Shear

In general, a stress tensor has nine components, but due to symmetry, this reduces to six.

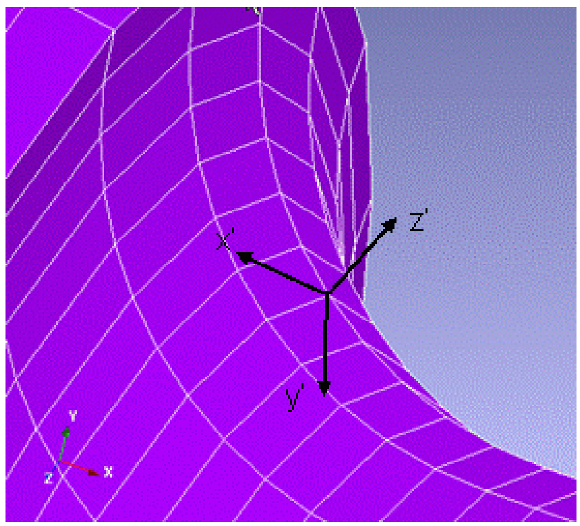
$$\sigma_{ij} \equiv \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{zx} \\ \sigma_{xy} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{yz} & \sigma_{zz} \end{bmatrix}$$

However, the majority of fatigue cracks initiate at free surfaces. At a free surface, we can assume that there is no direct or shear stress applied to the surface. This means that if we choose an appropriate Cartesian coordinate system—one where the z-axis is a surface normal—all stress components containing a “z” disappear and we are left with a stress tensor with only three non-zero components.

$$\sigma_{ij} \equiv \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & 0 \\ \sigma_{xy} & \sigma_{yy} & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

In DesignLife, stresses are flagged as 2-D at the translation stage if they are truly plane stresses (e.g., stresses from thin shell elements) or if they have been resolved to the plane of a free surface. See the section on FE Results for details of surface resolution. If the stresses are resolved to the plane of the surface, the zz, yz and zx stress components will be ignored (assumed to be zero). 2-D stresses may be processed more quickly.

Fig. 5-30 Surface resolution - z' as a surface normal, 2-D stresses



The coordinate system x'y'z' has z' as a surface normal. Stresses in this coordinate system can be assumed to be 2-D.

Most of the combination methods require the determination of principal stresses. The principal stresses are the eigenvalues of the stress tensor matrix.

In the case of 2-D stresses, the calculation of the two in-plane principal stresses reduces to the solution of a quadratic:

$$\sigma_{principal} = \frac{\sigma_{xx} + \sigma_{yy}}{2} \pm \sqrt{\left(\frac{\sigma_{xx} - \sigma_{yy}}{2}\right)^2 + \sigma_{xy}{}^2}$$

The third principal in this case is the surface normal stress, which is always zero.

Whether the stress tensor is flagged as 2-D or 3-D, the result is three principal stresses $\sigma_{1,2,3}$ which are ordered so that $\sigma_1 \geq \sigma_2 \geq \sigma_3$. These are useful for calculating the other combined stress options as follows.

AbsMaxPrincipal

The Absolute Maximum Principal stress is defined as the principal stress with the largest magnitude:

$$\sigma_{AMP} = \sigma_3 \ if \ |\sigma_3| > |\sigma_1| \ otherwise \ \sigma_{AMP} = \sigma_1$$

SignedVonMises

The Signed von Mises stress is the von Mises stress, but forced to take the sign of the Absolute Maximum Principal stress, i.e.,

$$\sigma_{SVM} = \frac{\sigma_{AMP}}{|\sigma_{AMP}|} \cdot \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}$$

SignedShear

The Signed Shear stress is the Maximum Shear stress (Tresca Criterion), forced to take the sign of the Absolute Maximum Principal stress. A factor of 2 is applied to ensure that the parameter has the same value as the Absolute Maximum Principal stress under uniaxial loading conditions.

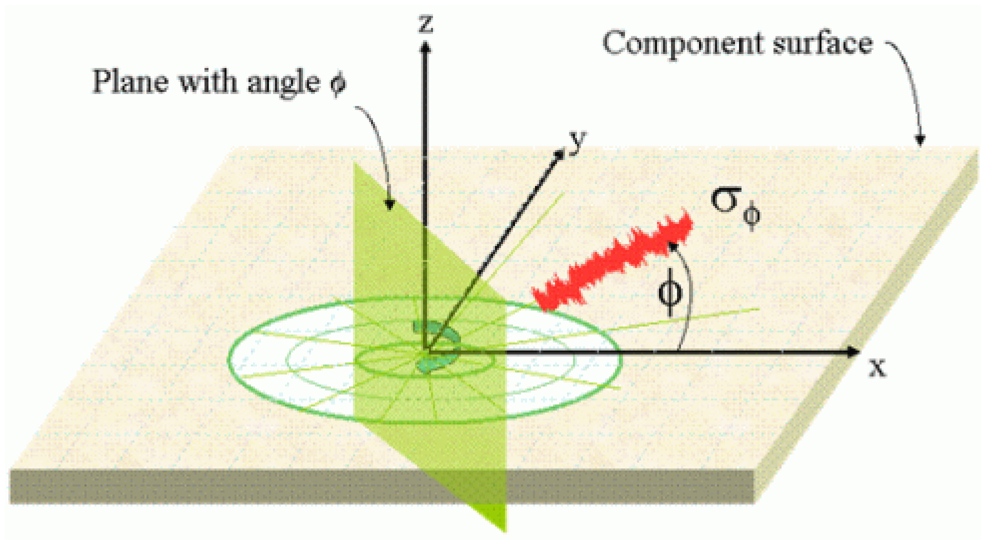
$$\sigma_{SSh} = \frac{\sigma_{AMP}}{|\sigma_{AMP}|} \cdot (\sigma_1 - \sigma_3)$$

CriticalPlane

The normal stress is calculated and rainflow counted on multiple planes. The critical plane is the plane with the most predicted fatigue damage. For stresses flagged as 2-D the planes on which the normal stress is determined have normals that lie in the plane of the physical surface, i.e., in the x-y plane of the 2-D stress results coordinate system. The orientation of each plane is defined by the angle ϕ made with the local x-axis.

Note that if the Critical Plane option is being used with solid elements (3D stresses) the distribution of critical planes to be analysed is also three dimensional. In order to achieve a resolution of 10 degrees 204 planes must be analysed, each plane being defined by the orientation of its normal, using two angles (theta and phi). This will significantly increase run-time.

Fig. 5-31 Resolution of normal stress for critical plane analysis

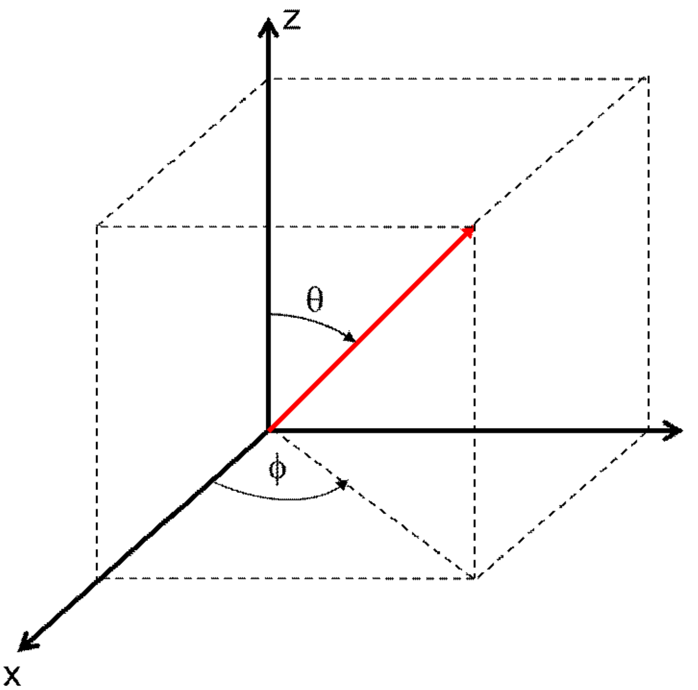


The stress on each plane is calculated from:

$$\sigma_{\phi} = \frac{\sigma_{xx} + \sigma_{yy}}{2} + \frac{\sigma_{xx} - \sigma_{yy}}{2} \cdot \cos 2\phi + \sigma_{xy} \sin 2\phi$$

ϕ can take the values 0, 10, 20, 30, ...170 degrees.

Fig. 5-32 Plane orientation angles illustrated relative to results co-ordinate system



MaxPrincipal

This is the maximum principal stress. If the three principal stresses are σ_1, σ_2, σ_3 where $\sigma_1 \geq \sigma_2 \geq \sigma_3$, this is σ_1. Use with caution as this isn't a recommended damage parameter.

VonMises

This is the von Mises stress. $\sigma_{VM} = \sqrt{\dfrac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}$

...where σ_1, σ_2, σ_3 are the three principal stresses. Since the von Mises stress is always positive, this is not a recommended damage parameter and should be used with caution.

Shear

This is the maximum shear stress, i.e., $\sigma_{sh} = \sigma_1 - \sigma_3$ where σ_1, σ_2, σ_3 are the three principal stresses and $\sigma_1 \geq \sigma_2 \geq \sigma_3$. A factor of 2 is applied to be consistent with the Signed Shear parameter. This is always positive and is not recommended as a damage parameter.

5.3.7 MeanStressCorrection

The main feature of a stress cycle that affects fatigue damage is its range. Fatigue damage is also influenced by the mean stress of each cycle. Mean stress correction methods allow the effect of mean stress to be modeled and taken into account in the life prediction. This section describes the different methods supported. Whether or not a particular mean stress correction method can be applied depends upon the formulation of the S-N data (SNMethod) that is being used. The allowable combinations are detailed in the table below:

Table 5-4 Allowable S-NMethod/MeanStressCorrection Combinations

		SNMethod							
		Standard	**Multi Mean Curve**	**MultiR Ratio Curve**	**Haigh**	**Bastenaire**	**Multi-Temperature Curve**	**Chaboche**	**Chaboche Transient**
Mean Stress Correction	None	✓	✓*	✓*	×	✓	✓	✓	✓
	Goodman	✓	✓*	✓*	×	✓	✓	✓	✓
	Gerber	✓	✓*	✓*	×	✓	✓	✓	✓
	Interpolate	×	✓	✓	✓	×	×	×	×
	FKM	✓	✓*	✓*	×	✓	✓	×	×
	Goodman-Tension-Only	✓	✓*	✓*	×	✓	✓	✓	✓
	GerberTensionOnly	✓	✓*	✓*	×	✓	✓	✓	✓
	Chaboche	×	×	×	×	×	×	✓	✓
	Walker	✓	×	×	×	×	×	×	×

✓ = Allowed
× = Not allowed
✓* = Allowed but a curve for R = -1 or zero mean stress must be present

Each of the mean stress correction methods is described below.

None

Mean stress is not taken into account. This can work with all S-N data types, but if it is to function with MultiMeanCurve or MultiRRatioCurve data, the material dataset must include datasets corresponding to zero mean or R = -1, otherwise an error message is issued. If a standard SN curve is used, no account is taken of the RR value for that curve; it is used as-is.

Goodman

The Goodman mean stress correction calculates an effective stress amplitude based on the mean stress and UTS of each cycle. Again, this can work with all S-N data types, but if it is to function with MultiMeanCurve or MultiRRatioCurve data, the material dataset must include datasets corresponding to zero mean or R = -1, otherwise an error message is issued.

In its original form, it is used to calculate an effective stress S_e that can be compared to an R = -1 S-N curve, based on the stress amplitude S_a, mean stress S_m and the material UTS:

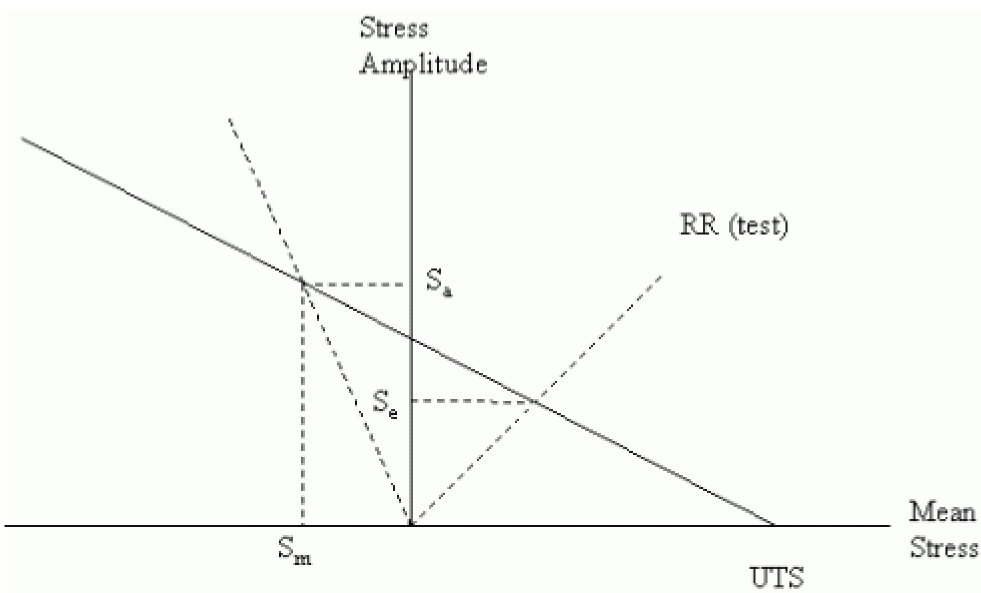
$$\frac{S_a}{S_e(R=-1)} + \frac{S_m}{UTS} = 1$$

In DesignLife, this has been extended to allow the equivalent stress to be determined for any R-ratio:

$$S_e(RR) = S_a \frac{UTS}{UTS - S_m + S_a(1+RR)/(1-RR)}$$

This is illustrated below:

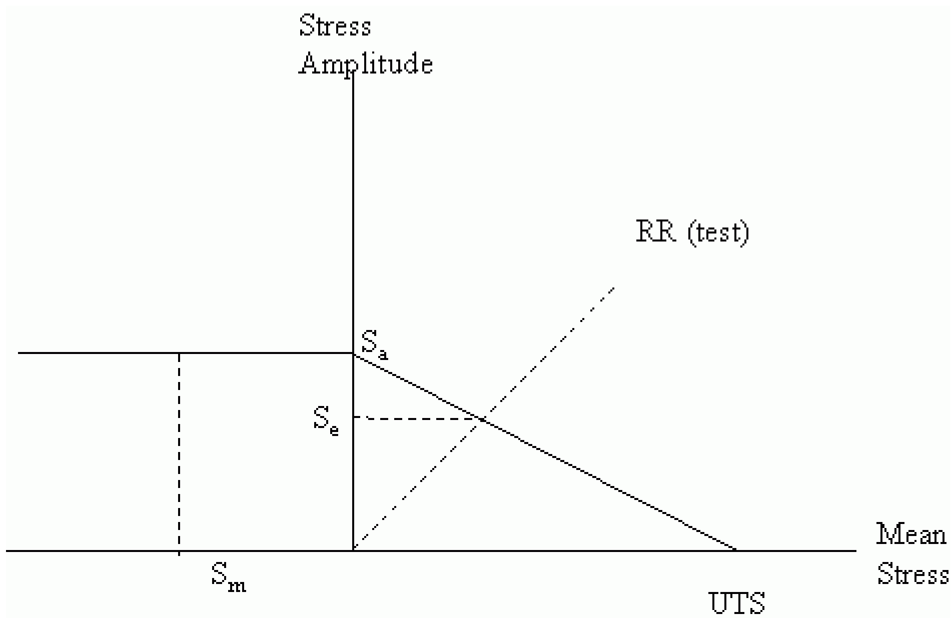
Fig. 5-33 Graphical interpretation of Goodman correction



GoodmanTensionOnly

The Goodman correction in the form described above can be rather non-conservative for cycles with compressive mean stresses. GoodmanTensionOnly addresses this by flattening off the constant life curve when the mean stress is compressive, as illustrated below:

Fig. 5-34 Graphical representation of GoodmanTensionOnly



Gerber

The Gerber correction in its original form is similar to Goodman, except that the second term is squared:

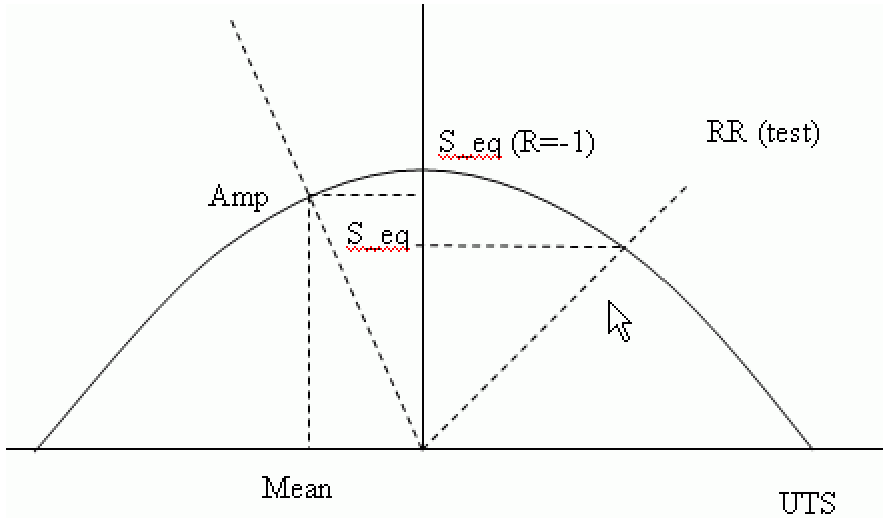
$$\frac{S_a}{S_e(R=-1)}+\left(\frac{S_m}{UTS}\right)^2=1$$

We can then calculate the equivalent stress for any other R-ratio RR:

$$S_e(RR)=\left(\sqrt{\left(1+\frac{4\cdot S_e{}^2(R=-1)\cdot(1+RR)^2}{(1-RR)^2\cdot UTS^2}\right)}-1\right)\cdot\frac{(1-RR)^2UTS^2}{2\cdot S_e(R=-1)\cdot(1+RR)^2}$$

Graphically, this looks like:

Fig. 5-35 Graphical interpretation of the Gerber correction in its original form

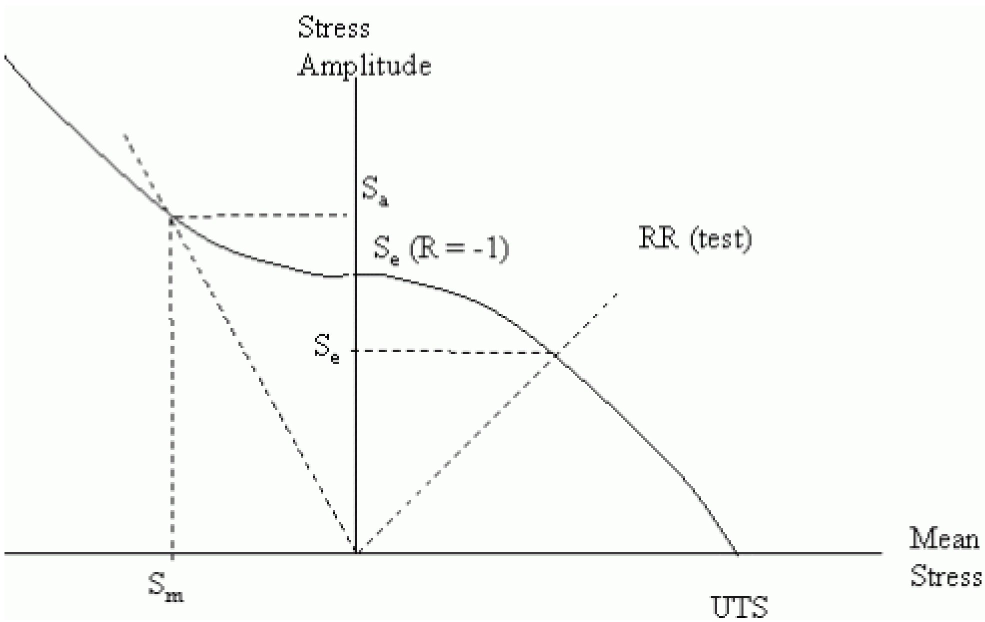


In practice this is not very realistic, because the reduction in fatigue strength under compressive loading is the same as it is under tensile loading. The Gerber correction in its original form will in general be rather non-conservative for tensile mean stresses and rather pessimistic for compressive mean stresses. This is particularly unrealistic, and so the method as implemented in nCodeDT is modified so that for compressive loadings:

$$\frac{S_a}{S_e(R=-1)} - \left(\frac{S_m}{UTS}\right)^2 = 1$$

Nevertheless, this method (illustrated in Figure 5-36) is not highly recommended. Also (although this is not likely to be an issue in practice) the user can imagine that this method may have problems finding a unique solution for the equivalent stress, or any solution at all, when RR (the R-ratio of the test data) lies deep in the compressive region.

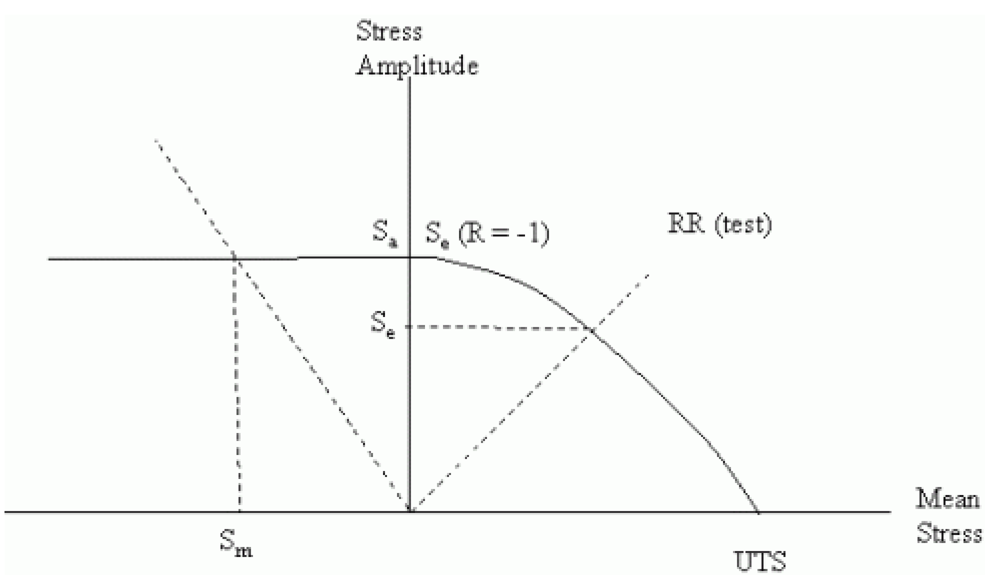
Fig. 5-36 Modified Gerber mean stress correction as implemented in DesignLife



GerberTensionOnly

This option provides a better solution by flattening off the constant life diagram in the compressive region in the same way as for GoodmanTensionOnly.

Fig. 5-37 GerberTensionOnly



Interpolate

When Interpolate is selected, the mean stress effect will be taken into account by interpolation (or extrapolation) from multiple curves. The exact method depends upon the S-N data type and the differences are detailed in the section on SNMethod (MultiMeanCurve, MultiRRatioCurve and Haigh).

FKM

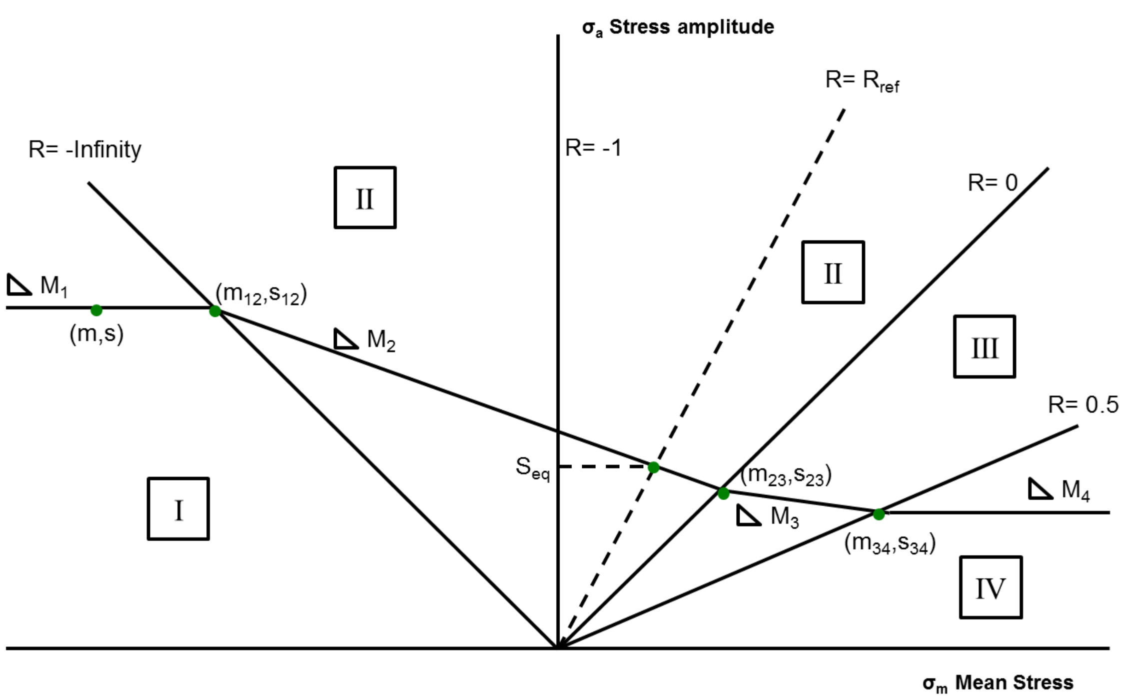
The FKM method as implemented here is based on the method described in the FKM Guideline "Analytical Strength Assessment of Components in Mechanical Engineering", Tr. E. Haibach. 2003.

In essence it uses 4 factors M_{1-4} which define the sensitivity to mean stress in 4 regimes:

1. R>1
2. –infinity <= R < 0
3. 0 <= R < 0.5
4. 0.5 <= R < 1

...where R is the stress ratio (min/max) of the loading cycle. The method allows us to determine the equivalent stress amplitude S_{eq} at a particular material R-ratio, R_{ref}. The method is illustrated in the form of a constant life or Haigh diagram in Figure 5-38:

Fig. 5-38 Graphical representation of the FKM mean stress correction



The values of $M_{1\text{-}4}$ can be determined from material tests or estimated as follows:

$M_1 = 0$

$M_2 = -M_\sigma$

$M_3 = -M_\sigma /3$

$M_4 = 0$

...where the value of M_σ is estimated as follows for the supported material types:

$M_\sigma = a_M{*}10^{-3}{*}R_m + b_M$ where a_M and b_M are constants and R_m is the UTS in MPa.

Values of a_M and b_M for the different supported material classes are as follows:

Table 5-5 FKM Mean Stress Correction Parameters

Material Type	Steel	GS (cast steel)	GGG (nodular cast iron)	GT (malleable cast iron)	GG (cast iron with lamellar graphite)	Wrought Al alloy	Cast Al alloy
nCode Material Type No.	13,14,16-22-25,26-99	9-12,15	5-8	2-4	1	100-105	106
a_M	.35	.35	.35	.35	0	1.0	1.0
b_M	-0.1	0.05	0.08	0.13	0.5	-0.04	0.2

In the software, if $M_{1\text{-}4}$ are undefined, and the material type is one of those listed, all the parameters will be estimated using these rules. If only M_2 is defined, then M_1 and M_4 will be set to zero and M_3 to $M_2/3$.

Chaboche

The Chaboche mean stress correction can be used with the Chaboche and ChabocheTransient method. See "Chaboche" on page 110.

Walker

The Walker mean stress correction calculates an effective stress amplitude based on the maximum stress, the R Ratio of each cycle and the Walker material parameter Gamma. This method currently only works with standard S-N data types.

In its original form it is used to calculate an effective stress *Se* that can be compared to an R = -1 S-N curve, based on the maximum stress amplitude *Smax* and the *R Ratio* (RR):

$$\sigma_{ar} = \sigma_{max}\left(\frac{1-R}{2}\right)^{\gamma}$$

Expressed in terms of the stress amplitude:

$$\sigma_{ar} = \sigma_a\left(\frac{2}{1-R}\right)^{\gamma}$$

This method uses two parameters *GammaP* for tensile mean stress cycles and *GammaN* for negative mean cycles.

5.3.8 InterpolationLimit

This property applies only when the SNMethod is set to MultiMeanCurve or MultiRRatioCurve. It controls whether or not extrapolation will be carried out if the mean or R-ratio of a cycle lies outside the range defined by the curve set. See the section on "SNMethod" on page 92 for more details.

5.3.9 MultiaxialAssessment

Multiaxial assessment provides information about how the stress state varies throughout the loading history.

There are four multiaxial assessment options:

- None
- SimpleBiaxiality
- Standard
- Auto

All except the None option require the stresses to be flagged as 2-D. The basic input to the multiaxial assessment calculation is a 2-D stress tensor time history; multiaxial assessment options do not work with a Vibration Load Provider.

The four options are described in more detail below.

MultiAxialAssessment = None

No multiaxial assessment is carried out.

MultiAxialAssessment = SimpleBiaxiality

This option is very similar to that previously implemented in FE-Fatigue. It provides some information about the behavior of the stress tensor in terms of the biaxiality ratio and the orientation of the absolute maximum principal stress. It works as follows:

The input to the calculation is a 2-D or surface-resolved stress tensor history $\sigma_{ij}(n)$ where n is the point number from the time history or peak-valley sequence. The zz, xz and yz components will be assumed to be zero, i.e.,

$$\sigma_{ij} = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & 0 \\ \sigma_{xy} & \sigma_{yy} & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

At each point (n), the principal stresses are determined. For the purposes of this calculation we will designate the principal stresses σ_1, σ_2 and σ_z, where

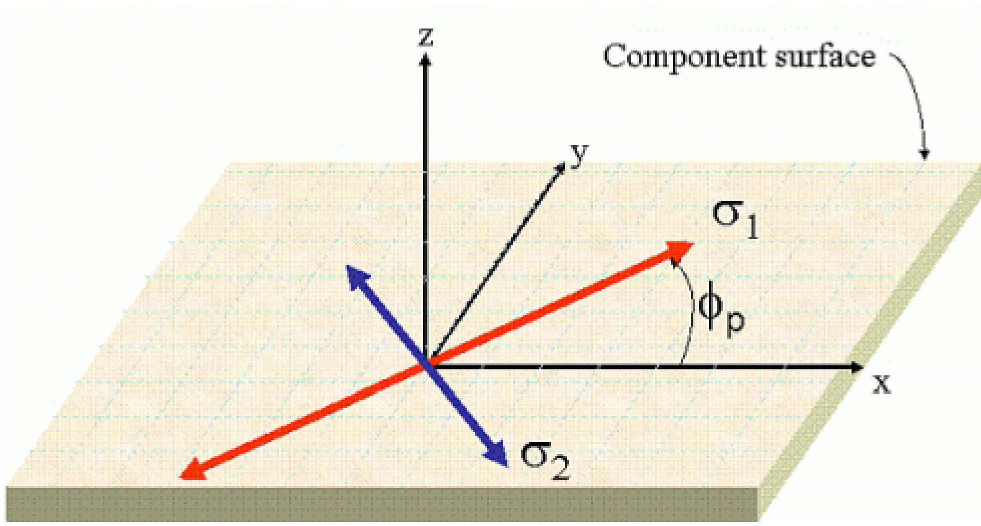
$\sigma_z = \sigma_{zz} = 0$ (the surface normal stress)

and

$|\sigma_1| \geq |\sigma_2|$ (the in-plane principal stresses)

So σ_1 is the Absolute Maximum Principal Stress and σ_2 is the other in-plane principal. At the same time, the orientation ϕ_p of σ_1 relative to the local x-axis is determined.

Fig. 5-39 Surface stress state reduced to 2 principal stresses and their orientation



The biaxiality ratio is defined

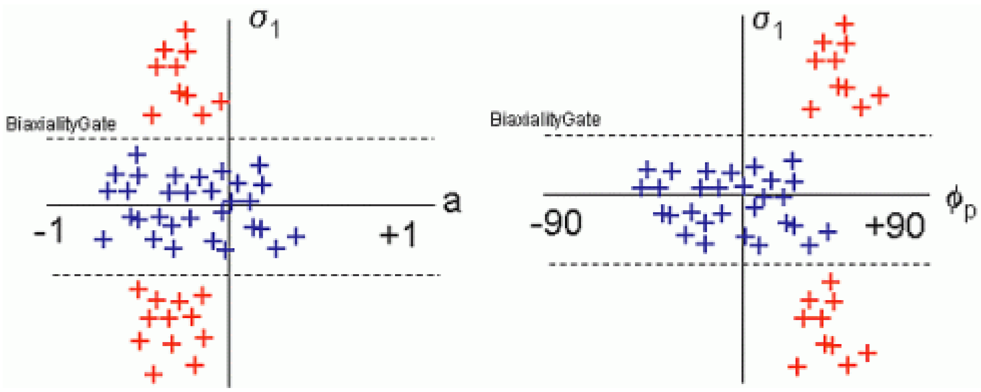
$$a = \frac{\sigma_2}{\sigma_1}$$

Statistics of a(n) and ϕ_p(n) are calculated. In order to reduce the impact of data points (n) where the stresses are very small, the number of points that contribute to the statistics are reduced through the use of the BiaxialityGate property, which is defined as a percentage of the material UTS (typically 20%). Points n will be excluded from the biaxiality analysis if

$$|\sigma_1(n)| < \frac{BiaxialityGate}{100} \cdot UTS$$

This is illustrated in Figure 5-40; the data points in blue will be excluded from the statistics.

Fig. 5-40 Gating out small stresses from biaxiality calculations



The following quantities are reported in the analysis results:

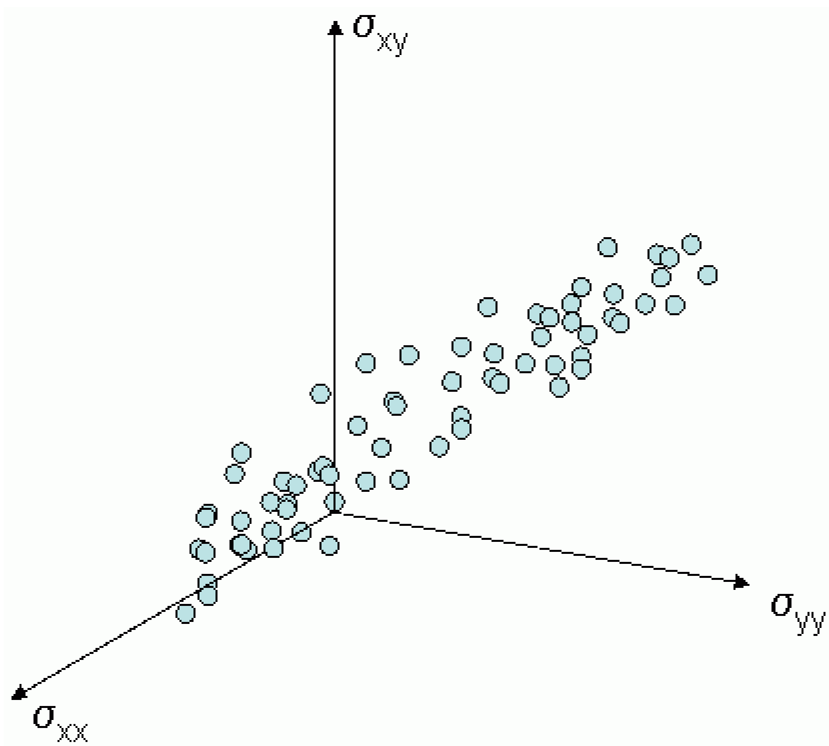
- **Mean Biaxiality Ratio** —The mean value of a(n) during the loading sequence
- **Standard Deviation of the Biaxiality Ratio**—The standard deviation of a(n) during the loading sequence. It helps to understand how proportional the loading is. If it is zero, then all loading is of the same type.
- **Angle Range** —To calculate this, the values of ϕ_p are collected into three degree bins. We then look for the largest continuous block of unoccupied bins and subtract this angle from 180 degrees.
- **Most Popular Angle** —This refers to the most populated bin (see above).
- **Biaxiality Gate (In stress units**—Taking into account the %Biaxiality gate and the material UTS)
- **Maximum Principal Stress Range**—The maximum range of σ_1 or the largest range between σ_1 and σ_2

MultiAxialAssessment = Standard

This method is a more recent development that provides a more robust measure of the biaxiality and non-proportionality of the local loading (stress state).

Visualize the stress history as a cloud of data in which every point in the stress history is represented by a point on a 3-D scatter plot of σ_{xx} vs σ_{yy} vs σ_{xy}. See Figure 5-41. By analyzing the shape and orientation of this data cloud, we can obtain numerical measures of the biaxiality and non-proportionality of the loading.

Fig. 5-41 3-D data cloud. Each point represents a point in the loading history.

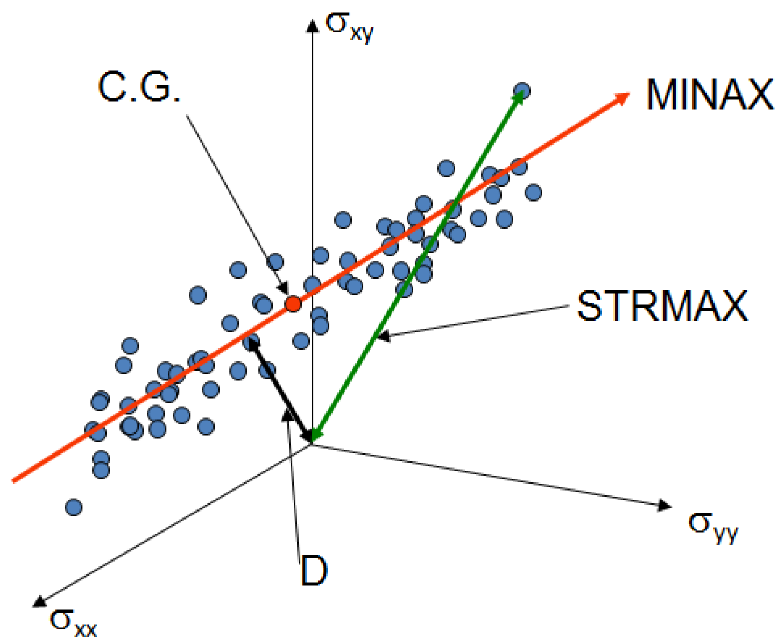


The calculation procedure is as follows:

1. Determine the position of the center of gravity of the data cloud.
2. Determine the principal moments of inertia of the data cloud about the center of gravity $I_1 \geq I_2 \geq I_3$.
3. Find the vector **MINAX** corresponding to the principal axis with the minimum moment of inertia I_3.
4. Find **STRMAX** (the distance to the point that is furthest from the origin).

5. Find the offset D of MINAX from the origin.

Fig. 5-42 Analysis of data cloud for Standard Multiaxiality Assessment



Essentially, we perform a 3-D linear regression on the data cloud. If the cloud of data correlates closely with a straight line through the origin, the loading is proportional. The larger the aspect ratio of the cloud and/or its offset from the origin, the more non-proportional the loading. The orientation of the vector **MINAX** tells us the average biaxiality. So the calculation proceeds as follows:

6. Define the cloud aspect ratio:

$$ASPECT = \frac{I_3}{I_1}$$

7. Define the cloud offset factor:

$$OFFSET = \frac{D}{2 \times STRMAX}$$

8. Then we define a non-proportionality factor:

$$NONPROP = \sqrt{ASPECT + OFFSET^2}$$

9. The three components (σ_{xx}, σ_{yy}, σ_{xy}) of the vector **MINAX** define a 2-D stress tensor σ_{ij} from which we can obtain two principal stresses σ_1 and

σ_2 where $|\sigma_1| \geq |\sigma_2|$. The mean biaxiality is then calculated from:

$$BIAXIALITY = \frac{\sigma_2}{\sigma_1}$$

10. At the same time, we can determine the orientation of the plane corresponding to **MINAX**—i.e., the orientation of σ_1 from step 9. This gives us the Dominant Stress Direction, which will correspond to the most likely critical plane orientation.

The mean biaxiality and non-proportionality factors are reported in the analysis results, together with the Dominant Stress Direction.

MultiAxialAssessment = Auto

Auto mode uses a two-pass approach, which overrides the *CombinationMethod* setting. When *Auto* is selected, four additional properties must be set:

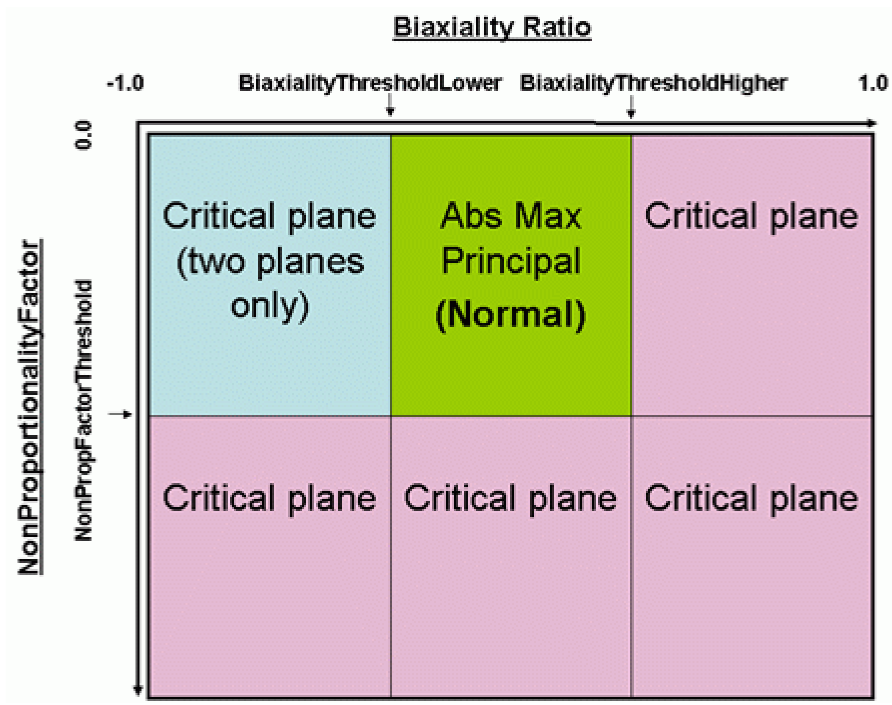
Table 5-6 Additional Properties Associated with Auto Mode

Property	Default Setting
ZeroDamageStressPercent	10
NonPropFactorThreshold	0.25
BiaxialityThresholdLower	-0.6
BiaxialityThresholdUpper	0.6

1. In the first pass, *CombinationMethod* is set to *AbsMaxPrincipal* (the "Normal" method), and a *Standard* assessment is carried out.
2. A second pass using a different stress combination method may be carried out *if* the value of *STRMAX* (see more about STRMAX in section in "MultiAxialAssessment = Standard" on page 138) exceeds the percentage of the material strength (UTS) defined by *ZeroDamageStressPercent* AND *if* the *non-proportionality* and/or *biaxiality* values lie outside the ranges

defined for the *Normal* method by the relevant threshold values. The Auto mode is summarized in the following diagram.

Fig. 5-43 Application of thresholds in the Auto multiaxial option



The *AbsMaxPrincipal* and *CriticalPlane* stress combination methods have already been defined. The other possible option is the *Two Plane* option. This is similar to the Critical Plane method except that calculations are made on two planes only—in the Dominant Stress Direction, and at 90 degrees to the Dominant Stress Direction.

The results reported for *Auto* mode are as for Standard, with the addition of the Calculation Method (Normal, Critical Plane, Two Plane).

Note on Duty Cycle Processing

When a duty cycle is being processed, the biaxiality parameters are calculated for the individual unique events that make up the duty cycle, as well as for the duty cycle as a whole. The statistics for the duty cycle as a whole take into account the number of repeats of each event.

5.3.10 Certainty of Survival

The certainty of survival allows statistical variations in material behavior to be taken into account. The usual application of this is to provide a more conservative prediction to ensure a safer design. In general, the variability in material properties is characterized by a standard error parameter, which will normally be obtained when fitting material curves to S-N test data. The certainty of survival value is converted into a number of standard errors using the lookup table and this is used to adjust the fatigue strength or calculated fatigue life. The exact way this is done depends upon the formulation of the S-N data and is described in more detail in each case in the section on "SNMethod" on page 92. Note that if the standard error parameter in the material dataset is set to zero, changing the Certainty of Survival will have no effect on the results. If this is the case, it does not mean there is no scatter in the material behavior, merely that it has not been calculated.

5.3.11 ScaleFactor

This scale factor will be applied to the stresses before the fatigue calculation. The default value is 1.0.

5.3.12 OutputMaxMin

When this property is set to True, the maximum and minimum values of the combined stress, as passed into the rainflow counter, will be reported in the results.

5.3.13 BackCalcMode

The options here are **None** or **ScaleFactor**. When **ScaleFactor** is selected, the S-N Analysis engine will iteratively search for a Scale Factor, which, when applied to the stresses before the fatigue calculation, will produce the **TargetDamage** within a reasonable tolerance set by **BackCalcAccuracy.** This setting overrides the ScaleFactor property setting on the analysis engine (see above). It provides a convenient way of determining a safety factor for complex or variable amplitude loadings. When this option is selected, the damage reported in the results will approximate to the target damage (within the tolerance defined by BackCalcAccuracy), and the resulting scale factor will be reported, too.

Note When back calculations are being done, a static failure check is also carried out by the SN glyph.

5.3.14 TargetDamage

Sets the target damage value to be used for back calculations. Note that it is not always possible to make a successful back calculation. For example, if your loading is constant amplitude and the target damage corresponds to a flat part of the S-N curve, the back calculation will fail, and an appropriate error message issued.

5.3.15 BackCalcAccuracy

This defines a percentage error on the target damage. The back calculation will iterate until the calculated damage matches the target damage within this tolerance.

5.3.16 BackCalcMaxScale

This defines the maximum value of the scale factor that will be reported in the analysis results.

5.3.17 SmallCycleCorrection

This feature adjusts the SN curve to allow for the effect of small cycles as part of a variable amplitude load. The corrections are:

- **None** - No correction, the materials data is unchanged
- **Extrapolate** - The second slope, b2, is set to the same as b1, i.e. no fatigue limit effects
- **Haibach** - If the largest stress cycle is above the SN curve transition point Nc1, the second slope is adjusted so that b2 = b1/(b1 + 2).
  Otherwise b2 = 0 and damage is zero. In Haibach's terms, the first slope is m and the second slope is 2m-1.

BS7608:1993 - If the largest stress cycle is above the SN curve transition point Nc1, the second slope is adjusted so that b2= -b1/(2*b1 - 1). Otherwise b2=0 and damage is zero. Another way of putting this is to say that if the first slope is m (m=-1/b1) the second slope is m+2. Note that Haibach and BS7608:1993 methods are identical for the special case where m = 3 (b1 = -1/3)

As this property operates on the slopes b1 & b2, it is restricted to the Standard SN curve definition and does not support other definitions.

Fig. 5-44 Modified slope b2 in "Extrapolate" small cycle correction

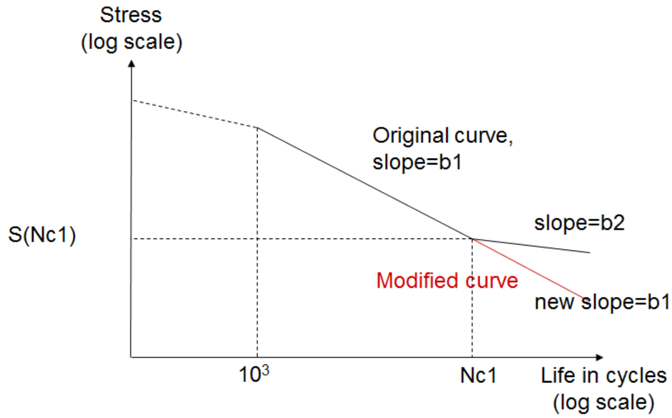
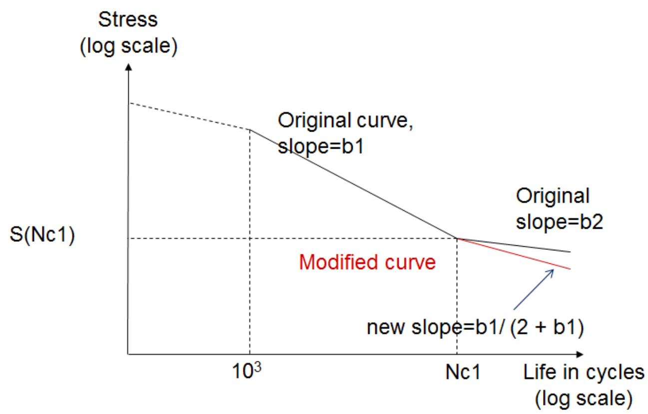


Fig. 5-45 Modified slope b2 in "Haibach" small cycle correction



5.3.18 EventProcessing

A duty cycle is a sequence of events to be processed through the analysis engine. The creation and various types of different duty cycle are described in the section on Loading. Duty cycles can be processed by the S-N Analysis Engine in three different ways, which can be summarized as follows:

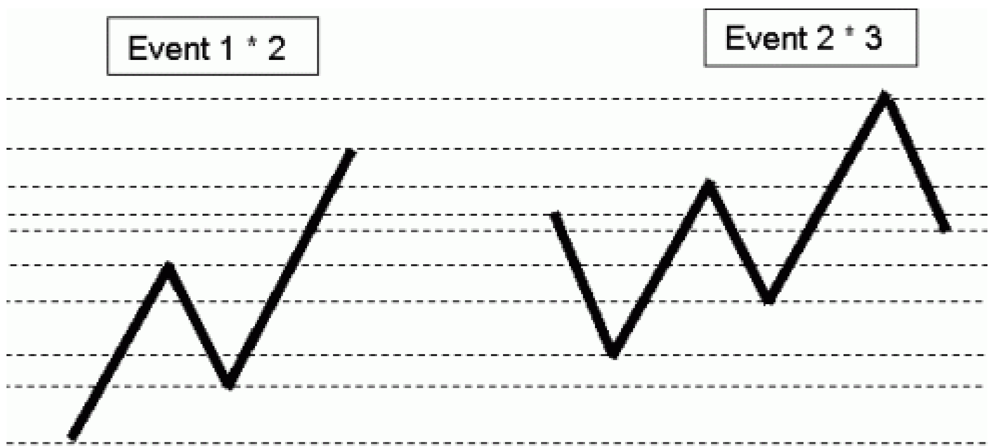
- **Independent** mode. Each unique event in the duty cycle is processed separately, and the damage from each is multiplied by the number of times it appears in the complete sequence. The total damage is the sum of the damage from all the different events. This is fast, but may miss significant cycles that cross different events.
- **CombinedFull** mode. All the events in the duty cycle are concatenated, including all repeats of each event, and the resulting stress history is processed as if it was one long event. This is accurate, but is computationally inefficient and could take a long time to process.

- **CombinedFast** mode. An intermediate method that cycle counts events individually but also captures the cycles that cross events. It will in general be almost as accurate as the CombinedFull option.

When a duty cycle includes one or more Vibration Fatigue Load Providers, only Independent mode will work.

The methods are best explained with the aid of a simple example.

Fig. 5-46 Simple example for duty cycle processing

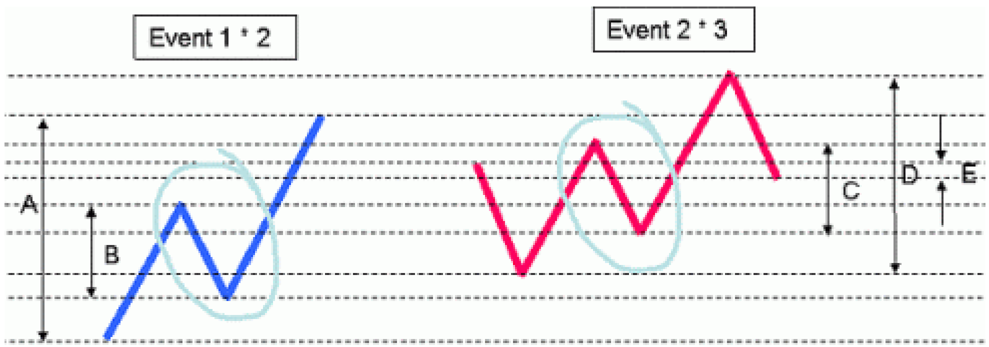


Consider a duty cycle consisting of Event 1, which has two repeats, and Event 2, which has three repeats. We will now consider how the resulting combined stress histories for these two events are handled by the different duty cycle processing options.

Note	Please read the earlier note on rainflow cycle counting first (see 5.3.2 Note on Rainflow Cycle Counting).

Independent

Fig. 5-47 Independent duty cycle processing



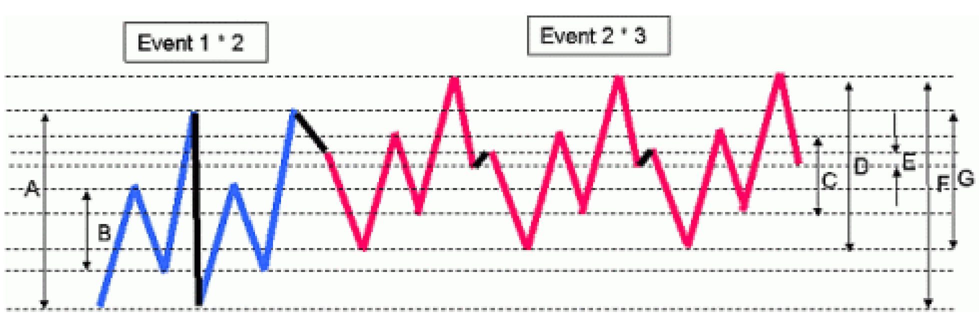
In Independent mode, Event 1 is cycle counted. This results in a cycle with range B and a residual, which is also closed to give cycle A. Each of these cycles appears twice in the duty cycle, so their damage is calculated and multiplied by 2. The same procedure is applied separately to Event 2, resulting in three repeats of cycles C, D and E.

The result can be summarized as 2*A, 2*B, 3*C, 3*D, 3*E.

This method has the virtue of being quick and simple, but it does miss out the obvious larger cycle formed by the minimum of Event 1 and the maximum of Event 2. However, for many applications where there are no big mean shifts between events, it may be sufficiently accurate.

CombinedFull

Fig. 5-48 CombinedFull duty cycle processing



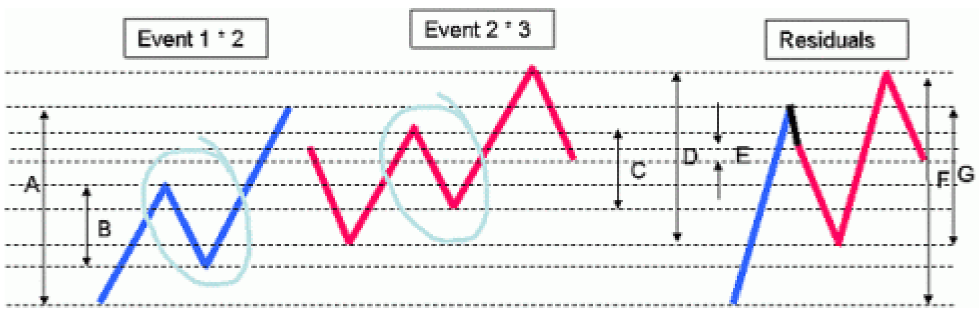
In this method, the entire duty cycle is concatenated, including every repeat of each event in the correct sequence, and the result is rainflow counted from the beginning and the residuals closed down.

The result in terms of cycles can be summarized as: 1*A, 2*B, 3*C, 2*D, 2*E, 1*F, 1*G.

This method has the virtue of being accurate—no assumptions or approximations are made. However, for duty cycles with many repeats of individual events, it could be prohibitively slow.

CombinedFast

Fig. 5-49 CombinedFast duty cycle processing



The CombinedFast option provides a good compromise between speed and accuracy. It works as follows:

1. Each unique event is cycle counted. Cycles that close within individual events are multiplied by the total number of repeats of that event. The residual for each event is also closed and multiplied by the total number of repeats of that event less the number of times the event appears in the duty cycle. In the case of Event 1, we get two repeats of cycle B and one repeat of cycle A.
2. The residuals for each event are concatenated in the sequence in which they occur and then cycle counted, including closing down any residuals.

In this case, the resulting cycles are: 1*A, 2*B, 3*C, 2*D, 2*E, 1*F, 1*G. For this simple example, the result is exactly the same as for the CombinedFull option, but in general this cannot be guaranteed. However, for almost all practical applications, this method should be sufficiently accurate.

5.3.19 OutputEventResults

If this property is set to True, the accumulated damage (and other calculated results) associated with each individual event from the duty cycle will be output in addition to the total damage. If either of the “Combined” options is being used, some cycles may cross events, for example, cycles F and G in the example from the previous section. The damage associated with these cycles is split equally between the two events with which it is associated.

5.3.20 CheckStaticFailure

Static failure checking compares the maximum value of the CombinedStress, as cycle counted, with the material UTS. It also compares the Equivalent stress amplitude (after any mean stress correction) with the UTS. There is currently no criterion applied for compressive failure. There are three options:

- **Warn**—The calculation continues, but a warning message is issued. The damage for this entity is also assigned a numeric value equal to the value of the StaticFailure Damage property, typically 2.0.
- **Stop**—The calculation for the entity stops with a message and the damage for this entity is assigned in the same way as for "Warn".
- **False**—No static failure check is made. Damage is calculated up to the numerical limit defined by the MaxDamage property. Note that this may use an extrapolation of the S-N curve and may predict damage values per cycle greater than 1.0. Setting CheckStaticFailure to False has an additional effect if the SNMethod property is set to Standard. In this case, the adjustment of the S-N curve below 1000 cycles to take into account the static strength of the material is disabled.

Note When the loading is from a Rainflow Histogram, i.e. Vibration Load Provider:

1. The maximum stress is NOT checked against UTS in the same way, hence:
   a. A *static failure* result is not generated.
   b. A warning is not issued.
2. You should check the "return to UTS" time to determine the likelihood if exceeding the UTS.
3. The damage calculation is affected in two ways by this option:
   a. At lives of less than 1000 cycles, the slope of the SN curve is modified to take into account the limitation in fatigue strength imposed by the static strength of the material, so that, at a life of 1 cycle, the maximum stress is equal to the UTS.
   b. For cycles where the maximum stress is above the UTS the life is assumed to be 1 and the damage returned will therefore be the number of cycles/1.

5.3.21 DamageFloor

This parameter sets a lower limit on calculated damage. If a cycle has less predicted damage, the damage value is set to zero. This parameter is somewhat redundant because similar functionality may be accessed using the fatigue cutoff Nfc in the material definition.

5.3.22 MaxDamage

This parameter sets a numerical upper limit on the damage that can be predicted for each cycle.

5.3.23 StaticFailureDamage

This parameter sets a numerical value that may be set for the damage to a particular node or element in the event of the stress exceeding the UTS.
(See CheckStaticFailure above.)

5.3.24 StressGradients

This property has three possible values:

- **Auto**—A correction for stress gradient is applied if stress gradients are present (i.e., if they were requested at the translation stage in the Analysis Group properties).
- **User**—A user-defined stress gradient correction is applied (if stress gradients are available) based on a lookup table provided in a file. A valid file must be specified using the StressGradientsUser property.
- **Off**—No stress gradient correction is applied.

It is well known that the fatigue damage at a stress concentration (e.g., a notch) is normally overpredicted by the local stress at the root of the notch. Another way of putting this is to say that the fatigue strength reduction factor of a notch K_f is always less than the elastic stress concentration factor K_t. The difference between K_f and K_t depends upon the nature of the material and the notch geometry and size, and is attributable to a combination of factors including a statistical/size effect, the redistribution of stress due to plasticity and the stress gradient. Of these, particularly in the high-cycle regime, the most significant factor is probably the stress gradient through which any crack initiating at a notch must grow.

There have been many attempts to address this over the years. For example, Peterson proposed an empirical equation to describe the relationship between fatigue (K_f) and stress concentration (K_t) notch factors in terms of a notch sensitivity factor q as follows:

$q = \dfrac{K_f - 1}{K_t - 1}$ where $q = \dfrac{1}{1 + \dfrac{\alpha}{r}}$ and r is the notch root radius and α is a material property.

$$\alpha = 0.0254\left(\frac{2070\ MPa}{\sigma_u}\right)^{1.8} mm$$

This sort of approach is quite reasonable, but not very appropriate for application to FE-based fatigue calculations, where the loading and geometry may be quite complex.

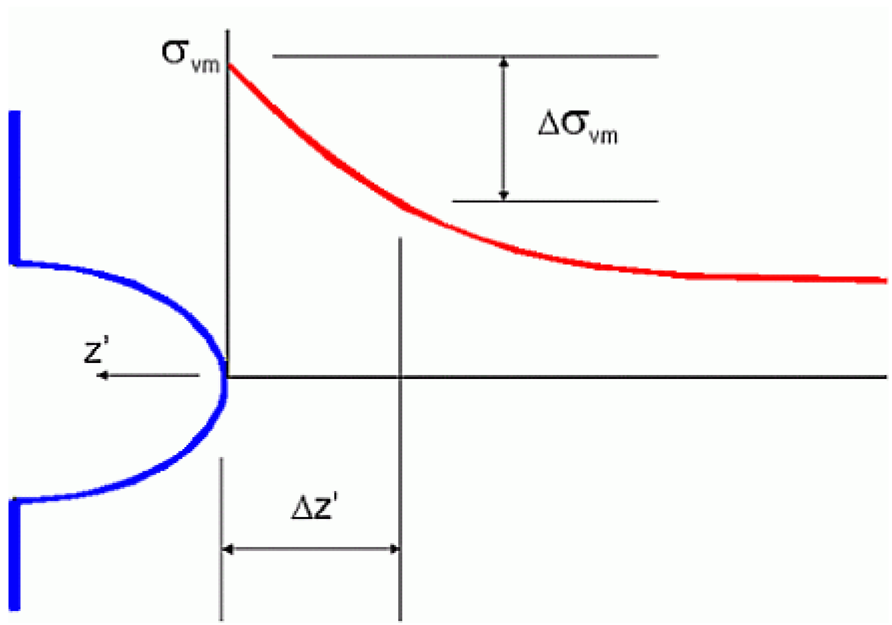
In DesignLife/nCodeDT, an alternative approach has been implemented based on the stress gradient approach described in the FKM Guideline "Analytical Strength Assessment of Components in Mechanical Engineering", Tr. E. Haibach. 2003. The FKM method describes a method in which the fatigue strength of a material is increased by a factor based on the surface normal stress gradient and the strength and type of material. In the general case of an FE-based fatigue calculation, there are six stress gradients (corresponding to the six components of stress) in the surface normal direction, and these can vary from moment to moment. We have therefore adapted this method so that rather than adjusting the fatigue strength, we adjust the stress before the fatigue calculation.

The procedure is as follows:

1. At the FE results file translation stage, in addition to the stresses being extracted (six components), the gradients of these stresses in the direction of the surface normal are also determined. See the section on FE Results for details of how the stress gradients are determined. If we define a local coordinate system x′y′z′ where z′ is an outward surface normal, we can denote the stress gradient: $\dfrac{\partial \sigma_{ij}}{\partial z'}$
2. From this data, the gradient of the von Mises stress in the surface normal direction is determined: $\dfrac{\partial \sigma_{vm}}{\partial z'}$
3. This stress gradient is normalized with respect to the von Mises stress at the surface to give the normalized von Mises stress gradient.

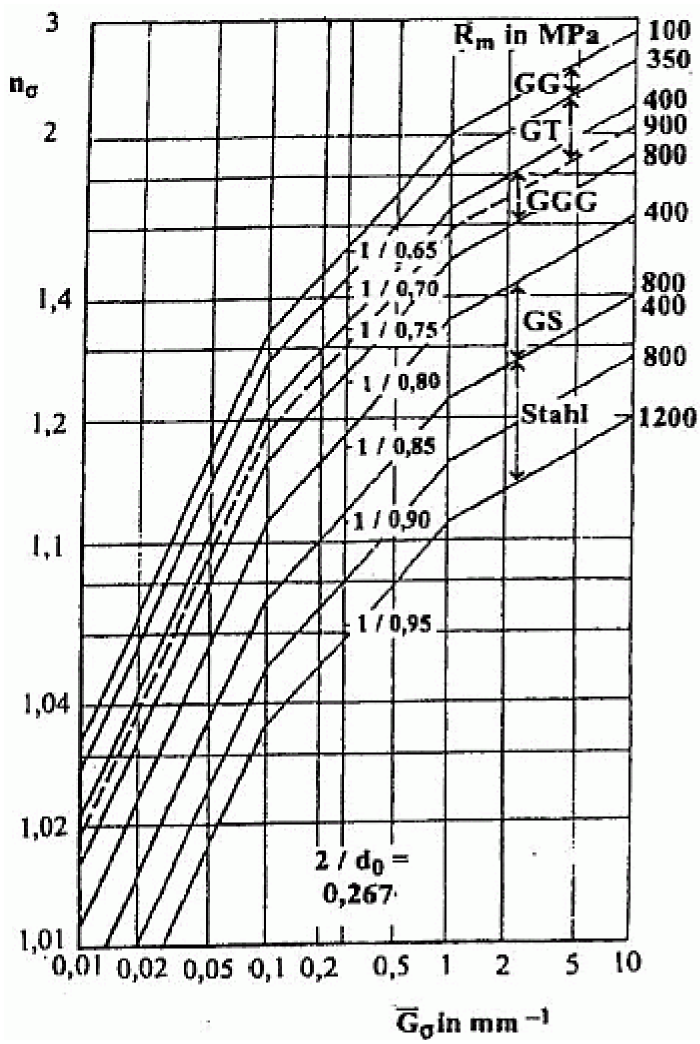
$$\overline{G}_{\sigma} = \left(\frac{\partial \sigma_{vm}}{\partial z'} \right) / \sigma_{vm}$$

Fig. 5-50 Determination of normalized stress gradient



The stress gradient is then used to determine a correction factor n_{σ}, based on the type and UTS of the material. These correction factors are illustrated in Figure 5-51 below.

Fig. 5-51 Stress gradient correction curves from FKM guideline



4. An effective time history is then calculated, before being passed into the usual stress combination, multiaxial assessment, and rainflow cycle counting routines. ${}^{effective}\sigma_{ij}(t) = \dfrac{\sigma_{ij}(t)}{n_\sigma(t)}$

The correction factors n_σ are calculated as follows:

For $G_\sigma \le 0, \quad n_\sigma = 1$

For $0 < G_\sigma \le 0.1, \quad n_\sigma = 1 + G_\sigma \cdot 10^{-\left(a_G - 0.5 + \frac{R_m}{b_G}\right)}$

For $0.1 < G_\sigma \le 1, \quad n_\sigma = 1 + \sqrt{G_\sigma} \cdot 10^{-\left(a_G + \frac{R_m}{b_G}\right)}$

For $1 < G_\sigma \le 100, \quad n_\sigma = 1 + \sqrt[4]{G_\sigma} \cdot 10^{-\left(a_G + \frac{R_m}{b_G}\right)}$

Please note that:

- Dimensions are in mm; stress gradients in mm^{-1}.
- Rm is the UTS (in MPa). The required constants are material dependent and are listed in the following table:

Table 5-7 Material Constants

Material	Stainless Steel	Other Steel	GS	GGG	GT	GG	Wrought Al Alloy	Cast Al Alloy
nCode mat type no.	23-25	13,14,16-22,26-99	9-12,15	5-8	2-4	1	100-105	106
a_G	.4	.5	.25	.05	-0.05	-0.05	0.05	-0.05
b_G	2400	2700	2000	3200	3200	3200	850	3200

The effect of a given stress gradient depends on the material which is defined by the *Material Type* code. This is usually specified by the entry in the *Materials Manager* database.

If the material type is undefined, then it will use Material Type = 99 (Steel of unknown heat treatment)

If the material properties have been generated from UTS then:

- Ferrous will use Material Type = 99 (Steel of unknown heat treatment)
- Aluminium will use Material Type = 100 (Wrought aluminium)

Note that *Other* and *Titanium* are not supported by the FKM method.

Alternatively, if your material does not match one of these types, it is possible to define the stress gradient correction using the *StressGradientsUser* option.

5.3.25 StressGradientsUser

If StressGradients is set to User, the name of a valid file must be provided here.

The user file is an ASCII file that provides a lookup table of G_σ vs n_σ. It must be in the following format:

```
Stress gradient correction factor file v1.0
Dimension=mm
#Normalized stress gradient G, correction factor n
StartCorrectionData
0,1
```

```
1,1.1
10,1.2
100,1.3
EndCorrectionData
```

Text after # on any line is ignored. Blank lines are ignored.

First line must be as above.

A Dimensions line must precede the data section.

The data section must be bound by StartCorrectionData and EndCorrectionData.

The options for Dimension are all supported units types for LENGTH.
(See `unitconv.sys` in the `nssys` directory of your installation.)

5.3.26 StressGradientMethod

This property has two possible values:

- VonMises - Stress gradient corrections are based on a scalar measure of stress (see StressGradients above). By default Von Mises stress is used.
- AbsMaxPrincipal - The stress gradient calculation can also be based on the absolute maximum principal stress. In this case the gradient of the AbsMaxPrincipal stress is determined in the surface normal direction and this is normalised with respect to the AbsMaxPrincipal stress at the surface.

5.3.27 OutputStressGradientFactors

This property has two possible values:

- True - The results output will include the normalised stress gradients and the notch sensitivity factors.
- False - The stress gradient parameters will not be output (Default).

5.3.28 Extracting Temperatures Cycle by Cycle

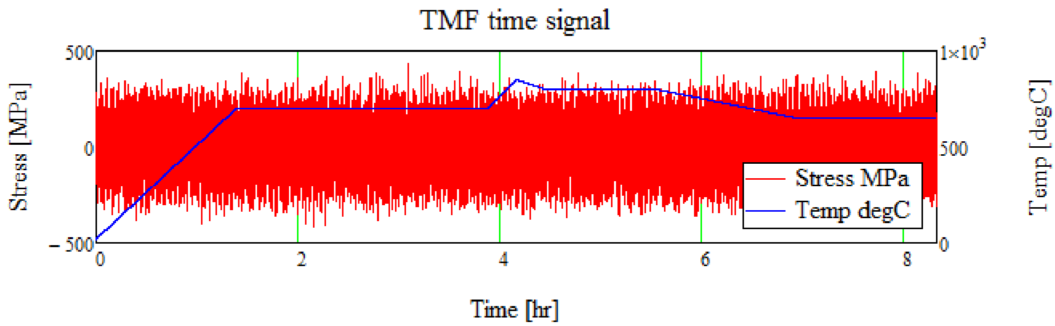
When performing a high-temperature analysis involving multiple temperature curves or the Chaboche (non-transient) method, DesignLife can calculate the damage for each cycle extracted using the rainflow cycle counter, based on the temperature during that cycle.

Clearly, if the temperature is constant across the stress history (either because the temperature is constant for the group or one of the constant methods has been applied on the SN Engine or the hybrid load provider), then this option will calculate the same result, with reduced performance.

However, if the temperature varies in time with the stress, then some cycles may have significantly different damage if their individual temperatures are used, compared to using an overall temperature across the whole history.

A good example of the use of this method would be an exhaust component that has a mechanical load operating across a relatively steady-state temperature.

Fig. 5-52 Stress across a steady state temperature plot



The current rainflow cycle algorithm used in DesignLife stores the start point and end point of each extracted fatigue cycle. In many cases, a single fatigue cycle can span a long period of time, for example, the Ground-Air-Ground (GAG) cycle for an aircraft. It is therefore important to make sure we have covered an adequate time span over which to establish the temperature variations.

Figure 5-53 (a) illustrates a basic approach using only the two points identified by a rainflow solver to form a cycle. In reality this approach only covers half the fatigue cycle so if the cycle were spread over a significant time period we could ignore some significant effects.

Figure 5-53 (b) shows a more realistic attempt to determine the extent of the cycle. This now includes the complete hysteresis loop. Alternative methods could take pre- and post-history into account (e.g.,Figure 5-54) but these are not included at this time as they are possibly too conservative (if using the maximum temperature across the cycle).

Fig. 5-53 Cycle extraction time periods—simple and extended

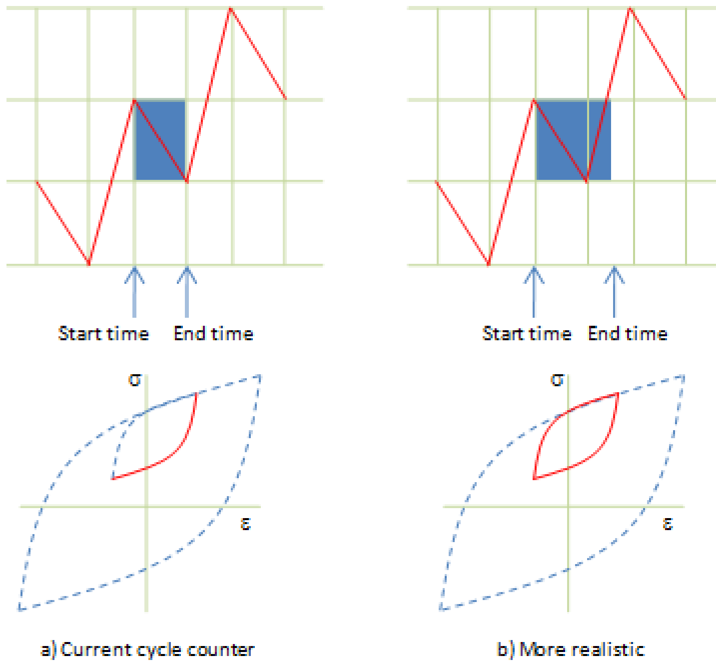
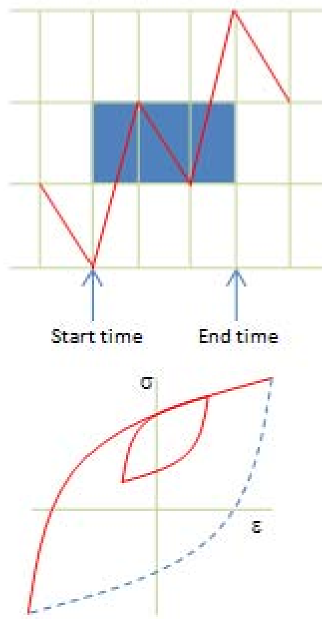


Fig. 5-54 Including pre-and post-history, considered too conservative



Once a time period (actually based on matching temperature and stress points) is established, then either the maximum or minimum temperature across the cycle can be extracted (by picking CycleMax or CycleMin as the TemperatureSelection property option) from the temperature history between the start point of the cycle and its estimated end point. To get the estimated end point, the nearest point in the temperature history is calculated by linear interpolation. If this option is used, the output cycles history contains the start point, end point, and estimated end point for each cycle, plus the extracted temperature.

Note that some cycles "wrap around" the end of the history so that their closure point appears to be before the start point. We assume that the signal is repeating and so this is a valid result, and is taken into account when finding the temperature.

In the case of combined duty cycles, when some cycles span events, those cycles take on the maximum temperature over the whole duty cycle—the code does not attempt to find the correct temperature across events.

5.4 Standard EN Analysis Engine

Fig. 5-55 Standard EN analysis engine properties

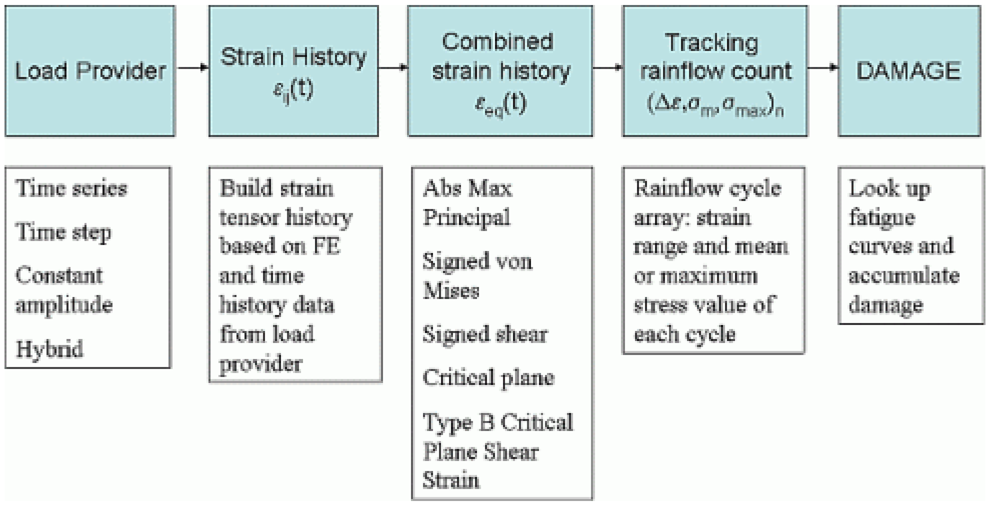
Object Name: ENEngine (Standard EN analysis engine)

Name	Value	Description
General		
ENMethod	Standard	The method used to calculate damage from a stress cycle
CombinationMethod	AbsMaxPrincipal	The method used to combine component stresses/strains
MeanStressCorrection	Morrow	The method used to correct the damage calculation for mean stress
InterpolationLimit	UseMaxCurve	Multicurve material interpolation limit
MultiAxialAssessment	Standard	Whether to perform assessment of the multi-axial stress state
ElasticPlasticCorrection	HoffmannSeeger	The correction method for elastic plastic transformation in biaxiality
PlasticLimitLoadCorrection	None	Method used to include shape effect in Elastic Plastic Correction
CertaintyOfSurvival	50	Required certainty of survival (%) on damage results
ScaleFactor	1	The scale factor to apply prior to damage calculation
OutputMaxMin	False	Whether to output max and min stresses
OutputMaterialNames	False	Whether to output material names to the results
OutputDistributedSource	False	Whether to output details of the distributed process that generated each result
BackCalculation		
BackCalcMode	None	Whether to perform a back-calculation or not
TargetDamage	1E-6	Target damage for back calculation
BackCalcAccuracy	1	The accuracy of the back calculation
BackCalcMaxScale	5	The max scale factor to allow in back calculation
BackCalcMinScale	0.2	The min scale factor to allow in back calculation
DutyCycle		
EventProcessing	Independent	How to process separate events in duty cycles
OutputEventResults	False	Whether to output results per event or not for duty cycle processing
Advanced		
SWTMethod	Iterative	The Smith-Watson-Topper calculation method
DamageCalcMethod	LookupTable	The method used to calculate the damage for each cycle
LookupTableSize	256	The size of the lookup table used for the damage calculation
CheckStaticFailure	Warn	The action to take on static failure
DamageFloor	1E-20	The calculated damage value below which the damage is set to zero
MaxDamage	1E30	The maximum damage value
StaticFailureDamage	1E10	The damage value to insert on static failure
StressGradients		
StressGradients	Auto	Controls whether stress gradient corrections are done
StressGradientsUser		The name of the file that contains the stress gradient parameters
StressGradientMethod	VonMises	The measure of stress used in stress gradient calculations
OutputStressGradientFactors	False	Whether to output stress gradient factors
Vibration		
OutputVibrationStats	False	Whether to output Vibration PSD parameters such as ExpectedZeroUpcrossings and Ex
VibrationLoopMultiplier	1	The factor by which vibration RMS stress is multiplied when determining cycle centres

5.4.1 Summary - Standard EN steps

The basic operation of the standard EN analysis engine can be summarized in the following diagram:

Fig. 5-56 Basic E-N analysis engine steps

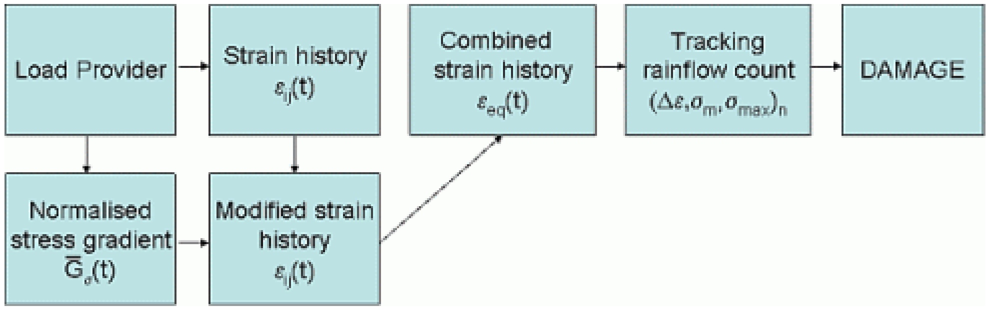


A typical E-N analysis has the following basic steps, which are executed for each analysis entity (Node, Element, Node-on-Element), shell surface and critical plane where appropriate.

1. The strain tensor history, as a function of time or data points, is assembled from the information provided in the load provider. This applies to time-series, time step, constant amplitude, duty cycle and hybrid load providers. See the section on Loading for details. In most cases, the strains are linear elastic strains at this point.
2. From the strain tensor (still elastic), extract a combined strain parameter (Abs Max Principal, Signed von Mises, Signed Shear, Critical Plane or Type B Critical Plane Shear Strain).
3. Rainflow count the combined strain history to obtain a list of rainflow cycles, at the same time estimating the total local strain (elastic-plastic) using a notch rule (e.g., Neuber's rule) and tracking the shape and position of each hysteresis loop. This procedure is called a tracking rainflow count and is described in more detail later. This more complex rainflow procedure is necessary because the local strain approach requires that we know the total (elastic-plastic) strain range for each cycle, and often we also require the mean or maximum stress of each cycle, too.
4. Calculate and accumulate damage from E-N curve definitions, using linear damage accumulation (Miner's rule).

When the stress gradient correction is used, the process is modified slightly, as follows:

Fig. 5-57 E-N analysis process with stress gradient correction



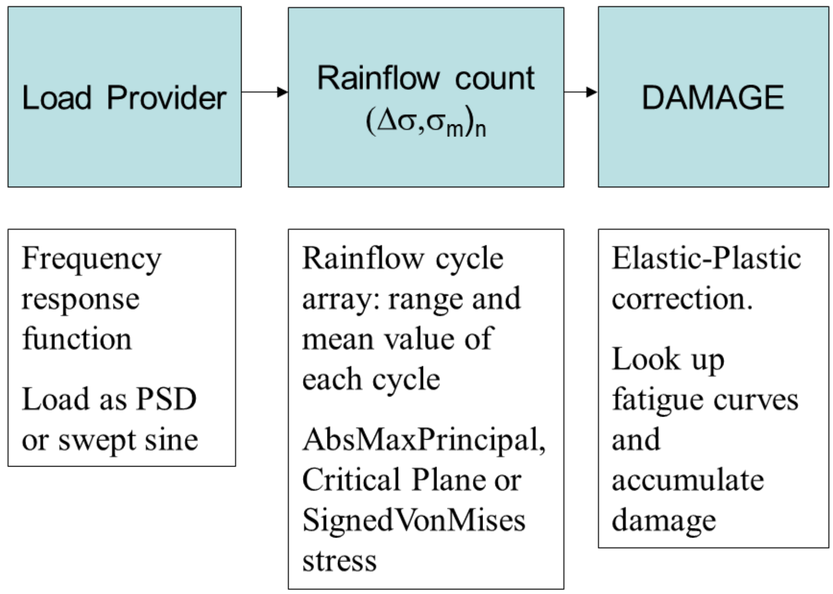
Now the load provider also provides the information necessary to calculate a normalized von Mises stress gradient and this information is used to modify the strain tensor history to take into account the strain gradient effect. (Actually, we use the normalized von Mises strain gradient, but because it is normalized, and because von Mises stress is proportional to von Mises strain, the end result is the same.) The strain tensor history is then processed as before.

When the load provider is a Vibration Load Provider, the process is rather different, in that a time history is not generated. The rainflow count is generated directly, based on the PSD, Multi-PSD, swept-sine, sine dwell or sine-on-random loading and the Frequency Response Function (FRF). Actually, because what is really being predicted is the probability of a rainflow cycle occurring (probable rainflow count per second), fractional cycles are possible. This is discussed in more detail in the section describing the Vibration Load Provider.

The rainflow count from the vibration load provider is elastic and because EN analysis requires elastic-plastic (total) strain and stresses an elastic-plastic correction is necessary. For this elastic-plastic correction, all cycles are centred on the mean stress and mean strain of a loop based on the RMS nominal Stress. The stress range of this loop is centred on zero (or the static offset stress, if any); its amplitude is the RMS stress multiplied by VibrationLoopMultiplier.

Where plastic effects are negligible, this results in all cycles being centred about zero stress and strain (or the static offset stress and corresponding strain). Where plastic effects are not negligible, they affect the mean strain of the loop, so that cycles are centred about a strain value that is influenced by VibrationLoopMultiplier.

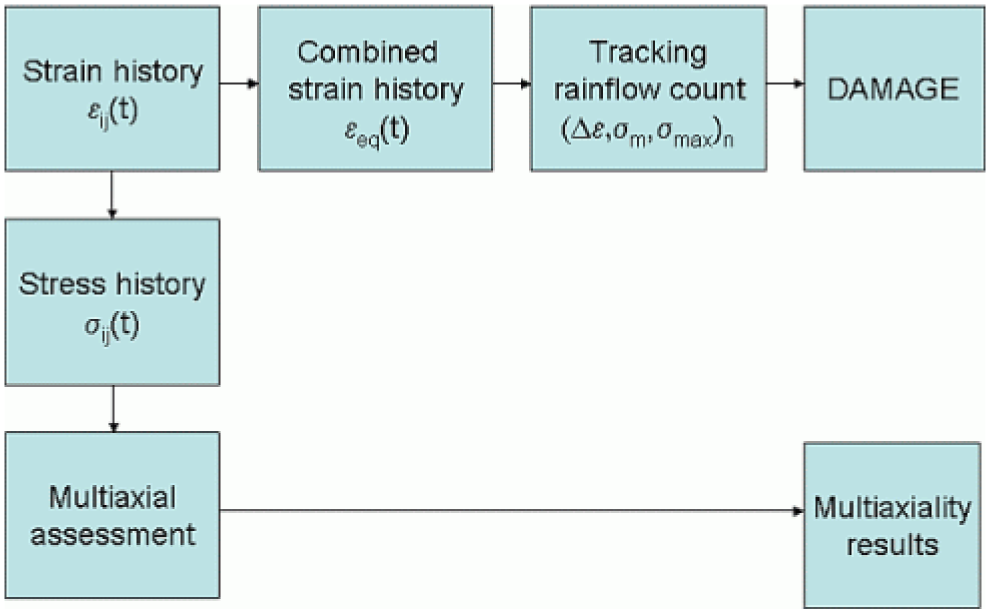
Fig. 5-58 E-N fatigue analysis process with Vibration Load Provider



5.4.2 Summary - Multiaxial Assessment steps

When a multiaxial assessment is requested (other than the *Auto* option), the multiaxial assessment is based on the stress tensor, which is recalculated, from the strain tensor (after stress gradient correction where applied). If the *Hoffmann-Seeger* method is being used, the results of the multiaxial assessment are required for the tracking rainflow counter.

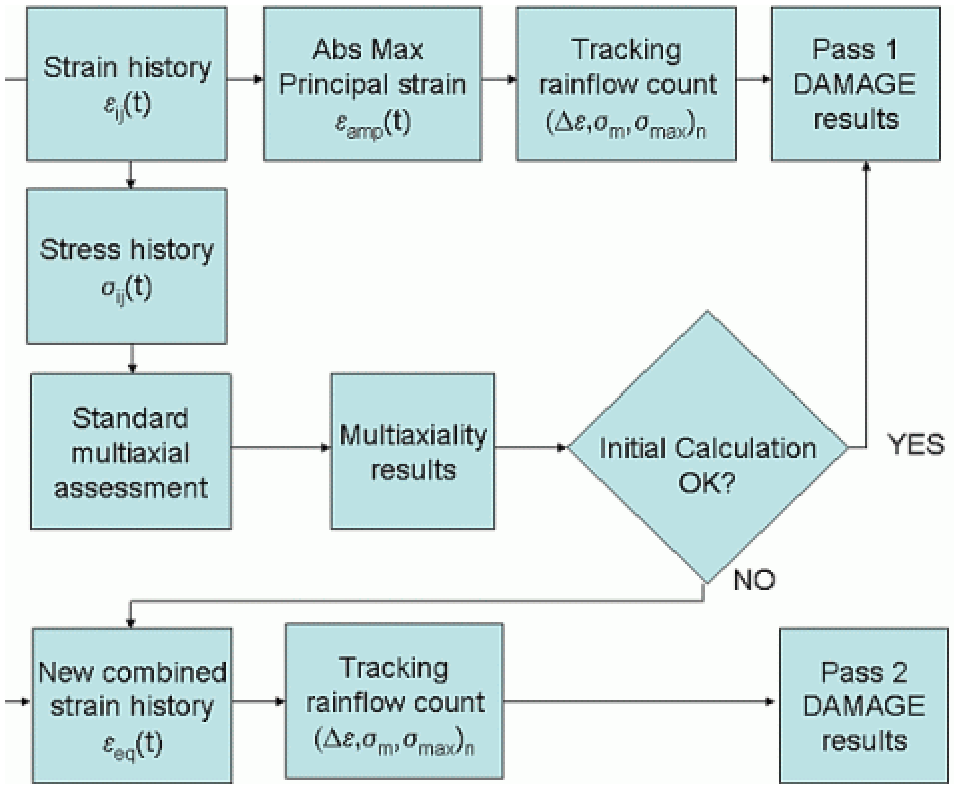
Fig. 5-59 E-N analysis with multiaxial assessment



Auto mode

The *Auto* mode employs a two-pass approach. In the first pass, a calculation is made using the Absolute Maximum Principal Strain, and a Standard multiaxial assessment is also carried out. Based on the multiaxial assessment results, a second pass calculation may be made using a different combined strain option (e.g., critical plane). Where a second pass calculation is made, its results overwrite the first.

Fig. 5-60 E-N calculation process with Auto multiaxial assessment



The handling of duty cycles is discussed later.

5.4.3 Note on Stresses and Strains

E-N calculations are normally based on linear elastic FE. Note that they may be based on linear stresses or linear strains from FE. If stresses σ_{ij} are used, they have to be converted to linear strains ε_{ij} at the beginning of the calculation, using Hooke's Law.

$$\varepsilon_{xx} = \frac{\sigma_{xx}}{E} - \frac{\nu_e}{E}\left(\sigma_{yy} + \sigma_{zz}\right)$$

$$\varepsilon_{yy} = \frac{\sigma_{yy}}{E} - \frac{\nu_e}{E}\left(\sigma_{zz} + \sigma_{xx}\right)$$

$$\varepsilon_{zz} = \frac{\sigma_{zz}}{E} - \frac{\nu_e}{E}\left(\sigma_{xx} + \sigma_{yy}\right)$$

$\varepsilon_{xy} = \frac{\gamma_{xy}}{2} = \frac{\sigma_{xy}}{2G}$ where $G = \frac{E}{2(1+\nu_e)}$ etc.

G is the shear modulus and ν_e is the elastic Poisson ratio.

The Elastic Modulus and Poisson ratio should match those used in the FE, especially if the calculation is based on elastic FE strains.

Note that when 2-D linear strains are obtained from shell elements, the out-of-plane z-strains are frequently not calculated by the FE code. However, it is not correct to assume that these are zero. In these circumstances, the out-of-plane strain is calculated by nCodeDT from the in-plane strains and the Poisson ratio as follows:

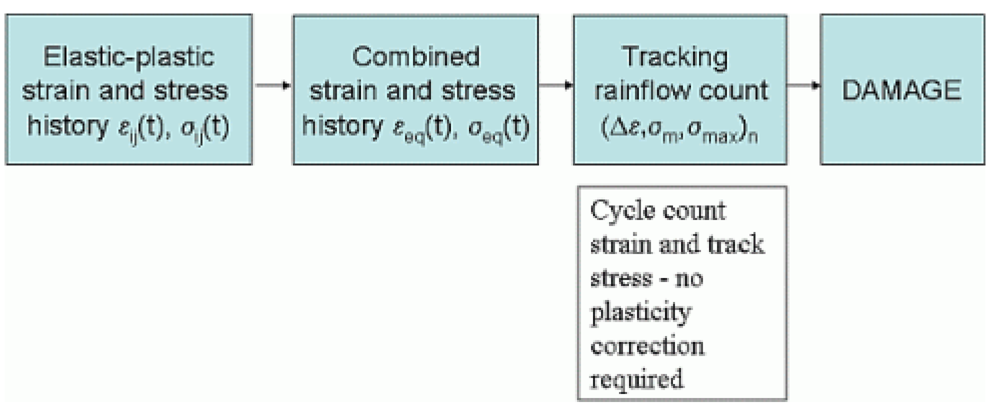
$$\varepsilon_{zz} = \frac{-\nu_e(\varepsilon_{xx} + \varepsilon_{yy})}{1-\nu_e}$$

Translation will have taken the full stress or strain tensor and rotates it to the plane of the free surface (if resolve to local was selected).

- If the data is flagged as 2D and is STRESS, the out of plane stress components will be set to zero (xz, yz, zz) i.e. a state of plane stress will be forced. This modified stress tensor is then converted to strain.
- If the data is flagged as 2D and is STRAIN, the out of plane strain components xz and yz will be set to zero and the zz component recalculated from the x and y components and the poisson ratio.

Alternatively, non-linear (elastic-plastic) stresses and strains may be used. In this case, both the stress and strain are used in the calculation directly, without the need for calculation of hysteresis loops.

Fig. 5-61 Summary of calculation process based on elastic plastic strains and stresses



In all cases, the stresses or strains may be 3-D (typically in global coordinates) or 2-D (as you might expect from shell elements or after surface resolution).

To summarize the possibilities:

Table 5-8 Options for Stress and Strain Input for E-N Analysis Engine

2-D/3-D	Stress/strain	Comments
2-D	Linear stress	Convert to strain before cycle counting and plasticity (notch) correction.
	Linear strain	Calculate out-of-plane strain.
	Stress & strain	No plasticity correction required.
3-D	Linear stress	Convert to strain.
	Linear strain	
	Stress & strain	No plasticity correction required.

5.4.4 Standard Strain-life Material Properties

The standard strain-life material properties, which can be found in the MXD material database, consist of a set of generic properties (which can be inherited from a generic parent) and a set of specific properties that define the shapes of the stress-strain and strain-life curves used by the E-N analysis engine.

The generic properties are as follows:

Parameter Name	Description
MaterialType	Numeric code defining the type of material. The material type is required for correct application of surface finish and stress gradient corrections.
UTS	Ultimate tensile strength. This is required for the correct definition of the upper part of the S-N curve, and to apply the static failure criterion.
YS	Yield stress. Optional (not currently used).
E	Modulus of elasticity. Required for S-N when the FE results are elastic strain.
me	Elastic Poisson ratio (defaults to 0.3 if not defined)
mp	Plastic Poisson ratio (defaults to 0.5 if not defined)
Comments	
References	

The specific E-N properties are as follows:

Parameter Name	Description
Sf'	Fatigue strength coefficient, σ_f'
b	Fatigue strength exponent
c	Fatigue ductility exponent
Ef'	Fatigue ductility coefficient, ε_f'
n'	Cyclic hardening exponent
K'	Cyclic strength coefficient
n'90	Cyclic hardening exponent for 90-degree out-of-phase multi-axial loading. This is not used and therefore not required for standard EN calculations.
K'90	Cyclic hardening coefficient for 90-degree out-of-phase mul-tiaxial loading. This is not used and therefore not required for standard EN calculations.
Nc	Fatigue cutoff. Damage is set to zero beyond this point. It is set in REVERSALS. 2 reversals = 1 cycle
SEe	Standard error of log(plastic strain)
SEp	Standard error of log(elastic strain)
SEc	Standard error of log(cyclic strain)
Ne	Endurance limit – in reversals
FSN	Fatemi-Socie parameter (not currently used)
S	Wang-Brown parameter. This is not used and therefore not required for standard EN calculations.

The properties are used to define two basic relationships, as follows:

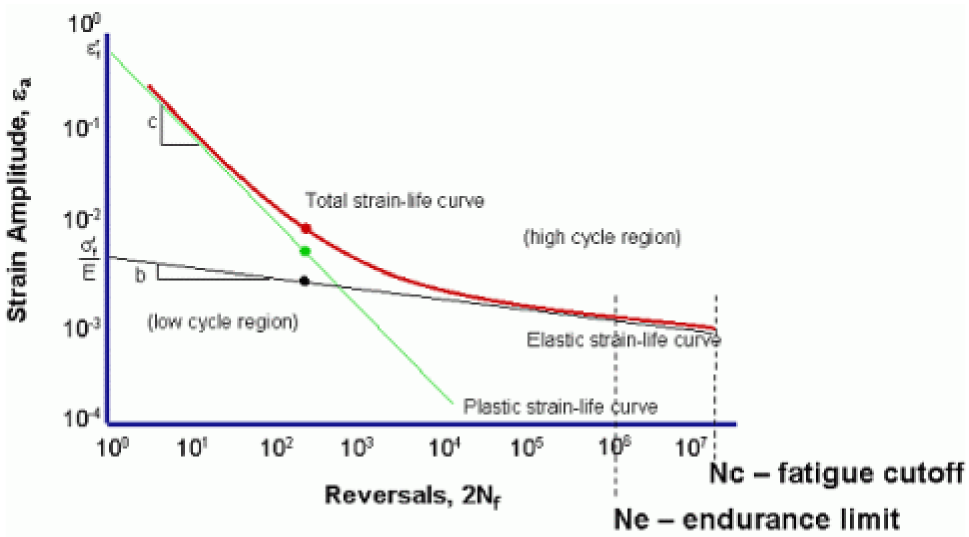
Strain-life Relationship

This is the Coffin-Manson-Basquin formula, defining the relationship between strain amplitude ε_a and the number of cycles to failure N_f:

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

Graphically, it looks like this:

Fig. 5-62 Coffin-Manson-Basquin strain-life relationship

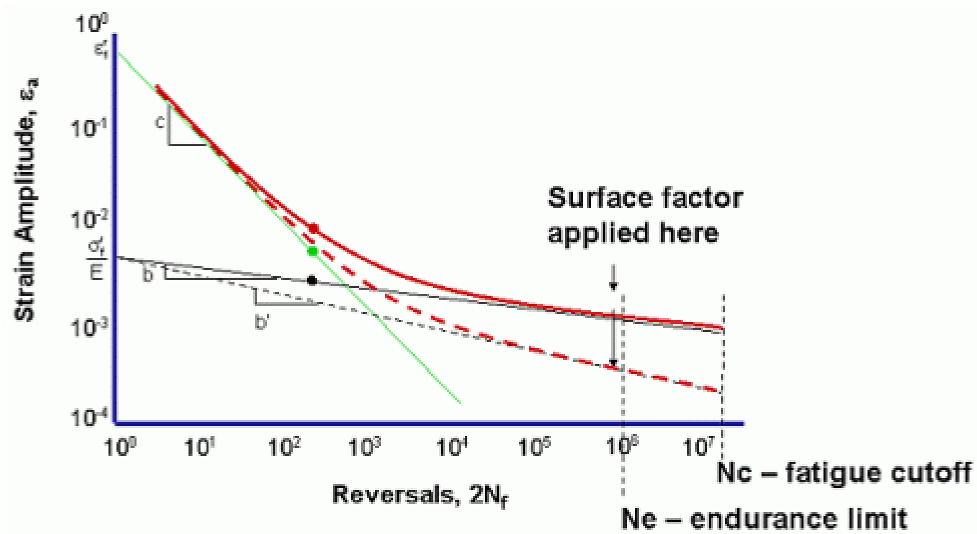


Fatigue damage is predicted in the same way as for the S-N method—the damage due to an individual cycle is calculated by looking up the strain amplitude of that cycle on the curve to find the number of reversals to failure $2N_f$. The damage assigned to that cycle is then $1/N_f$.

If the life N_f corresponding to a particular cycle is greater than the cutoff Nc, zero damage is predicted. Ne is an endurance limit. The main application for this is in the consideration of surface finish.

The effect of surface condition in DesignLife is modeled by means of a surface factor K_{sur} which, in the case of a rough surface finish, reduces the fatigue strength. The derivation of surface factors is described in "Surface Finish and Surface Treatment Settings" on page 60. In the case of the E-N method, the effect of the surface condition is modeled by changing the slope of the elastic part of the strain-life curve so that the fatigue strength at the endurance limit Ne is reduced by the surface factor.

Fig. 5-63 Application of surface factors in the local strain approach



If the surface factor is denoted K_{sur}, the modified slope b′ is calculated from:

$$b' = b - \frac{\log(K_{sur})}{\log(Ne)}$$

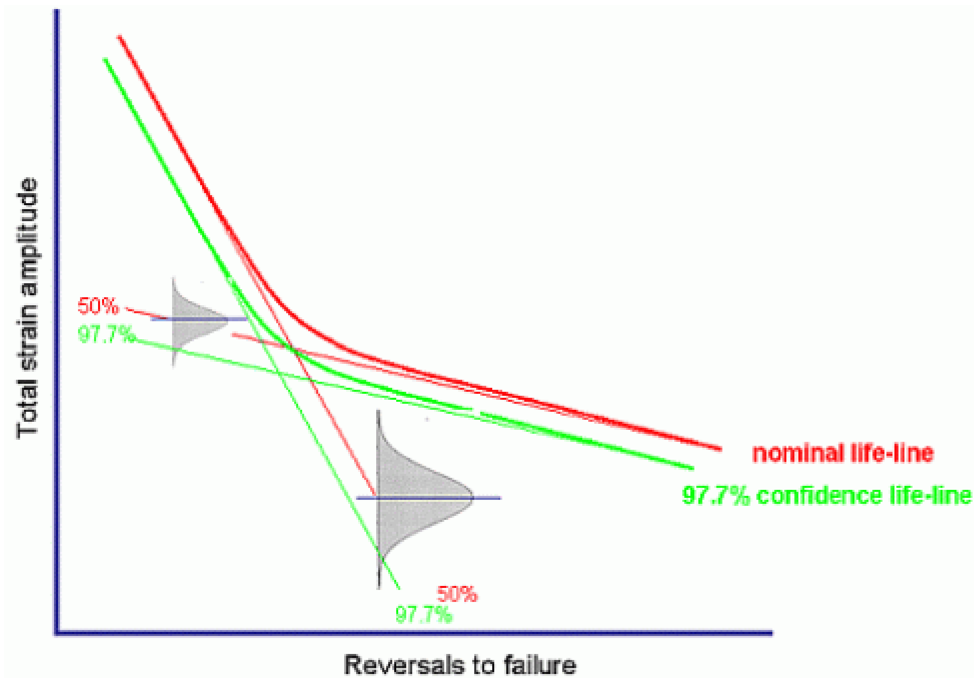
Note that if Ne is not defined, the fatigue cutoff Nc will be used instead.

The standard error parameters are used to determine the life associated with a particular design criterion (certainty of survival). Referring to Table 5-1 (in the S-N Analysis section), the desired number of standard deviations z from the mean is determined. For example, if a design criterion of 97.7% is desired, z = -2.

The strain-life curve is then shifted downwards as follows:

$$\varepsilon_a = \frac{\sigma_f'}{E}(2N_f)^b \cdot 10^{z \cdot SEe} + \varepsilon_f'(2N_f)^c \cdot 10^{z \cdot SEp}$$

Fig. 5-64 Adjustment of strain-life curve to account for design criterion



Stress-strain Relationship

The local strain approach also requires that the cyclic stress-strain behavior be modeled. This requires that we know the relationship between stress amplitude and strain amplitude (cyclic stress-strain curve), and how to predict the shape of hysteresis loops. The uniaxial cyclic stress-strain curve is modeled using a Ramberg-Osgood relationship.

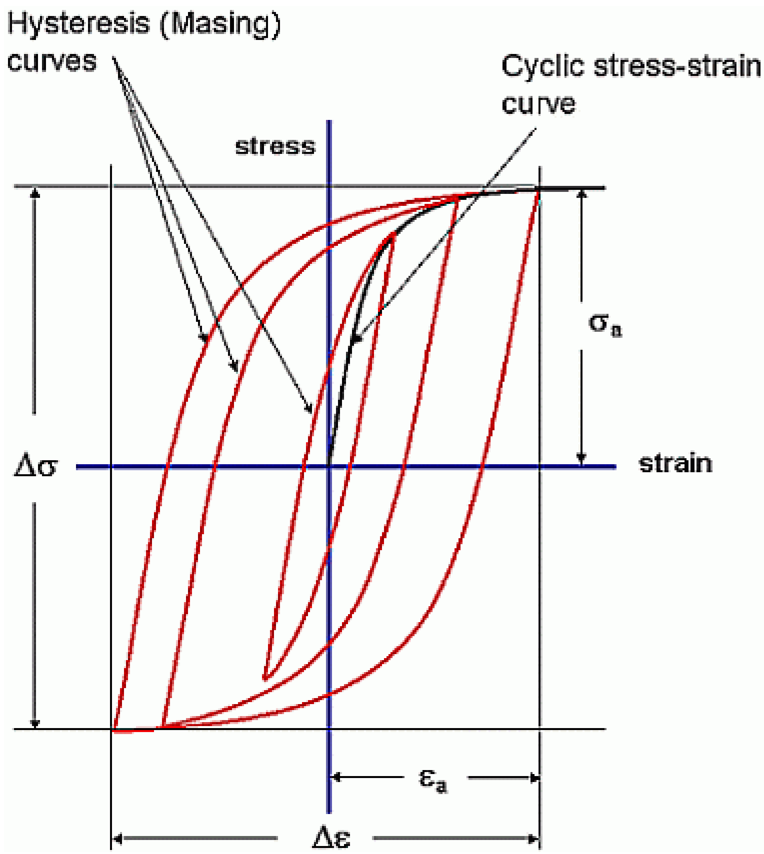
$$\varepsilon_a = \frac{\sigma_a}{E} + \left(\frac{\sigma_a}{K'}\right)^{\frac{1}{n'}}$$

Masing's hypothesis is used to predict the shape of each hysteresis loop by doubling the size of the cyclic stress-strain curve.

$$\Delta\varepsilon = \frac{\Delta\sigma}{E} + 2\left(\frac{\Delta\sigma}{2K'}\right)^{\frac{1}{n'}}$$

The relationship between the two curves is illustrated in Figure 5-65 below.

Fig. 5-65 Cyclic stress strain and hysteresis curves



If SEc > 0, then the design criterion (certainty of survival) will also adjust the stress strain behavior as follows, based on the number of standard errors from the mean:

$$\Delta\varepsilon = \frac{\Delta\sigma}{E} + 2\left(\frac{\Delta\sigma}{2K'}\right)^{\frac{1}{n'}} \cdot 10^{-z \cdot SEc}$$

The effects of this adjustment can be counter-intuitive, particularly when the Smith-Watson-Topper method is being used. It is probably best to leave SEc set to 0.

5.4.5 Gray Iron Strain-life Material Properties

The gray iron data set consists of the following properties:

Parameter Name	Description
UTS	Ultimate tensile strength. This is required to apply the static failure criterion.
E	Elastic Modulus (MPa). As used in the FE analysis
me	Elastic Poisson's Ratio. As used in the FE analysis
E0	Tangent modulus at zero
nt	Tensile strain hardening exponent
Kt	Tensile strain hardening coefficient
Nc	Compressive strain hardening exponent
Kc	Compressive strain hardening coefficient
mc	Compressive secant slope
mu	Unloading secant slope
Ks	Damage curve co-efficient
ns	Damage curve exponent
RC	Damage curve cutoff
SD	Standard error of log life
mt	Tensile secant slope
Comments	Comments.
References	References.

This data set is typically a child of a generic data set. See GlyphWorks Fatigue Theory >*3.2.1 Materials Data Types: Standard Parametric Data.*

5.4.6 Gray Iron Material Behavior

Gray irons have significantly different stress-strain behavior to most other alloys, due to the weakness of the graphite flakes in tension. In order to determine the properties, standard tests are performed at a variety of strain levels and life is noted.

The tensile cyclic stress strain curve is defined as

$$\varepsilon_t = \frac{\sigma}{E_0 + m_t\sigma} + \left(\frac{\sigma}{K_t}\right)^{1/n_T}$$

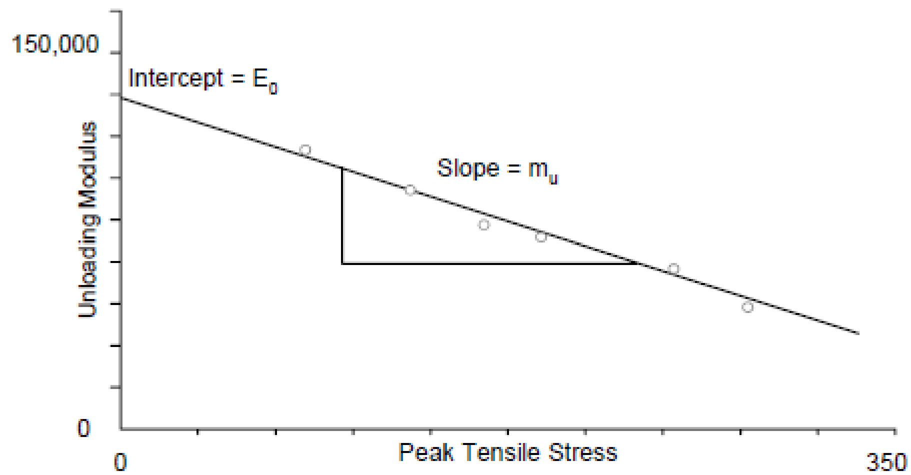
Total strain amplitude is the sum of the secant strain and the residual non-elastic strain.

The unloading modulus Eu is related to maximum tensile stress according to

$E_u = m_u S_{max} + E_0$

For each strain amplitude, plot unloading modulus vs. peak tensile stress. Linear regression will provide a slope, mu, and intercept Eo.

Fig. 5-66 Grey iron unloading modulus v peak tensile stress

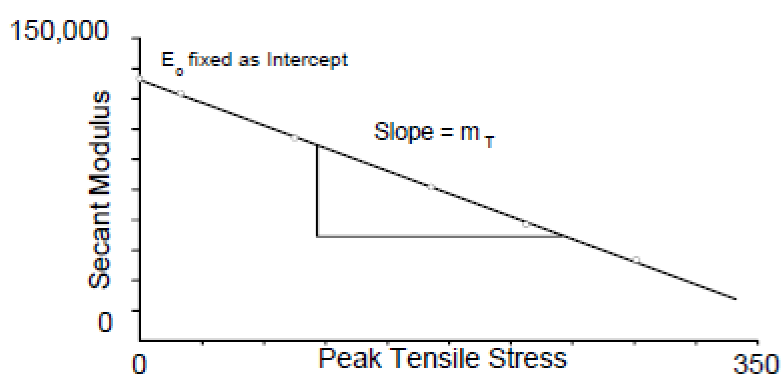


The secant modulus is related to maximum tensile stress according to

$$E_s = \frac{\sigma_{max}}{\varepsilon_a} = m_t \sigma_{max} + E_0$$

For each strain amplitude, divide the initial peak tensile stress by the strain amplitude to get the secant modulus and then plot these moduli against the peak stress. Regression analysis through the linear portion of the plot, including the value of E_0 calculated above, will give a slope of m_t, the tensile secant slope.

Fig. 5-67 Secant modulus v peak tensile stress



Secant strain is calculated from

$$\varepsilon_s = \frac{\sigma}{(E_0 + m_t\sigma)}$$

and the remaining non-elastic strain from

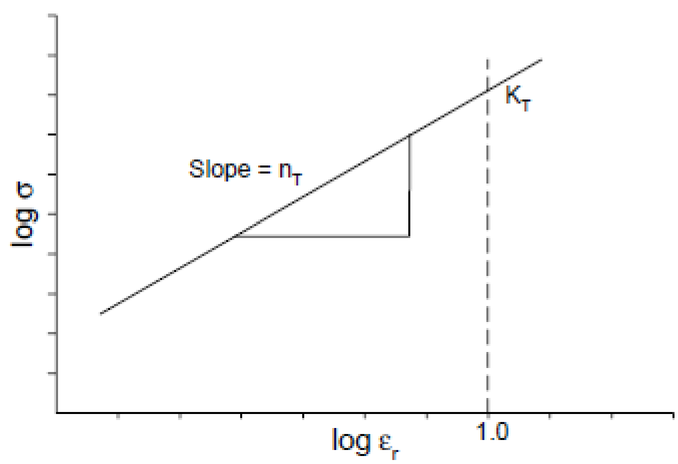
$$\varepsilon_T = \varepsilon_t - \varepsilon_s$$

K_t and n_t are related to residual non-elastic strain

$$\varepsilon_T = \left(\frac{\sigma}{K_t}\right)^{1/n_t}$$

Log-log regression of remaining non-elastic strain vs. peak stress will provide values for K_t and n_t.

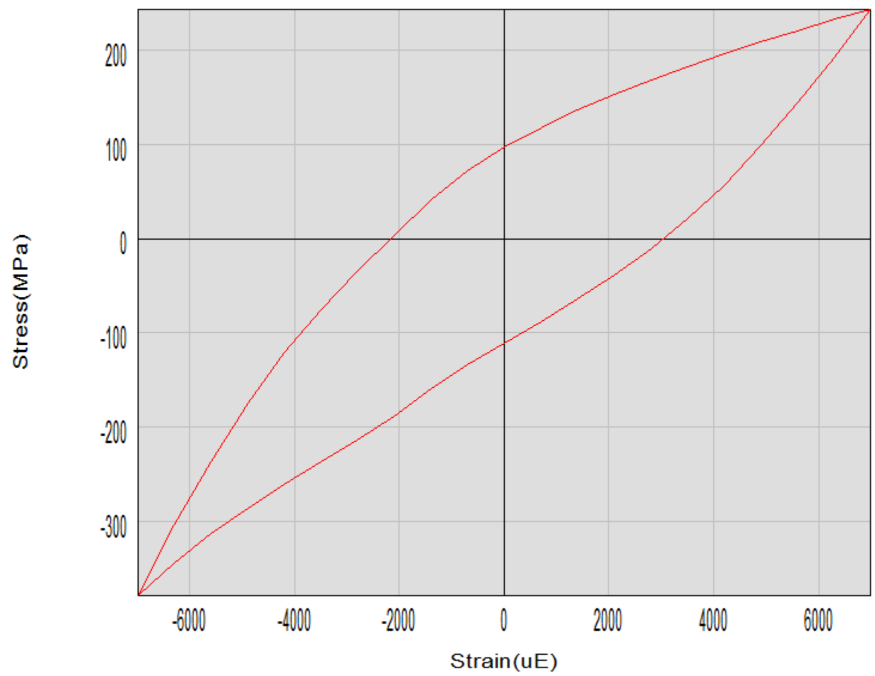
Fig. 5-68 Log-log regression of remaining non-elastic strain vs. peak stress



The compressive cyclic stress strain properties m_c, K_c, and n_c are determined in the same way as for the tensile equivalents.

A typical hysteresis loop for gray iron using these equations looks like this:

Fig. 5-69 Hysteresis loop for gray iron



Note that the curve shape changes in loading and unloading and in tension and compression.

The damage curve is determined by log-log regression of the SWT parameter against life to failure. The equation is

$$\sigma_{max}\varepsilon_a = A(N_f)^b$$

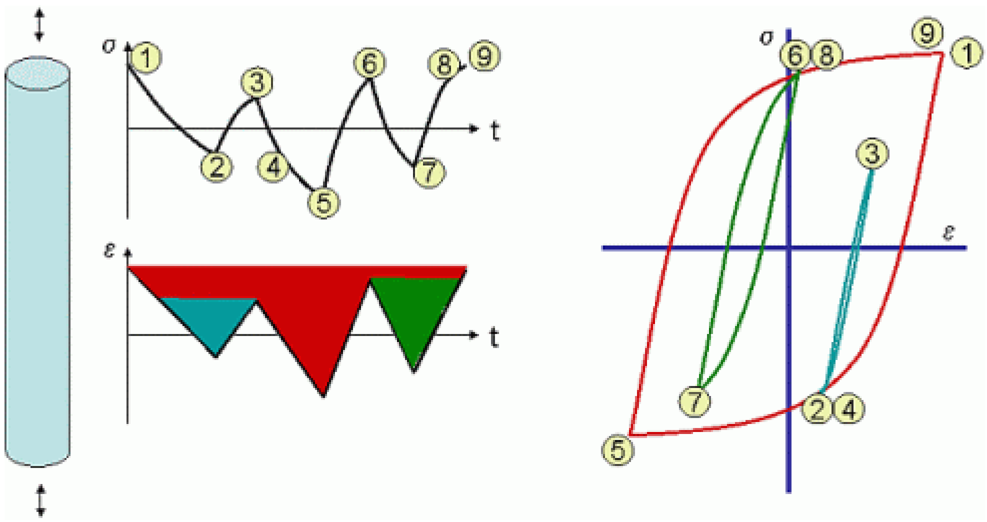
This gives the values of A (Ks in the data set) and b (ns in the data set).

5.4.7 Note on Rainflow Cycle Counting for Strain-life Calculations

Rainflow cycle counting for the local strain (E-N) approach is based on the same principles as for the S-N method, but there are additional requirements in that we need to know the total (elastic-plastic) strain range for each cycle, together with its mean or maximum stress.

Consider the case of a simple bar, subjected to a variable amplitude loading under uniaxial conditions, and suppose we can determine the total strain and stress histories. If we plot stress against strain, the stress-strain history will appear as a number of nested hysteresis loops, each corresponding to a rainflow cycle, as illustrated in Figure 5-70.

Fig. 5-70 Hysteresis loop formation



Each excursion takes the shape of part of the hysteresis curve.

Starting at point 1, we follow the hysteresis curve as far as point 2. At this point the strain direction reverses, so we start again from point 2 up to 3, and from point 3 we reset to the beginning of the hysteresis curve again until point 4. 2-3-4 represents a closed rainflow cycle. At point 4, the well-known material memory effect has to be considered. The material appears to "remember" that it was interrupted after starting from point 1, so when it gets to point 4, it resumes along the hysteresis curve starting from 1 until it reaches the next turning point at 5.

By continuing in this way, we will find that the entire history can be tracked and reduced to a number of hysteresis loops that can be identified with rainflow cycles. For each cycle, we can determine the important parameters we need for a fatigue calculation, namely the strain range and mean or maximum stress.

However, FE-based E-N fatigue calculations are very often based on elastic FE results, so that the equivalent strain history generated by the E-N analysis engine based on the load provider is a pseudo-elastic strain history. In reality, there may be plasticity, especially at critical locations, and this needs to be estimated if realistic life predictions are to be obtained. In addition, the stress state may not be uniaxial. In these circumstances, the tracking rainflow cycle counting procedure has to be modified to include a notch correction procedure such as the Neuber or Hoffmann-Seeger methods, to estimate the total strain range of each cycle, and to allow the resulting hysteresis loops to be correctly positioned.

The Neuber and Hoffmann-Seeger methods are described later.

The steps in the calculation are as follows:

1. Reduce the elastic strain history to a peak-valley sequence, and re-order it to start from the Abs Max value, with the first point repeated at the end.
2. Position this point in stress-strain space using the cyclic stress strain curve and using a notch correction method such as the Neuber method to estimate the elastic-plastic stress and strain.
3. From this point, calculate the next excursion using the hysteresis curve and the Neuber correction again.
4. Repeat this process. When a cycle or cycles close, remove these points from the sequence, noting the calculated total strain range, mean and maximum stress for the cycle, and reset the starting point for the next strain excursion to the last remaining point.
5. Continue until the end of the history is reached and all cycles have closed.

The following sections describe in detail the settings of the EN analysis engine.

5.4.8 ENMethod

There are five options here:

- Standard
- MultiMeanCurve
- MultRRatioCurve
- MultiTemperatureCurve
- GrayIron

Standard

This option uses the standard Coffin-Manson-Basquin strain life curve formulation, as previously described. This formulation can be readily adapted to include the effects of mean stress using the Morrow and SWT methods (described below).

A few companies, notably in the aerospace industry, do not like to rely on the generic Morrow and SWT methods for modeling the effect of mean/max stress, but prefer to take mean effects into account by interpolation between curves obtained at different mean or load ratios. The next two options are for these companies:

MultiMeanCurve

When MultiMeanCurve is selected, the E-N analysis engine expects the E-N data in the form of a family of E-N curves representing the fatigue strength of the material at different mean stress levels (nCode EN Mean Stress Curves). This type of data is found only in the MXD database (not supported in the nSoft MDB database). The data consists of a set of generic data with a number of child datasets storing the individual curves. The generic data consists of the following parameters:

Parameter Name	Description
MaterialType	Numeric code defining the type of material. The material type is required for correct application of surface finish and stress gradient corrections.
UTS	Ultimate tensile strength. This is required to apply the static failure criterion.
E	Modulus of elasticity
n'	Cyclic hardening exponent
K'	Cyclic hardening coefficient
me	Elastic Poisson ratio
SEc	Standard Error of Cyclic Strain
Ne	Endurance limit
Nfc	Fatigue cutoff
Strain Type	Range or Amplitude
Comments	
References	

Each child dataset (E-N curve) defines a lookup curve of strain vs. life and has the following parameters:

Parameter Name	Description
MeanStress	Mean Stress for this strain-life curve
StrainValues	Comma-separated list of strain values
LifeValues	Comma-separated list of life values (in reversals)

These curves are normally used by interpolating between the curves for each cycle depending on its mean stress. The interpolation scheme is the same as for “MultiMeanCurve” on page 96 in the Standard S-N Analysis engine. Please see the appropriate section for details.

Note that the effect of design criterion and surface condition are not currently supported for this data type.

MultiRRatioCurve

The handling of MultiRRatioCurve data is very similar to that of MultiMeanCurve data. See “MultiMeanCurve” on page 96.

When MultiRRatioCurve is selected, the E-N analysis engine expects the E-N data in the form of a family of E-N curves representing the fatigue strength of the material at different R-ratio levels (nCode EN R-ratio Curves). This type of data is found only in the MXD database (not supported in the nSoft MDB database).The data consists of a set of generic data with a number of child datasets storing the individual curves.

The generic data consists of the following parameters:

Parameter Name	Description
MaterialType	Numeric code defining the type of material. The material type is required for correct application of surface finish and stress gradient corrections.
UTS	Ultimate tensile strength. This is required to apply the static failure criterion.
E	Modulus of elasticity
n'	Cyclic hardening exponent
K'	Cyclic hardening coefficient
me	Elastic Poisson ratio
SEc	Standard Error of Cyclic Strain
Ne	Endurance limit
Nfc	Fatigue cutoff
Strain Type	Range or Amplitude
Comments	
References	

Each child dataset (E-N curve) defines a lookup curve of strain vs. life and has the following parameters:

Parameter Name	Description
R-Ratio	R-ratio for this strain-life curve
StrainValues	Comma-separated list of strain values
LifeValues	Comma-separated list of life values

These curves are normally used by interpolating between the curves for each cycle depending on its R-ratio stress. The interpolation scheme is the same as for "MultiRRatioCurve" on page 100 in the Standard S-N Analysis engine. Please see the corresponding section for details.

Note that the effect of design criterion and surface condition are not currently supported for this data type. See "MultiRRatioCurve" on page 176.

MultiTemperatureCurve

MultiTemperatureCurve datasets provide a set of E-N curves at different temperatures. The fatigue curves for intervening temperatures are determined by interpolation. Each dataset consists of a generic parent, and a number of children containing the temperature sensitive data. The generic data is as follows:

Parameter Name	Description
MaterialType	Numeric code defining the type of material. The material type is required for correct application of surface finish and stress gradient corrections.
Comments	
References	

There can be a number of child datasets at different temperatures, including all the parameters that are temperature sensitive, in addition to the obvious strain-life curve parameters.

Parameter Name	Description
Temperature	Temperature
UTS	Ultimate tensile strength. This is required for the correct definition of the upper part of the S-N curve, and for applying the static failure criterion.
YS	Yield stress. Optional (not currently used)
E	Modulus of elasticity. Required for S-N when the FE results are elastic strain.
me	Elastic Poisson ratio
mp	Plastic Poisson ratio
Sf'	Fatigue strength coefficient, σ_f'
b	Fatigue strength exponent
c	Fatigue ductility exponent
Ef'	Fatigue ductility coefficient, ε_f'
n'	Cyclic hardening exponent
K'	Cyclic strength coefficient
Nc	Fatigue cutoff. Damage is set to zero beyond this point. It is set in REVERSALS. 2 reversals = 1 cycle

Parameter Name	Description
SEe	Standard error of log(plastic strain)
SEp	Standard error of log(elastic strain)
SEc	Standard error of log(cyclic strain)
Ne	Endurance limit—in reversals
FSN	Fatemi-Socie parameter (not currently used)
S	Wang-Brown parameter (not currently used)

Fatigue properties at a given temperature are interpolated from the two adjacent temperature datasets. There is no extrapolation—if the temperature lies above or below the range of temperatures for which material data is available, the nearest dataset will be used.

Interpolation is carried out as follows:

Assume that we are interpolating to a temperature T, which lies between data at temperature T_1 and T_2.

UTS uses a logarithmic interpolation, that is

$$UTS = 10^{\left(\log UTS_1 + \left(\log UTS_2 - \log UTS_1\right)\left(T - T_1\right)/\left(T_2 - T_1\right)\right)}$$

The Young Modulus **E** is linearly interpolated.

The derivation of n′ and K′ is a little more complicated. Based on the Ramberg-Osgood formulation for the cyclic stress strain curves, a series of values of plastic strain for different stress values are calculated for the two temperatures. These plastic strains are then interpolated to the new temperature logarithmically.

$$\varepsilon_p = 10^{(log\varepsilon_{p1} + \left(log\varepsilon_{p2} - log\varepsilon_{p1}\right)(T-T_1)(T_2-T_1))}$$

This provides a set of stress-plastic strain values at the intermediate temperature, and from these n′ and K′ are derived by linear regression.

Of the other parameters, Sf′ and Ef′ are determined by logarithmic interpolation and the rest by linear interpolation.

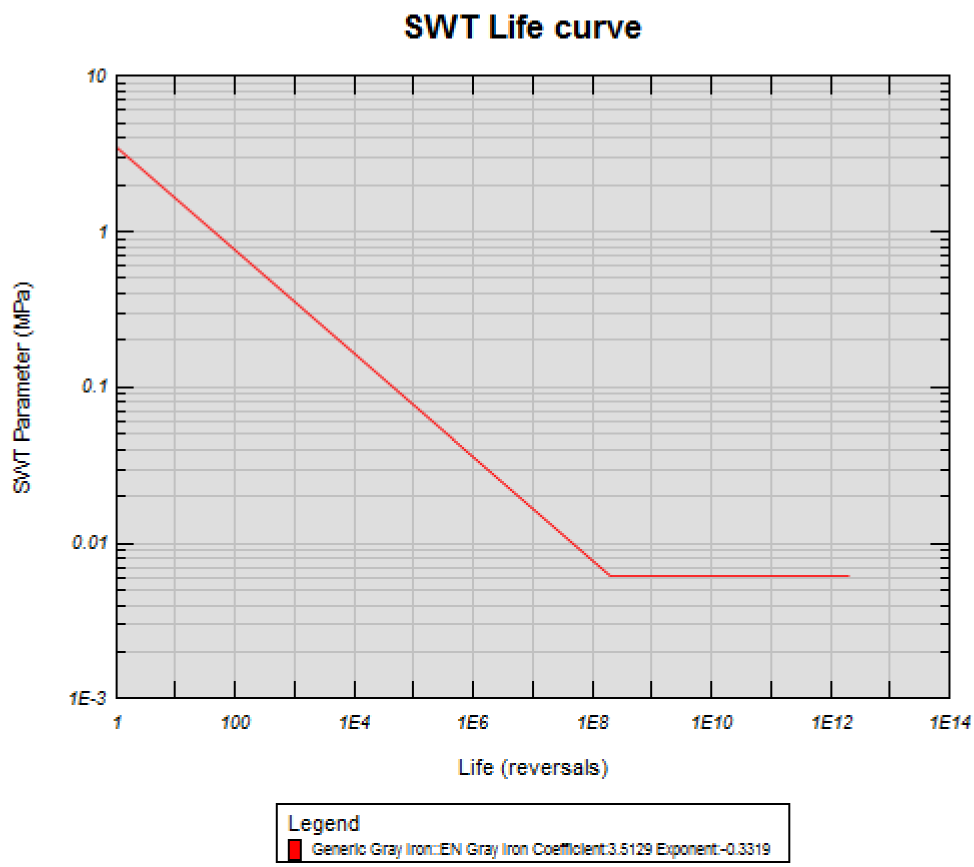
Once the material properties have been modified to take into account the temperature, the calculation proceeds as for a Standard EN analysis.

Gray Iron

This option works with Stress (Linear) or Linear Strain only. If linear strain is supplied then the first operation will be to calculate the equivalent linear stress using the same values for Young’s Modulus (E) and Poison’s Ratio (me) from the material definition; these should be the same values that were used in the Finite element analysis. After the normal stress combination and all stress scaling the stress is used to calculate the linear stress history using the Gray Iron secant equation described in "Gray Iron Strain-life Material Properties" above.

This Gray Iron method uses a very simple linear Smith-Watson-Topper relationship to calculate damage per cycle.

Fig. 5-71 SWT curve for gray iron



The equation is:

$$SWT = K_s.(2Nf)^{Ns}$$

The properties Ks and Ns can be found in the materials data set for gray iron, as can the cut-off value (in cycles). The cut-off defines the life beyond which testing shows insignificant or no damage, and is shown by the flat part of the curve in the plot. See Figure 5-71.

A standard error (based on log life) can also be provided and the SWT equation is modified using this value (SD) and the number of standard deviations from the mean Z (calculated from the certainty of survival—see GlyphWorks Fatigue Theory Guide > *Table 3-1 Certainty of Survival Conversion to Standard Deviation.*

$$SWT = 10^{Z.SD}.K_s.(2Nf)^{Ns}$$

5.4.9 CombinationMethod

For all load providers apart from the Vibration Load Provider, the analysis engine creates a stress tensor history, $\sigma_{ij}(t)$. In order to make a fatigue calculation, we need to reduce this stress tensor to a scalar value, so that we can cycle count it and compare the resulting cycles to an E-N curve or curves. This process is called stress combination. The options, based on the strain tensor ε_{ij} and the principal strains $\varepsilon_{1,2,3}$, are as follows:

AbsMaxPrincipal

The Absolute Maximum Principal strain is defined as the principal strain with the largest magnitude:

$$\varepsilon_{AMP} = \varepsilon_3 \ if \ |\varepsilon_3| > |\varepsilon_1| \ otherwise \ \varepsilon_{AMP} = \varepsilon_1$$

SignedVonMises

The Signed von Mises strain is the von Mises strain, but forced to take the sign of the Absolute Maximum Principal strain, i.e.

$$\varepsilon_{SVM} = \frac{\varepsilon_{AMP}}{|\varepsilon_{AMP}|} \cdot \frac{1}{(1+\nu_e)\sqrt{2}} \cdot \sqrt{(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2}$$

SignedShear

The Signed Shear strain is the Maximum Shear strain (Tresca Criterion) forced to take the sign of the Absolute Maximum Principal strain. A factor is applied to ensure that the parameter has the same value as the Absolute Maximum Principal strain under uniaxial loading conditions.

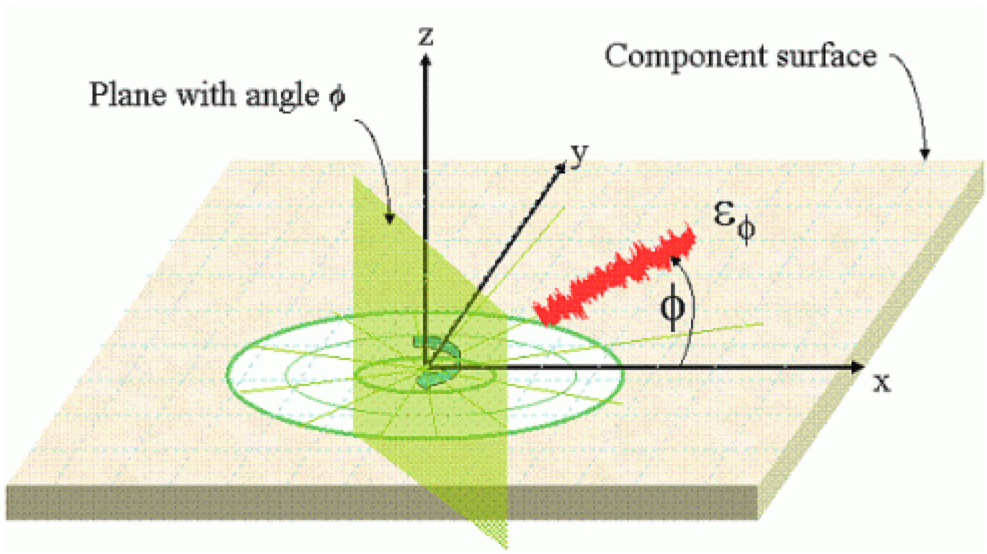
$$\varepsilon_{SSh} = \frac{\varepsilon_{AMP}}{|\varepsilon_{AMP}|} \cdot \frac{(\varepsilon_1 - \varepsilon_3)}{(1+\nu_e)}$$

CriticalPlane

The normal strain is calculated and rainflow counted on multiple planes. The critical plane is the plane with the most predicted fatigue damage. For stresses flagged as 2-D the planes on which the normal strain is determined have normals that lie in the plane of the physical surface, i.e., in the x-y plane of the 2-D stress or strain results coordinate system. The orientation of each plane is defined by the angle ϕ made with the local x-axis.

Note that if the Critical Plane option is being used with solid elements (3D stresses) the distribution of critical planes to be analysed is also three dimensional. In order to achieve a resolution of 10 degrees 204 planes must be analysed, each plane being defined by the orientation of its normal, using two angles (theta and phi). This will significantly increase run-time.

Fig. 5-72 Resolution of normal strain for critical plane analysis

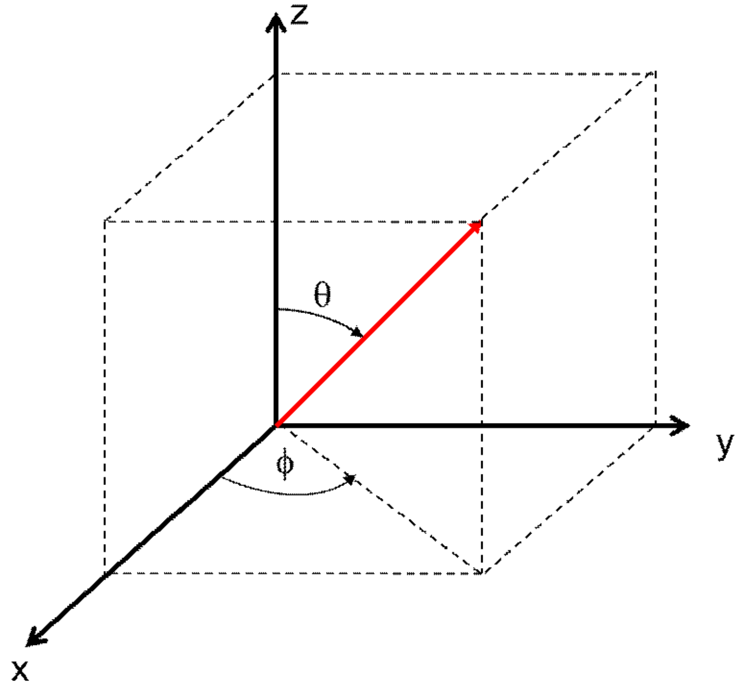


The strain on each plane is calculated from

$$\varepsilon_\phi = \frac{\varepsilon_{xx} + \varepsilon_{yy}}{2} + \frac{\varepsilon_{xx} - \varepsilon_{yy}}{2} \cdot \cos 2\phi + \varepsilon_{xy} \sin 2\phi$$

ϕ can take the values 0, 10, 20, 30, ...170 degrees.

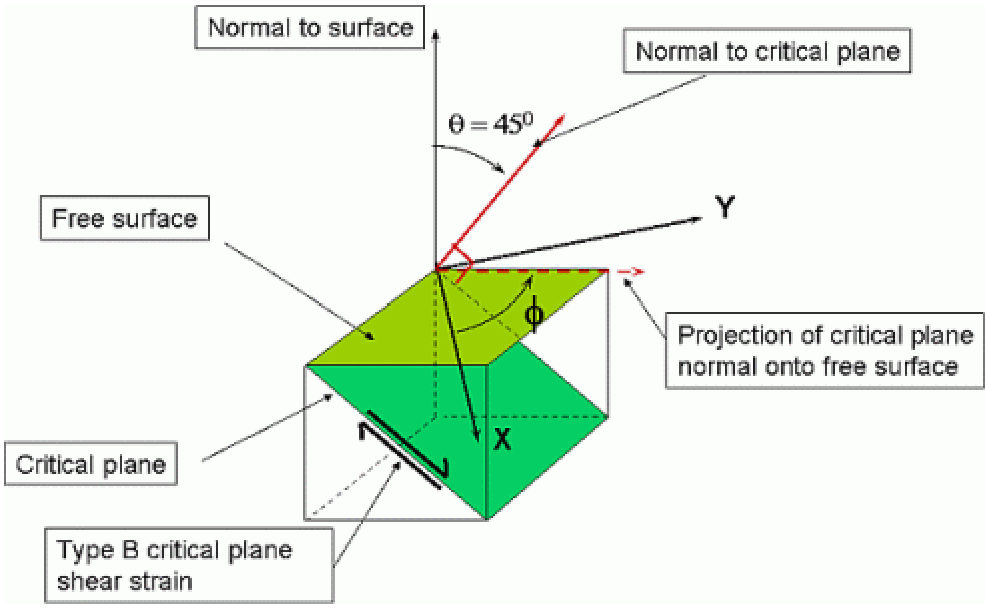
Fig. 5-73 Plane orientation angles illustrated relative to results co-ordinate system



TypeBCriticalPlaneShearStrain

The Type B Critical Plane Shear Strain is derived from the out-of-plane shear strain on planes inclined at 45 degrees to the free surface. It would typically be used with 2-D FE Results (typically from thin shells or resolved to the surface). The orientation of the planes and the direction of the shear strain are illustrated in Figure 5-74.

Fig. 5-74 Orientation of Type B Critical Plane Shear Strain



The parameter is calculated from

$$\varepsilon_{TBCPS,\phi} = \frac{\left(\varepsilon_\phi = \frac{\varepsilon_{xx}+\varepsilon_{yy}}{2} + \frac{\varepsilon_{xx}-\varepsilon_{yy}}{2}\cdot\cos 2\phi + \varepsilon_{xy}\sin 2\phi - \varepsilon_{zz}\right)}{(1+\nu_e)}$$

The denominator in the above equation ensures that under uniaxial conditions, the value of the Type B Critical Plane Shear Strain parameter on the critical plane will approximate to the uniaxial strain.

ϕ can take the values 0, 10, 20, 30, ...170 degrees.

Note

Some of the strain combinations above include the elastic value of the Poisson ratio, which is appropriate when the input data is elastic stress or strain.

When the input data for the calculation is “StressAndStrain”, that is elastic-plastic stresses and strains are being used as input to the local strain (EN) fatigue calculation, strictly speaking the effective Poisson ratio should be used. However, the instantaneous value of the effective Poisson ratio is of little use, and a cycle by cycle calculation requires significant complication of the code. At this stage, for the sake of simplicity, the elastic value is therefore used.

Note also that in some circumstances, notably when using thin shell elements, the through thickness strain (Ezz in local coordinates) is not available in the FE results. Whether or not this is true depends on the FE code. In any case, when the data is flagged as 2-D (i.e., surface resolved, whether from solids or shell elements), DesignLife estimates the through thickness strain, based on the in-plane strain components and the elastic value of the Poisson ratio.

The effect of these two approximations is to introduce an error into the calculation of some of the strain combinations, notably SignedShear and TypeBCriticalPlaneShearStrain. The error is minimal when the strain is predominantly elastic (high cycle fatigue) and at its worst when the strain is mostly plastic (low cycle fatigue). When the strain is mostly plastic, assuming an elastic Poisson ratio of 0.3, the combined strains may be underestimated by as much as 18% (equibiaxial loading) or overestimated by as much as 15% (pure shear loading). Since this may result in non-conservative calculations, the user may wish to treat results with some caution. To be clear, the situation will arise when elastic-plastic stress and strain inputs are used with shell elements or surface resolved stresses and strains AND the combined strain parameter requires the local z component of strain (Ezz). Combinations requiring Ezz are: SignedVonMises, SignedShear and TypeB-CriticalPlaneShearStrain. “Auto” multiaxial mode may also be affected if it selects one of these criteria.

Note that if the Type B Critical Plane Shear Strain option is being used with solid elements (3D stresses) the distribution of critical planes to be analysed is also three dimensional. In order to achieve a resolution of 10 degrees 204 planes must be analysed, each plane being defined by the orientation of its normal, using two angles (theta and phi). This will significantly increase run-time.

5.4.10 MeanStressCorrection

The main feature of a strain cycle that affects fatigue damage is its range. Fatigue damage is also influenced by the mean **stress** of each cycle. Mean stress correction methods allow the effect of mean stress to be modeled and taken into account in the life prediction. This section describes the different methods supported.

None

There will be no correction for mean stress.

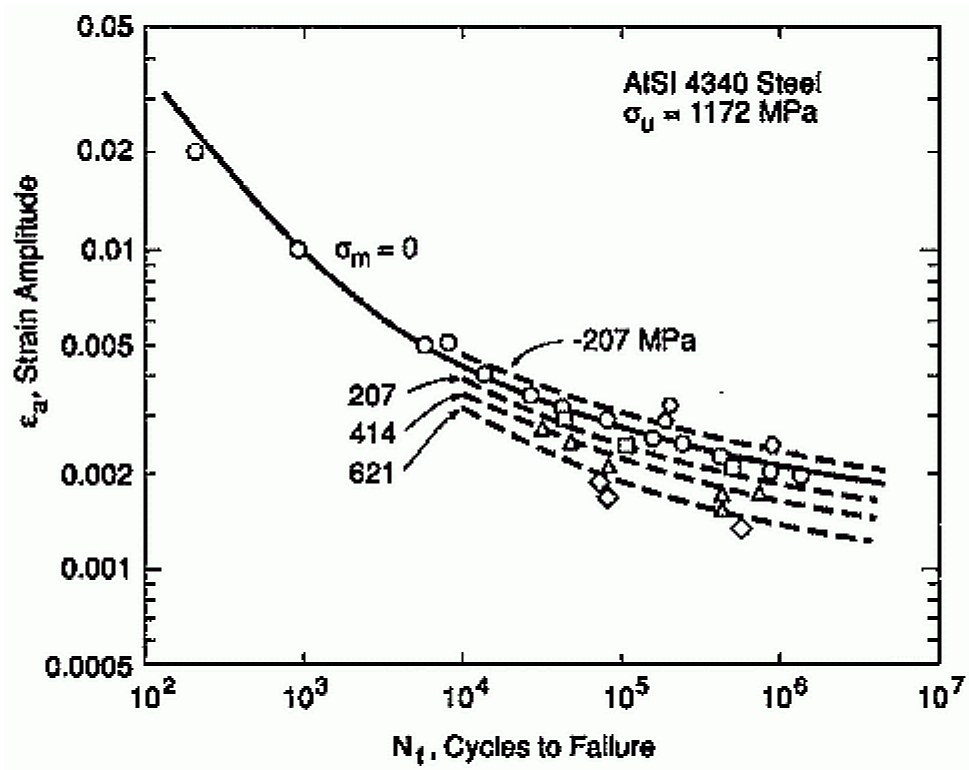
Morrow

The Morrow mean stress correction adjusts the value of the intercept of the elastic part of the strain life curve before looking up the life/damage, on a cycle-by-cycle basis as follows:

$$\varepsilon_a = \frac{(\sigma_f' - \sigma_m)}{E}(2N_f)^b + \varepsilon_f'(2N_f)^c$$

...where σ_m is the mean stress of each cycle. The effect of this mean stress correction, correlated with test data, is illustrated in Figure 5-75:

Fig. 5-75 Morrow mean stress correction



SmithWatsonTopper

While loosely termed a mean stress correction method, the Smith-Watson-Topper method actually defines a new damage parameter based on the product of the strain amplitude and the maximum stress of each cycle.

$$P_{SWT} = \varepsilon_a \sigma_{\max}$$

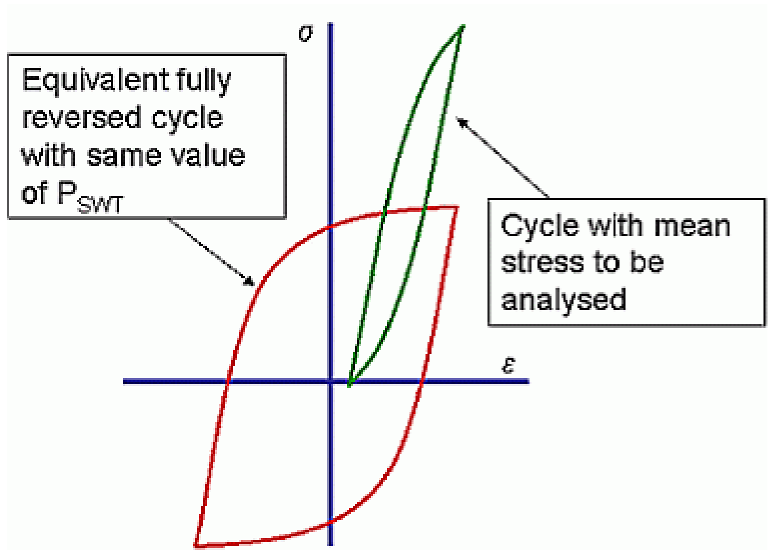
In practice, there are two ways this can be applied. These are set using the SWT-Method property.

1. In the **Formula** method, the following equation is solved for life/damage.

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

2. In the **Iterative** method, a fully reversed cycle that has the same value of P_{SWT} as the cycle being analyzed is sought.

Fig. 5-76 Iterative Smith-Watson-Topper method



The strain amplitude of the fully reversed cycle defines an equivalent strain amplitude that can be looked up on the standard strain-life curve. Because the strain amplitude and max stress of the equivalent fully reversed cycle are related by the cyclic stress-strain curve, the equations to be solved are:

$$P_{SWT} = \varepsilon_a \sigma_{\max} = \varepsilon_{a,equiv} \sigma_{\max,equiv}$$

$$\varepsilon_{a,equiv} = \frac{\sigma_{\max,equiv}}{E} + \left(\frac{\sigma_{\max,equiv}}{K'} \right)^{\frac{1}{n'}}$$

The equivalent strain amplitude is then looked up on the standard strain-life curve as normal.

Generally, the Iterative method is considered a more correct approach.

Interpolate

When Interpolate is selected, the mean stress effect will be taken into account by interpolation (or extrapolation) from multiple curves. The exact method depends upon the E-N data type and the differences are detailed in the section on “ENMethod” on page 175 (MultiMeanCurve, MultiRRatioCurve).

5.4.11 InterpolationLimit

This property applies only when the ENMethod is set to MultiMeanCurve or MultiRRatioCurve. It controls whether or not extrapolation will be carried out if the mean or R-ratio of a cycle lies outside the range defined by the curve set. See the section on ENMethod and the section on the Standard SN Analysis Engine for more details of the interpolation schemes.

5.4.12 MultiAxialAssessment

There are four options:

- None
- SimpleBiaxiality
- Standard
- Auto

For the basic principles of multiaxial assessment, please refer to the appropriate part of the section on the "Standard SN Analysis Engine" on page 84. The basic principles of the *SimpleBiaxiality* and *Standard* methods are the same as in the S-N engine. The only difference in implementation is that the full strain tensor histories must first be converted to stresses (using an elastic assumption) before the assessment can take place.

Auto mode

Auto mode uses a two-pass approach, which overrides the *CombinationMethod* and *ElasticPlasticCorrection* property settings. When *Auto* is selected, six additional properties must be set:

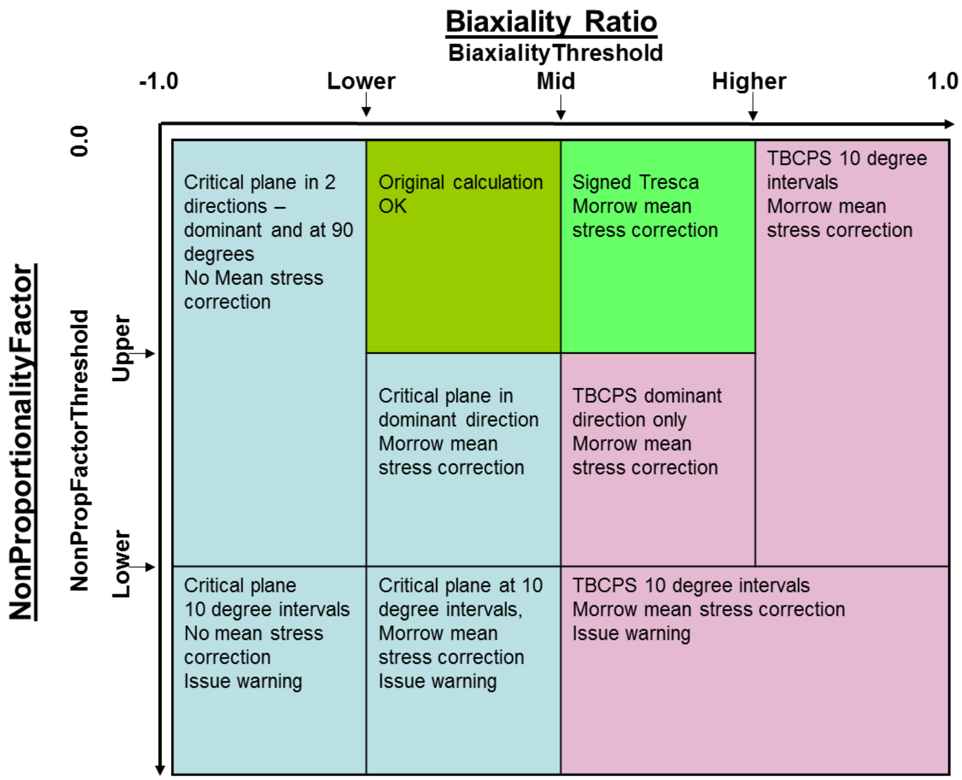
Table 5-9 Additional Properties Associated with Auto Mode

Property	Default Setting
ZeroDamageStressPercent	10
NonPropFactorThresholdLower	0.25
NonPropFactorThresholdUpper	0.5
BiaxialityThresholdLower	-0.6
BiaxialityThresholdMid	0.25
BiaxialityThresholdUpper	0.6

1. In the first pass, the *CombinationMethod* is set to *AbsMaxPrincipal* (the "Normal" method) and a *Standard* assessment is carried out. The *Hoffmann-Seeger* notch correction is used.
2. A second pass may be carried out *if* the value of *STRMAX* (see the section describing the Standard Multiaxial Assessment method, "MultiAxialAssessment = Standard" on page 138) exceeds the percentage of UTS defined by *ZeroDamageStressPercent* AND if the non-proportionality and/or biaxiality values lie outside the ranges defined for

the “Normal” method by the relevant threshold values and detailed in the following table.

Fig. 5-77 Application of thresholds in the Auto multiaxial option



The *AbsMaxPrincipal*, *SignedTresca*, *CriticalPlane* and *TypeBCriticalPlaneShear (TBCPS)* strain combination methods have already been defined. The other possible options are the *CriticalPlane* option, but looking at only two planes - the *Dominant Stress Direction* and at 90 degrees to the Dominant Stress Direction. The *TypeBCriticalPlaneShear* option may also be applied in the dominant direction only.

The results reported for *Auto* mode are as for *Standard*, with the addition of the Calculation Method (Normal, Critical Plane etc.).

Note When *MultiAxialAssessment=Auto* is used together with *EventProcessing=Independent*, some additional work is done to compress the analysis results prior to outputting them. This is because the *Auto* method can lead to a different number of results for each event, depending on the analysis method chosen as a result of the assessment.
Therefore, only the worst case damage is taken for each event. The damage for all events is summed from the individual events as normal.

Note on Duty Cycle Processing

When a duty cycle is being processed, the biaxiality parameters are calculated for the individual unique events that make up the duty cycle, as well as for the duty cycle as a whole. The statistics for the duty cycle as a whole take into account the number of repeats of each event.

5.4.13 ElasticPlasticCorrection

Elastic-plastic corrections are often required because the local strain approach requires total (elastic-plastic) strain as well as stress, yet for efficiency reasons, calculations often have to be based on linear elastic FE calculations. Notch corrections allow elastic plastic strains and stresses to be estimated based on elastic FE results.

There are three options in nCodeDT:

- None
- Neuber
- Hoffmann-Seeger

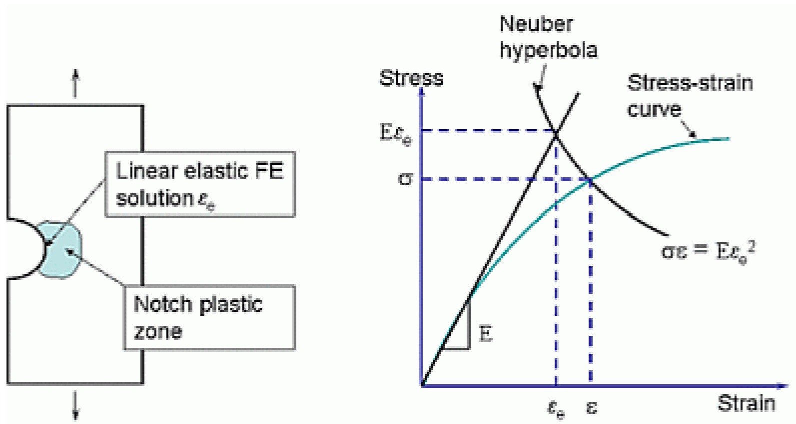
None

No elastic-plastic correction will be carried out. This option is valid only when both elastic-plastic strains and stresses are available from the FE. The Minimum and Maximum stress occurring **during each rainflow cycle** is determined and these values are used as input to any mean stress correction.

Neuber

The Neuber method provides a simple way of estimating the total elastic-plastic strain and stress at an "average" stress concentration, based on the local elastic stress/strain. Consider a simple notched specimen subjected to a uniaxial loading. See Figure 5-78. As long as the yield stress is not exceeded, linear FE analysis gives (assuming a good model) a reasonably accurate estimate of the strain and stress at the root of the notch. However, once the yield stress is exceeded, the elastic solution becomes increasingly unrealistic. In practice, as yielding occurs, there will be a redistribution of stress and strain around the notch, so that the real strain will be greater than the elastic value and the real stress less than the value from elastic analysis. The true solution must lie somewhere on the material stress-strain curve. To get a reasonably accurate estimate of the way this stress and strain is redistributed, we could carry out an elastic-plastic FE solution, taking into account the geometry of the specimen, but this could be rather time consuming, especially if we have to consider many loading cycles.

Fig. 5-78 Neuber method for estimating elastic plastic strain and stress at a notch



The Neuber method provides a simple alternative that provides a rough estimate for how the stress and strain might redistribute, without reference to the real geometry. The Neuber method assumes that the product of stress and strain before and after redistribution is constant. This is represented by the Neuber hyperbola, where the product of stress and strain is constant and equal to the elastic stress x elastic strain.

The Neuber method can be applied to monotonic or cyclic loading. In nCodeDT, we apply it to cyclic loading, in one of two ways:

1. To position the outside hysteresis loop by applying Neuber to the Abs Max value of the elastic strain in the strain history, together with the cyclic stress-strain curve. This is achieved by solving the pair of equations

$$E\left(\varepsilon_{e,AbsMax}\right)^2 = \sigma_{\max}\varepsilon_{\max}$$

$$\varepsilon_{\max} = \frac{\sigma_{\max}}{E} + \left(\frac{\sigma_{\max}}{K'}\right)^{\frac{1}{n'}}$$

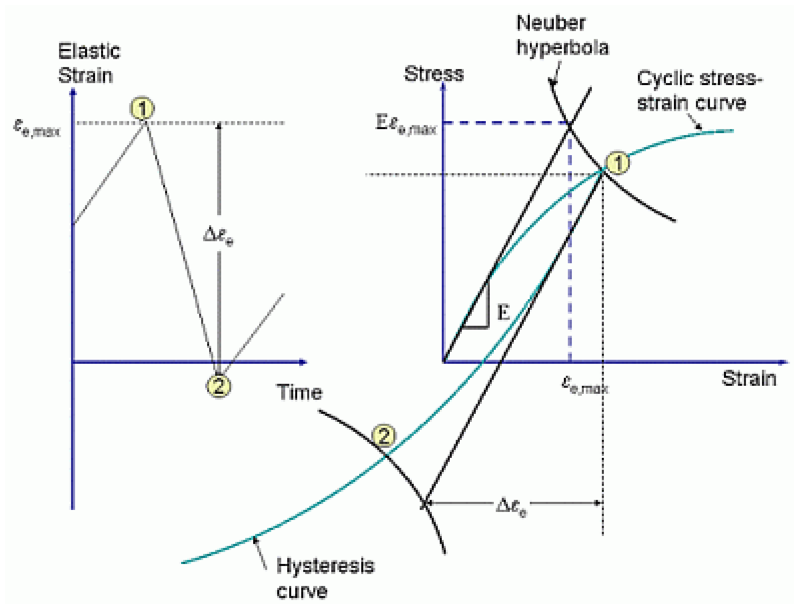
2. To calculate a subsequent strain excursion, by applying Neuber to the elastic strain range of the excursion, together with the hysteresis curve, i.e., by solving the pair of equations

$$E\left(\Delta\varepsilon_e\right)^2 = \Delta\sigma\Delta\varepsilon$$

$$\Delta\varepsilon = \frac{\Delta\sigma}{E} + 2\left(\frac{\Delta\sigma}{2K'}\right)^{\frac{1}{n'}}$$

These two ways of using Neuber are illustrated in Figure 5-79. First, the position of the Absolute Maximum value is estimated using the Neuber method at Point 1. All hysteresis loops are positioned relative to this point. The next strain excursion, to Point 2, is then corrected by applying the Neuber correction to the range of the excursion, using the hysteresis curve.

Fig. 5-79 Application of Neuber correction to cyclic loading



Note that the Neuber method as implemented in nCodeDT takes the damage parameter (Abs Max Principal, Signed von Mises, etc.) and treats it as if it is the strain under uniaxial loading conditions (biaxiality ratio a = 0). In reality, the stress state is very often not uniaxial, and the Hoffmann-Seeger method may give more realistic results.

Hoffmann-Seeger

The Hoffmann-Seeger method is a modified version of the Neuber method that takes into account the state of stress, allowing it to be extended to proportional multiaxial loadings. If the loading is very non-proportional, then most of the assumptions implicit in this method are violated; hardening is really kinematic and not isotropic, and the principal axes may not only rotate, but the principal stress and strain axes are no longer necessarily aligned with each other.

The Hoffmann-Seeger method requires 2-D stresses or strains (e.g., from thin shells, or surface resolved) and knowledge of the biaxiality ratio (see the notes on Multiaxial Assessment in the section "Standard SN Analysis Engine" on page 84). As implemented in nCodeDT, the Hoffmann-Seeger method makes the following assumptions:

- The principal stress and strain axes are fixed in orientation.
- The ratio of the in-plane principal strains is constant.
- The uniaxial stress-strain curve can be extended for use with von Mises equivalent stress and strain under different states of stress.
- Hencky's flow rules.

- Masing's hypothesis and material memory are implemented to achieve stress-strain tracking in the same way as with Neuber.

For convenience, the in-plane principal stresses and strains are denoted σ_1, σ_2, ε_1, ε_2 and the surface normal strain ε_z. The principals are ordered so that

$$|\sigma_1| \geq |\sigma_2|, \quad |\varepsilon_1| \geq |\varepsilon_2|$$

First, the elastic value of the signed von Mises strain is computed, which is defined as

$$\varepsilon_{eq} = \frac{\varepsilon_1}{|\varepsilon_1|} \cdot \frac{1}{(1+\nu_e)\sqrt{2}} \cdot \sqrt{(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_z)^2 + (\varepsilon_z - \varepsilon_1)^2}$$

If the chosen damage parameter (stress combination) is Signed von Mises, this is calculated directly. However, in practice, the calculation will often be based on the Abs Max Principal strain, Critical Plane strain, etc. In these cases, the elastic equivalent (Signed von Mises) strain can be deduced knowing the elastic Poisson ratio ν_e and the mean elastic biaxiality ratio a_e (convert to von Mises).

If using Abs Max Principal Strain or Critical Plane:

$$\varepsilon_{eq} = \frac{\varepsilon_1 \sqrt{1 - a_e + a_e^2}}{1 - a_e \nu_e}$$

If using Signed Tresca or Type B Critical Plane Shear, and $a_e \geq 0$:

$$\varepsilon_{eq} = \varepsilon_{SignedTresca} \sqrt{1 - a_e + a_e^2}$$

- or if $a_e < 0$:

$$\varepsilon_{eq} = \frac{\varepsilon_{SignedTresca} \sqrt{1 - a_e + a_e^2}}{1 - a_e}$$

Then the elastic equivalent stress can be deduced:

$$\sigma_{eq} = E\varepsilon_{eq}$$

where "eq" denotes elastic equivalent.

The strain biaxiality ratio is deduced from the mean value of the elastic biaxiality ratio (from the multiaxial assessment) as follows:

$$\frac{\varepsilon_2}{\varepsilon_1} = \frac{a_e - \nu_e}{1 - a_e \nu_e}$$

This strain ratio is assumed constant.

Now the Neuber correction is carried out as normal, but based on the elastic equivalent strain and stress, i.e., by solving

$$\varepsilon_q = \frac{\sigma_q}{E} + \left(\frac{\sigma_q}{K'}\right)^{1/n'} \quad and \quad \sigma_q \varepsilon_q = \sigma_{eq} \varepsilon_{eq} = E\varepsilon_{eq}^2$$

if positioning the largest hysteresis loop, or:

$$\Delta\varepsilon_q = \frac{\Delta\sigma_q}{E} + 2\left(\frac{\Delta\sigma_q}{2K'}\right)^{1/n'} \quad and \quad \Delta\sigma_q \Delta\varepsilon_q = \Delta\sigma_{eq} \Delta\varepsilon_{eq} = E\Delta\varepsilon_{eq}^2$$

...for subsequent strain excursions. The subscript "q" here denotes the estimated elastic-plastic strain and stress.

Once the plasticity correction has been carried out, the principal stresses and strains can be calculated, assuming Hencky's flow rules and that the principal strain ratio remains constant.

First, the effective Poisson ratio must be calculated:

$$\nu' = 0.5 - (0.5 - \nu_e)\frac{\sigma_q}{E\varepsilon_q}$$

Then:

$$\varepsilon_1 = \varepsilon_q \frac{1 - \nu' a}{\sqrt{1 - a + a^2}}$$

$$\varepsilon_2 = \left(\frac{\varepsilon_2}{\varepsilon_1}\right)\varepsilon_1$$

$$\sigma_1 = \sigma_q \frac{1}{\sqrt{1 - a + a^2}}$$

$$\sigma_2 = a\sigma_1$$

$$a = \frac{\sigma_2}{\sigma_1} = \frac{\frac{\varepsilon_2}{\varepsilon_1} + \nu'}{1 + \nu'\frac{\varepsilon_2}{\varepsilon_1}}$$

...where *a* is the elastic plastic biaxiality ratio. (If the ratio of the principal strains is considered fixed, the ratio of the principal stresses will in general change.)

$$a = \frac{\sigma_2}{\sigma_1} = \frac{\frac{\varepsilon_2}{\varepsilon_1} + \nu'}{1 + \nu'\frac{\varepsilon_2}{\varepsilon_1}}$$

Critical plane strain is treated as if it were Abs Max Principal Strain, and Type B Critical Plane Shear strain is treated as if it were Signed Tresca Strain.

Note that the assumption of constant strain ratio means that the Hoffmann-Seeger method will give slightly different results to the Neuber method for uniaxial loadings. However, the differences are small, and as long as 2-D FE results are available, the Hoffmann-Seeger method is a sensible default.

5.4.14 PlasticLimitLoadCorrection

Simple notch rules such as the Neuber and Hoffmann-Seeger rules estimate the redistribution of stress and strain around a notch with limited yielding, but with no reference to the geometry of the component. In practice, particularly when a notch is shallow or the loading is high, plasticity may become more widespread, and these simple notch rules may then significantly underestimate the plastic strains.

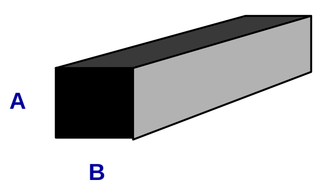
The Seeger-Heuler method provides a variation on the Neuber or Hoffmann-Seeger methods that increases the amount of plasticity estimated when net section yielding is predicted, based on a Shape Factor. If this property is set to “None”, there is no effect of shape factor considered. The property has no effect if ElasticPlasticCorrection = None.

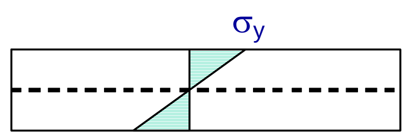
The Shape Factor or limit load ratio for any cross-section is defined: $\alpha_P = \frac{L_P}{L_Y}$

the ratio of the plastic limit load (load at which plasticity spreads across the section) to the yield load (load at which yielding starts). This limit load ratio or shape factor or formzahl can be calculated for simple sections assuming an idealized elastic-perfectly plastic stress-strain curve.

For example:

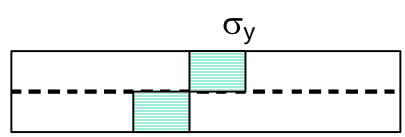
Assuming elastic-perfectly plastic loading, the yield moment for a *rectangular cross section bar in bending* is:





$$M_y = \frac{BA^2}{6}\sigma_y$$

The plastic limit moment is :



$$M_p = \frac{BA^2}{4}\sigma_y$$

So the shape factor $\alpha_p = M_p/M_y = 1.5$

Suppose we now add a **small** notch to the surface of the bar with a stress concentration factor of 2.5; the yield load will be reduced by a factor of 2.5, but if the notch is small, the plastic limit load will be more or less unaffected, so we might estimate a shape factor for the notched bar as 2.5 x 1.5 = 3.75

An unnotched bar in tension is a trivial case where $\alpha_p = 1$.

From similar arguments, a plate with a small round hole in tension would have $\alpha_p \sim 3$.

When *PlasticLimitLoadCorrection* = *SeegerHeuler*, the application of the notch rule is very similar to that described for the Neuber method described earlier, the only difference being that the stress-strain product is modified.

1. To determine the position of the largest hysteresis loop, we solve:

$$\sigma\varepsilon = E\varepsilon_e \cdot \alpha_p \cdot \left(\frac{\varepsilon_e}{\alpha_p} + \left(\frac{E\varepsilon_e}{\alpha_p K'} \right)^{\frac{1}{n'}} \right) \text{ and } \varepsilon = \frac{\sigma}{E} + \left(\frac{\sigma}{K'} \right)^{\frac{1}{n'}} \text{ for } \varepsilon \text{ and } \sigma.$$

2. To determine the range of stress and strain associated with each subsequent excursion, we solve:

$$\Delta\sigma\Delta\varepsilon = E\Delta\varepsilon_e \cdot \alpha_p \cdot 2\left(\frac{\Delta\varepsilon_e}{2\alpha_p} + \left(\frac{E\Delta\varepsilon_e}{2\alpha_p K'} \right)^{\frac{1}{n'}} \right) \text{ and } \frac{\Delta\varepsilon}{2} = \frac{\Delta\sigma}{2E} + \left(\frac{\Delta\sigma}{2K'} \right)^{\frac{1}{n'}} \text{ for } \Delta\varepsilon \text{ and } \Delta\sigma.$$

When the strain levels are low and/or the shape factor is large, the terms

$$\left(\frac{E\Delta\varepsilon_e}{2\alpha_p K'} \right)^{\frac{1}{n'}} \text{ and } \left(\frac{E\varepsilon_e}{\alpha_p K'} \right)^{\frac{1}{n'}}.$$

become insignificant, and the solution converges with the Neuber method.

Since this approach could equally well be applied to the equivalent stress and strain, it is compatible with the Hoffmann-Seeger method.

5.4.15 CertaintyOfSurvival

The certainty of survival (in %) allows statistical variations in material behavior to be taken into account. The usual application of this is to provide a more conservative prediction to ensure a safer design. The variability in material properties is characterized by standard error parameters, which should be determined when fitting material curves to Strain-Life and Cyclic Stress-Strain test data. The certainty of survival values are converted into a number of standard errors using the lookup table (Table 5-1 ; see the section on the Standard SN Analysis Engine) and this is used to adjust the cyclic stress-strain and strain-life curves, as described previously.

Please note the following:

1. If any of the standard error parameters in the material database are set to zero, changing the Certainty of Survival will have no effect on the results. This does not mean there is no scatter in the material behavior, merely that it has not been calculated.
2. There is currently no facility to model the effect on life of design criterion for Multi-Curve E-N methods.
3. The effect of SEc on life predictions can be counter-intuitive, particularly when applied to the SWT parameter. You can work around this by setting SEc to zero.

5.4.16 ScaleFactor

This scale factor will be applied to the elastic strains before the fatigue calculation. The default value is 1.0.

5.4.17 OutputMaxMin

If this property is set to True, the maximum and minimum stress and strain values for each entity (node/element) will be included in the results. These are elastic-plastic values taken from the tracking cycle counter, derived from the outer hysteresis loop after application of any notch correction.

5.4.18 BackCalcMode

The options are **None** or **ScaleFactor**. When **ScaleFactor** is selected, the E-N Analysis engine will iteratively search for a Scale Factor, that, when applied to the strains before the fatigue calculation, will produce the TargetDamage within a reasonable tolerance set by BackCalcAccuracy**.** This setting overrides the ScaleFactor property setting on the analysis engine (see above). It provides a convenient way of determining a safety factor for complex or variable amplitude loadings. When this option is selected, the damage reported in the results will approximate to the target damage, and the resulting scale factor will be reported, too.

None	Static failure checks are not carried out during EN back calculations.

5.4.19 TargetDamage

Sets the target damage value to be used for back calculations.

5.4.20 BackCalcAccuracy

This defines a percentage error on the target damage. The back calculation will iterate until the calculated damage matches the target damage within this tolerance.

5.4.21 BackCalcMaxScale

This defines the maximum value of the scale factor that will be reported in the analysis results following a back calculation.

5.4.22 EventProcessing

A duty cycle is a sequence of events to be processed through the analysis engine. The creation and types of different duty cycle are described in the section on Loading. Duty cycles can be processed by the E-N Analysis engine in three different ways, which can be summarized as follows:

- **Independent** mode. Each unique event in the duty cycle is processed separately, and the damage from each is multiplied by the number of times it appears in the complete sequence. The total damage is the sum of the damage from all the different events. This is fast, but may miss significant cycles that cross different events.

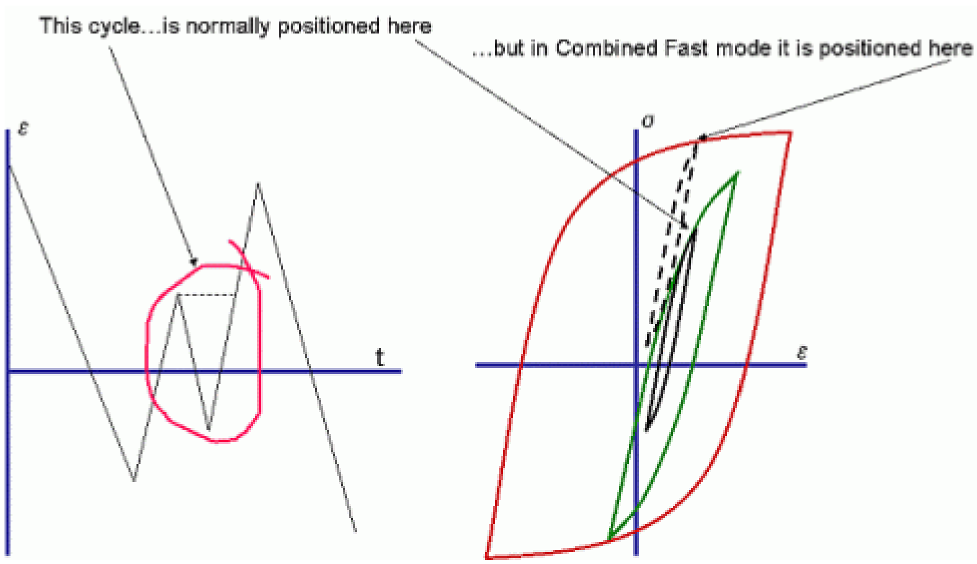
Note	When *MultiAxialAssessment* = *Auto* is used together with *EventProcessing* = *Independent*, some additional work is done to compress the analysis results prior to putting them out. This is because the *Auto* method can lead to a different number of results for each event, depending on the analysis method chosen as a result of the assessment. Therefore, only the worst case damage is taken for each event. The damage for all events is summed from the individual events as normal.

- **CombinedFull** mode. All the events in the duty cycle are concatenated, including all repeats of each event, and the resulting strain history is processed as if it was one long event. This is accurate, but may be very computationally inefficient.
- **CombinedFast** mode. An intermediate method that cycle counts events individually but also captures the cycles that cross events. It will in general be almost as accurate as the CombinedFull option and almost as quick as the Independent mode.

The details of these methods are described in the section on the Standard SN Analysis Engine. See “EventProcessing” on page 144. The same principles apply to the E-N engine. There is one significant complication, which is that the tracking cycle counter cannot work for the CombinedFast option in the same way as for Independent or CombinedFull modes. This is because the CombinedFast option cannot preserve the exact sequence of cycles. Normally, the tracking cycle counter works on the elastic strain history to track the stress and strain response, determine the rainflow cycles, and position the corresponding hysteresis loops. When CombinedFast mode is selected, the rainflow count is carried out on the elastic history, and the exact sequence of cycles cannot be preserved. The input to the tracking part of the cycle counter (the part that positions the hysteresis loops) is therefore no longer a strain history but a list of cycles. We don’t know the exact sequence of these cycles, but we can position the largest cycle in the normal way, and we do know whether each smaller cycle should be “hanging” or “standing” in the outside loop (i.e., whether it was an interruption to a positive going or negative going strain excursion).

The largest hysteresis loop is positioned in the normal way, based on the absolute maximum value. All smaller cycles are positioned as either “hanging” or “standing” loops. There is a slight loss of precision inherent in this process because cycles that would normally “hang” inside a “standing” loop, will instead be “hung” from the outside loop. This leads to a minor loss in accuracy in the mean stress of minor cycles, but is unlikely to lead to a significant error in the overall fatigue calculation.

Fig. 5-80 Positioning of minor hysteresis loops in Combined Fast mode



5.4.23 OutputEventResults

If this property is set to True, the accumulated damage (and other calculated results) associated with each individual event from the duty cycle will be output in addition to the total damage. If either of the “Combined” options is being used, some cycles may cross events. The damage associated with these cycles is split equally between the two events with which it is associated.

5.4.24 SWTMethod

There are two possible interpretations of the SWT (Smith-Watson-Topper) method: **Iterative** and **Formula**. These have been described above.

5.4.25 DamageCalcMethod

The solutions of the equations required to predict life/damage in the local strain approach require numerical methods. There are two options. **SingleShot** uses an iterative (Newton-Raphson) method, and **LookupTable**, as the name suggests, linearizes the curves into a lookup table. When there are a lot of cycles, the use of the lookup table may speed up the calculation, but at the cost of a slight loss of precision, depending upon the size of the table.

5.4.26 LookUpTableSize

Sets the number of values used in the lookup table (see above). Larger values will give increased accuracy at the expense of some speed.

5.4.27 CheckStaticFailure

Static failure checking compares the maximum value of the CombinedStress, from the tracking cycle counter (i.e., after application of the Neuber or Hoffmann-Seeger notch correction) with the material UTS. There is currently no criterion applied for compressive failure. There are three options:

- **Warn**. The calculation continues, but a warning message is issued. The damage for this entity is also assigned a numeric value equal to the value of the StaticFailure Damage property, typically 2.0.
- **Stop**. The analysis job stops with a message.
- **False**. No static failure check is made. Damage is calculated up to the numerical limit defined by the MaxDamage property. Note that this may use an extrapolation of the E-N curve and may predict damage values per cycle greater than 1.0.

5.4.28 DamageFloor

This parameter sets a lower limit on calculated damage. If a cycle has less predicted damage, the damage value is set to zero. This parameter is somewhat redundant, as similar functionality may be accessed using the fatigue cutoff Nc in the material definition.

5.4.29 MaxDamage

This parameter sets a numerical upper limit on the damage that can be predicted for each cycle.

5.4.30 StaticFailureDamage

This parameter sets a numerical value that may be set for the damage to a particular node or element in the event of the stress exceeding the UTS. (See CheckStaticFailure above.)

5.4.31 StressGradients

This property has three possible values:

- **Auto**. A correction for stress gradient is applied if stress gradients are present (i.e., if they were requested at the translation stage in the Analysis Group properties).

- **User**. A user-defined stress gradient correction is applied (if stress gradients are available) based on a lookup table provided in a file. A valid file must be specified using the StressGradientsUser property.
- **Off**. No stress gradient correction is applied.

The stress gradient correction works in exactly the same way as for the Standard S-N Analysis engine, with the correction factor being applied to the strain before strain combination, multiaxial assessment, rainflow cycle counting, etc.

5.4.32 StressGradientsUser

If StressGradients is set to **User**, the name of a valid stress gradients user file must be provided here.

The User file is an ASCII file that provides a lookup table of G_σ vs n_σ which will be interpolated. It must be in the following format:

```
Stress gradient correction factor file v1.0
Dimension=mm
#Normalized stress gradient G, correction factor n
StartCorrectionData
0,1
1,1.1
10,1.2
100,1.3
EndCorrectionData
```

- Text after # on any line is ignored.
- Blank lines are ignored.
- The first line in the stress gradient user file must be as above.
- A Dimensions line must precede the data section.
- The data section must be bound by StartCorrectionData and EndCorrectionData.
- The options for Dimension are all supported units types for LENGTH. (See `unitconv.sys` in the `nssys` directory of your installation.)

5.4.33 StressGradientMethod

See 5.3.26 StressGradientMethod in “Standard SN Analysis Engine”.

5.4.34 OutputStressGradientFactors

See 5.3.27 OutputStressGradientFactors in “Standard SN Analysis Engine”.

5.5 Multiaxial EN Analysis Engine

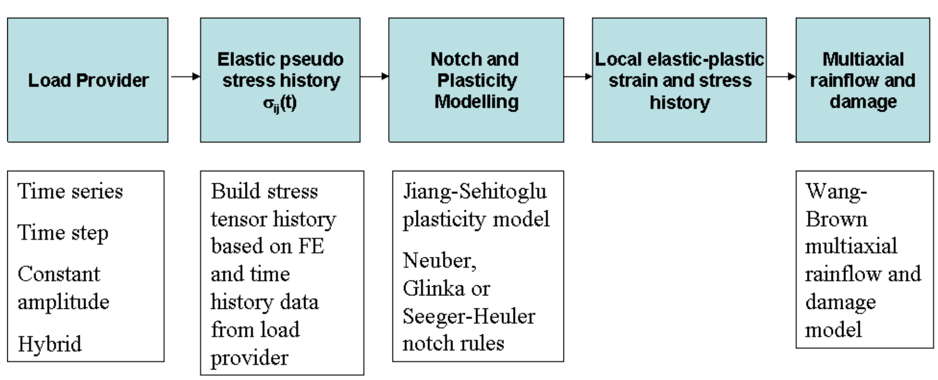
Fig. 5-81 Multiaxial EN analysis engine properties

General		
ENMethod	Standard	The method used to calculate damage from a stress cycle
DamageMethod	WangBrown	Method used to calculate fatigue damage
PlasticityModel	JiangSehitoglu	Plasticity Model
NotchCorrection	Neuber	Notch Correction
CertaintyOfSurvival	50	Required certainty of survival (%) on damage results
ScaleFactor	1	The scale factor to apply prior to the input time history
OutputMaterialNames	False	Whether to output material names to the results
OutputDistributedSource	False	Whether to output details of the distributed process that generated each result
DutyCycle		
EventProcessing	Independent	How to process separate events in duty cycles
OutputEventResults	False	Whether to output results per event or not for duty cycle processing
Advanced		
DamageFloor	0	The calculated damage value below which the damage is set to zero
MaxDamage	1E30	The maximum damage value
CheckMaxStrain	Stop	How to deal with time histories that exceed the maximum strain
StrainLimitDamage	2	Damage value set if strain limit is exceeded
Jiang Sehitoglu Plasticity M		
TimeHistoryProcessing	FromMaxVonMises	Method used to handle time history
NumberOfSurfaces	50	Number of surfaces used in plasticity model
Beee	5	Non-proportional rate parameter.
Chi	100	Ratcheting rate parameter.
IncrementMethod	Single	Determines the method used to increment the plasticity model.
MaxStrain	2E4	Maximum allowed strain (uE).
YieldStrain	50	Yield Strain (uE).
HardeningTolerance	0.05	Determines steady state.
KPrimeRatio	1	Default scale factor used to calculate K'90.
nPrimeRatio	1	Default scale factor used to calculate n'90.
IncrementSize		
IncrementSize	10	Maximum strain increment size (uE).

5.5.1 Summary

The basic operation of the Multiaxial EN Analysis Engine can be summarized in the following diagram:

Fig. 5-82 Basic Multiaxial EN Engine steps

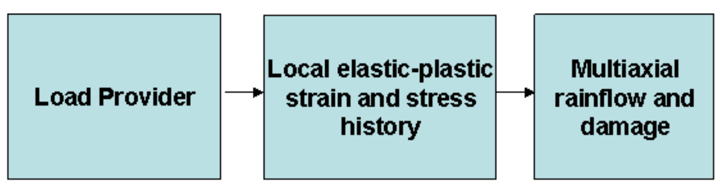


A typical Multiaxial EN analysis has the following basic steps, which are executed for each analysis entity (Node, Element, Node-on-Element) and shell surface, where appropriate:

1. The stress tensor history, as a function of time or data points, is assembled from the information provided in the load provider. This applies to time-series, time-step, constant amplitude, duty cycle and hybrid load providers. Vibration Load Providers are not supported. Note that stress input is required and that the stress tensor must be 2-D (surface resolved). See "Resolve to Local" in the section "FE Import Analysis Group" on page 16 for details
2. The stress tensor (this is the 2-D elastic pseudo-stress tensor) is processed using a multiaxial plasticity model, combined with a notch correction, to generate an estimated local elastic-plastic strain and stress history.
3. The elastic-plastic strain and stress history is fed through the Wang-Brown multiaxial rainflow counting and damage procedures to get the final damage and life prediction.

Alternatively, if a full elastic-plastic time-step FE solution has been carried out, the resulting elastic-plastic strains and stresses may be used directly. This is achieved by setting "EntityDataType = StressAndStrain" on the Analysis Group properties to ensure that both the elastic-plastic strains and the stresses are recovered from the FE results, and by-passing the notch plasticity calculation in the analysis engine. The process then looks like that illustrated in Figure 5-83 below. The FE stresses and strains (still required to be surface-resolved) are used directly by the Wang-Brown multiaxial rainflow counting and damage procedures.

Fig. 5-83 Multiaxial EN calculation with elastic-plastic strains and stresses from FE



5.5.2 Material Properties

The Multiaxial EN Analysis engine uses standard strain-life material properties, which can be found in the MXD material database. These consist of a set of generic properties (which can be inherited from a generic parent) and a set of specific properties which define the shapes of the stress-strain and strain-life curves used by the E-N and multiaxial EN analysis engines.

The generic properties are as follows:

Parameter Name	Description
MaterialType	Numeric code defining the type of material. The material type is required for correct application of surface finish and stress gradient corrections.
UTS	Ultimate tensile strength. This is required for the correct definition of the upper part of the S-N curve, and to apply the static failure criterion.
YS	Yield stress. Optional (not currently used).
E	Modulus of elasticity. Required for S-N when the FE results are elastic strain.
me	Elastic Poisson ratio (defaults to 0.3 if not defined)
mp	Plastic Poisson ratio (defaults to 0.5 if not defined)
Comments	
References	

The specific E-N properties are as follows:

Parameter Name	Description
Sf’	Fatigue strength coefficient, σ_f'
b	Fatigue strength exponent
c	Fatigue ductility exponent
Ef’	Fatigue ductility coefficient, ε_f'
n’	Cyclic hardening exponent
K’	Cyclic strength coefficient
n'90	Cyclic hardening exponent for 90-degree out-of-phase multiaxial loading (defaults to a value calculated from engine property nPrimeRatio if not defined)
K'90	Cyclic hardening coefficient for 90-degree out-of-phase multiaxial loading (defaults to a value calculated from engine property KPrimeRatio if not defined)
Nc	Fatigue cutoff. Damage is set to zero beyond this point. It is set in REVERSALS. 2 reversals = 1 cycle
SEe	Standard error of log(plastic strain)
SEp	Standard error of log(elastic strain)
SEc	Standard error of log(cyclic strain)
Ne	Endurance limit–in reversals.

Parameter Name	Description
FSN	Fatemi-Socie parameter (not currently used)
S	Wang-Brown parameter—defines the slope of the Wang-Brown constant life contour for Case A loadings (defaults to 1.0 if not defined)

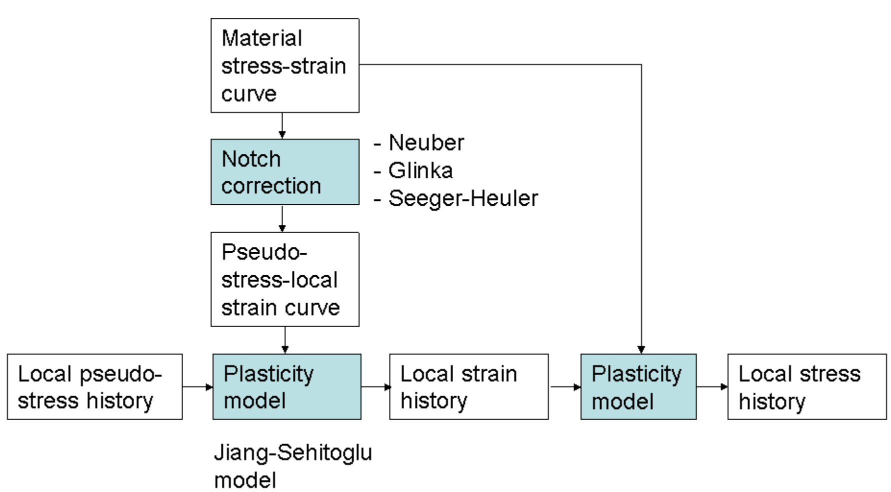
5.5.3 Multiaxial Plasticity and Notch Correction

The majority of local-strain (EN) fatigue life predictions are made, for reasons of speed and efficiency, on the basis of linear elastic FE calculations. The stresses and strains from such calculations are sometimes referred to as pseudo-stresses and pseudo-strains because they do not take into account the redistribution of stress and strain due to local yielding around stress concentrations. Because a local strain fatigue calculation requires that the local total (elastic-plastic) strains and stresses are known, it is necessary to have procedures for estimating these based on the pseudo stress or strain history. When the loading is essentially uniaxial or proportional, this can be handled by stress-strain tracking procedures in combination with notch correction methods such as the Neuber or Hoffmann-Seeger methods. These methods are described in the section "Standard EN Analysis Engine" on page 145.

Basic Principles

The basic process for the multiaxial plasticity and notch correction is illustrated in Figure 5-84 below:

Fig. 5-84 Multiaxial plasticity and notch correction procedure



The method separates the notch correction procedure from the multiaxial plasticity model, and uses two passes through the plasticity model. This general approach has been described by Lee, et al.

(Lee, Y. L., Chiang, Y. J. and Wong H. H., "A Constitutive Model for Estimating Multiaxial Notch Strains," ASME Journal of Engineering Materials and Technology, Vol. 117, pp. 33-40, 1995).

The multiaxial plasticity model is a component that takes a stress history and calculates the corresponding strain history, or vice versa. The plasticity model has to be initialized with a set of material properties. The most significant of these properties is the set of parameters describing the uniaxial cyclic stress-strain curve.

In this procedure, the notch correction is applied to the uniaxial cyclic-stress strain curve to generate a new stress-strain curve that is an estimated pseudo stress-local strain curve. This curve is used to initialize the plasticity model. The basic premise of this approach is that the plasticity model thus initialized can take the pseudo-stress history and predict the local strain history. The same plasticity model is then re-initialized using the original stress-strain curve, and the estimated local strain history is passed in so that the local stresses may be predicted.

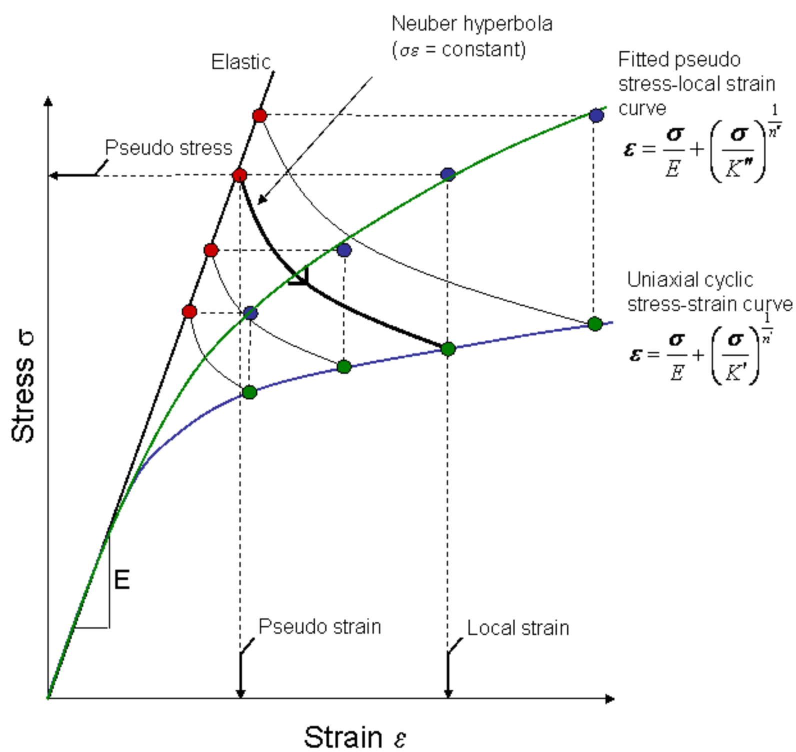
The next sections outline the theory behind the essential components of this process, namely the notch corrections and the plasticity model.

Notch Corrections

In the Multiaxial EN Engine, the notch corrections are used to determine a pseudo stress-local strain curve, based on the material uniaxial cyclic stress-strain curve.

This new curve is used to initialize the plasticity model. The general procedure is illustrated in Figure 5-85 below.

Fig. 5-85 Application of Neuber correction to determine pseudo stress-local strain curve



The starting point for the procedure is a series of pseudo strain values. (In the software, the values used are 0.0005, 0.001, 0.002, 0.005, 0.01 and 0.02.) These strain values are converted to pseudo stress by multiplying by the elastic modulus. The corresponding local strain values are then calculated using the chosen notch correction.

None

This method takes the unmodified stresses or strains and passes them through the plasticity model.

Neuber

(Neuber, H., 1961, "Theory of Stress Concentration for Shear-Strained Prismatical Bodies with Arbitrary Non-linear Stress-Strain Law," ASME Journal of Applied Mechanics, Vol. 28, pp. 544-550.)

The local strain is estimated by solving the following equations for the strain ε:

$$\sigma\varepsilon = E\left(\varepsilon_{pseudo}\right)^2$$

$$\varepsilon = \frac{\sigma}{E} + \left(\frac{\sigma}{K'}\right)^{\frac{1}{n'}}$$

Seeger-Heuler

(Seeger, T. and Heuler, P., 1980, "Generalized Application of Neuber's Rule," Journal of Testing and Evaluation, Vol. 8, No. 4, pp. 199-204.)

For the Seeger-Heuler method, the equations to be solved are:

$$\sigma\varepsilon = E\varepsilon_{pseudo} \cdot \alpha_P \cdot \left(\frac{\varepsilon_{pseudo}}{\alpha_P} + \left(\frac{E\varepsilon_{pseudo}}{\alpha_P K'}\right)^{\frac{1}{n'}} \right)$$

$$\varepsilon = \frac{\sigma}{E} + \left(\frac{\sigma}{K'}\right)^{\frac{1}{n'}}$$

...where α_p is the shape factor.

Glinka

(Molski, K. and Glinka, G., 1981, "A Method of Elastic-Plastic Stress and Strain Calculation at a Notch Root," Materials Science and Engineering, Vol. 50, pp. 93-100.)

The Glinka method equates the strain energy at the notch root with the pseudo-elastic strain energy, i.e., we need to solve:

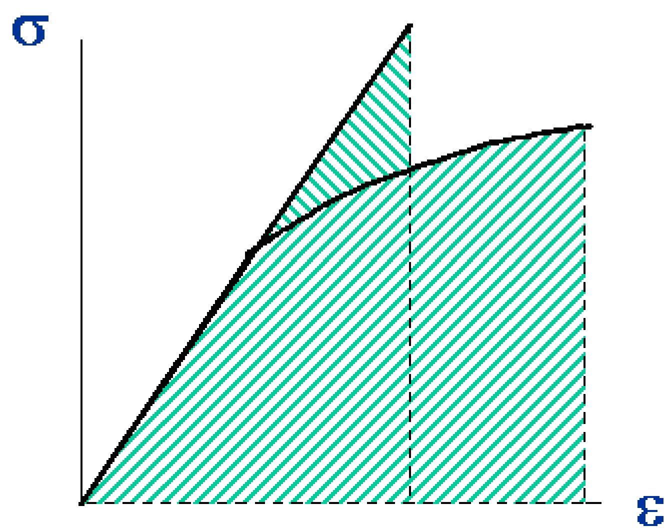
$$\sigma_{local}\left(\frac{\sigma_{local}}{E} + \left(\frac{\sigma_{local}}{K'}\right)^{\frac{1}{n'}}\right) - \int_{\sigma=0}^{\sigma_{local}} \left(\frac{\sigma}{E} + \left(\frac{\sigma}{K'}\right)^{\frac{1}{n'}}\right) d\sigma = \frac{1}{2}\frac{\sigma_{local}{}^{2}}{E}$$

Then the local strain is given by:

$$\varepsilon = \frac{\sigma_{local}}{E} + \left(\frac{\sigma_{local}}{K'}\right)^{\frac{1}{n'}}$$

Graphically, the areas under the elastic and elastic plastic curves at the loading point are equated, as illustrated in Figure 5-86.

Fig. 5-86 Graphical representation of Glinka Method



Whichever notch correction is used, the result is a set of pseudo stress-local strain pairs. Regression analysis is then carried out to fit the Ramberg-Osgood equation to this data to determine new parameters (K" or K'_{notch} and n'' or n'_{notch}) that define the pseudo stress-local strain curve.

Plasticity Model

Introduction

The plasticity model used in the Multiaxial EN engine is a version of that described by Jiang and Sehitoglu. The mathematical model is rather complicated, and the account given here is a summary. Please refer to the papers listed below for details.

(Jiang, Y. and Sehitoglu, H., 1996, "Modelling of Cyclic Ratcheting Plasticity, Part I: Development of Constitutive Relations," Trans. ASME, Vol. 63, pp. 720-725.

Jiang, Y. and Sehitoglu, H., 1996, "Modelling of Cyclic Ratcheting Plasticity, Part II: Comparison of Model Simulations With Experiments," Trans. ASME, Vol. 63, pp.726-733.)

The model makes a number of assumptions. These are:

- The material is homogeneous and initially isotropic,
- The temperature is constant in the sub-creep regime.
- The yield surface is rotationally invariant
- Material Parameters $b^{(i)}$, $\chi^{(i)}$ are constant for all i

As implemented here, the model requires 2D (free surface) stress states, including non-proportional hardening and ratcheting effects.

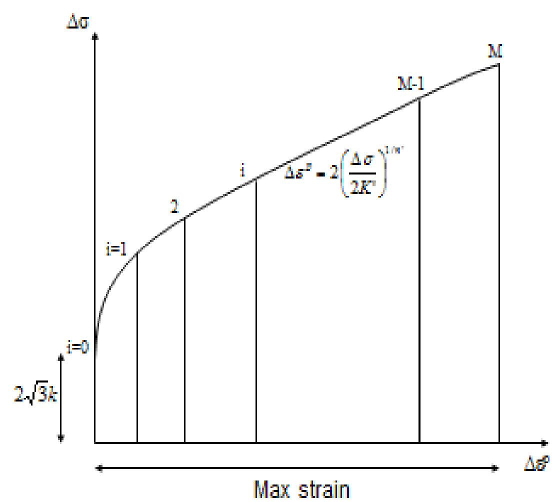
For each location, once the material has been initialized, the following steps are performed

1. Evaluate the yield surface.
2. Evolve Backstress surfaces, by the flow rule and compute the plastic modulus.
3. Calculate non-proportional hardening behaviour.
4. For each point, i in a time history, t and repeat steps 1 through 3, for each increment, where the total number of increments for each pair of time history points is determined by $Increments = \frac{(t_i - t_{i-1})}{IncrementSize}$
5. Repeat steps 1 through 4 for each point in the time history.

Material Model Initialization

The plasticity model first approximates the cyclic stress strain curve as described by the Ramberg-Osgood formulation over a series of intervals to give initial values of the material parameters c(i) and r(i), as illustrated in Figure 5-6 :

Fig. 5-87 Discretization of cyclic stress strain curve



The yield stress, k is based on the yield strain, which is defined as an analysis engine property (see later for more details).

The interval between here and the maximum allowed strain is divided up into M intervals

Logarithmically, where

$$\Delta\epsilon^p \propto YieldStrain \left(\frac{MaxStrain}{YieldStrain}\right)^{\frac{i}{M}}$$

and M corresponds to the analysis engine property "NumberOfSurfaces". Note that this curve is only defined up the value of MaxStrain, so appropriate choice of this parameter is crucial. Increasing the number of surfaces results in a better defined backstress curve, for more accurate results.

This defines a series of values of elastic strain range, which in turn define the initial values c(i):

$$c^{(i)} = \sqrt{\frac{2}{3}}\frac{2}{\Delta\epsilon_i^p}$$

The corresponding r(i) values are then calculated:

$$H_{(i)} = \frac{\Delta\sigma_{(i)} - \Delta\sigma_{(i-1)}}{\Delta\epsilon_{(i)}^p - \Delta\epsilon_{(i-1)}^p}$$

$$r^{(i)} = \frac{2}{3}\frac{\left(H_{(i)} - H_{(i-1)}\right)}{c^{(i)}}$$

The parameters $c_{(i)}$ and $r_{(i)}$ are pairs that represent the shape of the material stress-strain response under proportional loading. These pairs are dependent upon the position along the stress-strain curve, such that there exists a $c^{(i)}$ and $r^{(i)}$ for each backstress surface. In the software, these parameters are determined based on the uniaxial cyclic stress strain curve.

Yield Criterion

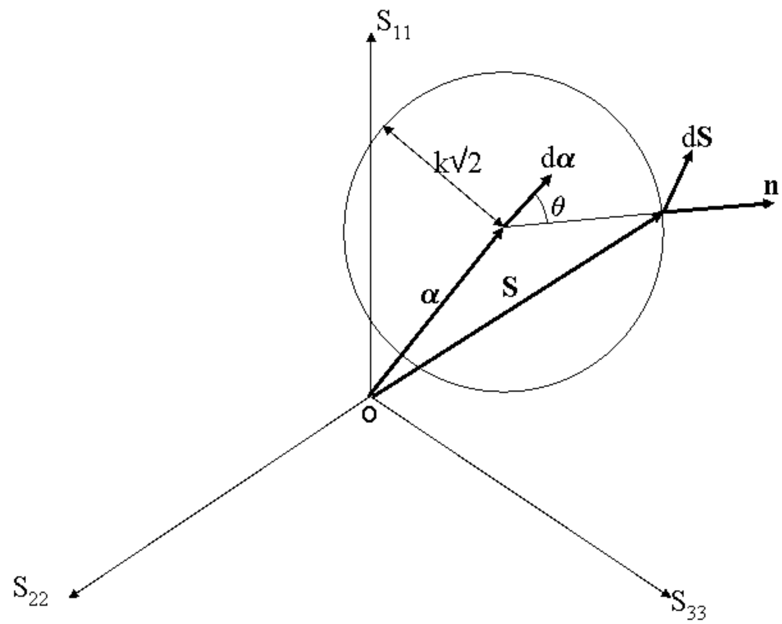
The von Mises yield function is employed, which can be written:

$$f = (\mathbf{S} - \alpha):(\mathbf{S} - \alpha) - 2k^2 = 0$$

...where **S** is the deviatoric stress tensor, α is the deviatoric backstress—the center of the yield surface f, and k is the yield stress in s simple shear. The colon (:) denotes a scalar product.

The yield surface is illustrated in *deviatoric* stress space in Figure 5-88.

Fig. 5-88 Yield surface and yield surface translation



While the current stress S lies within the yield surface, the material behavior is elastic, i.e., the stress and strain increments are related by Hooke's law.

Flow Rule

When plastic straining occurs, the plastic strain increment is collinear with the exterior normal to the yield surface at the loading point. This is the normality rule, which can be expressed:

$$d\boldsymbol{\varepsilon}^{p} = \frac{1}{h}\left\langle d\mathbf{S} : \mathbf{n} \right\rangle \mathbf{n}$$

"< >" denotes the MacCauley bracket, i.e., $\left\langle x \right\rangle = \left(\frac{x + |x|}{2} \right)$

n is the unit exterior normal to the yield surface at the loading point and h is a scalar function called the plastic modulus function.

Plastic Modulus Function

The consistency condition, which requires that the stress state lies on the yield surface during elastic-plastic deformation, leads to the following:

$$h = \sum_{i=1}^{M} c^{(i)} r^{(i)} \left[1 - \left(\frac{\|\alpha^{(i)}\|}{r^{(i)}} \right)^{\chi^{(i)}+1} \underline{L}^{(i)} : \underline{n} \right] + \sum_{i=1}^{M} \frac{\underline{\alpha}^{(i)} : \underline{n}}{r^{(i)}} \frac{dr^{(i)}}{dp} + \frac{\sqrt{2}dk}{dp} = \frac{\underline{n} : d\alpha}{dp} + \frac{\sqrt{2}dk}{dp}$$

...where:

$$dp = \sqrt{d\boldsymbol{\varepsilon}^p : d\boldsymbol{\varepsilon}^p}$$

is the equivalent plastic strain increment.

The second term in the definition of the plastic modulus function can be neglected if isotropic hardening does not need to be considered.

Hardening Rule

The hardening rule determines the evolution of the backstress (translation of the yield surface). The backstress, ? is decomposed into a number (M) of components, which evolve according to:

$$d\alpha^{(i)} = c^{(i)} r^{(i)} \left[\boldsymbol{n} - \left(\frac{\|\alpha^{(i)}\|}{r^{(i)}} \right)^{\chi^{(i)}+1} \boldsymbol{L}^{(i)} \right] dp + \frac{\alpha^{(i)}}{r^{(i)}} dr^{(i)}$$

where the unit vector associated with the i-th backstress, is expressed as:

$$\mathbf{L}^{(i)} = \frac{\boldsymbol{\alpha}^{(i)}}{\|\boldsymbol{\alpha}^{(i)}\|}$$

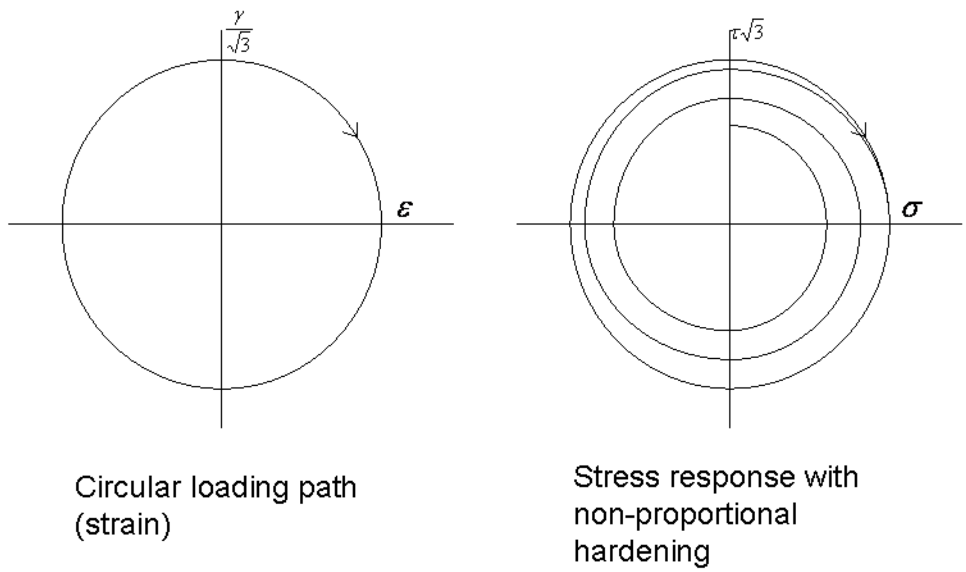
The second term in the equation forbids the backstress from leaving the bound set on $r^{(i)}$.

The ratcheting rate parameter $X^{(i)}$ has two effects-controlling the rate of ratcheting and also smoothing the transition between hardening rates as the different backstress components activate in the Jiang-Sehitoglu plasticity model. A large value will result in a stress-strain response which is effectively piecewise linear, and will also suppress ratcheting. In this implementation a single value is used for all values of i.

By default, a value of 100 is used to reduce ratcheting to a negligible level.

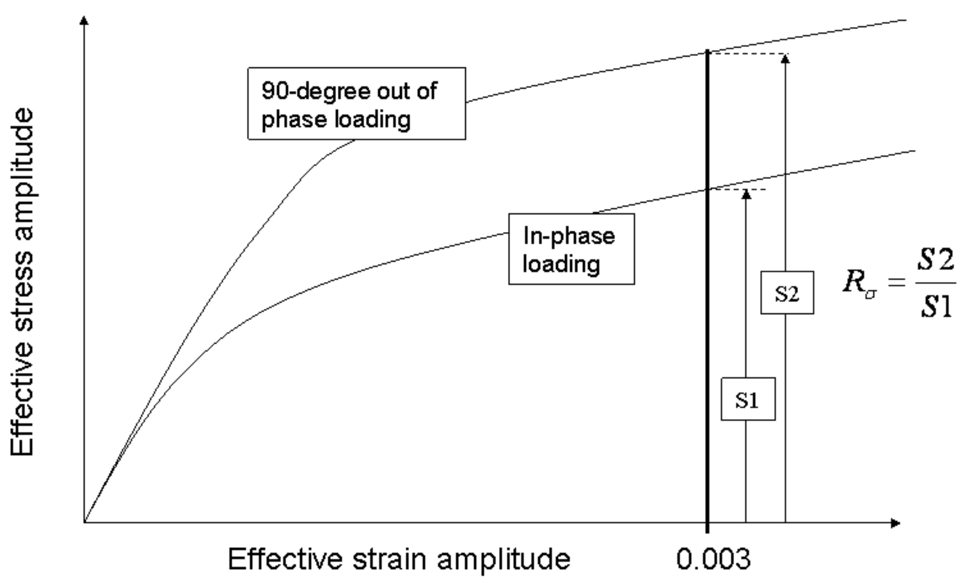
Non-proportional Hardening

Fig. 5-89 Non-proportional hardening



In general the stabilized effective stress amplitude in response to a constant amplitude strain will be greater under non-proportional loadings, with 90 degree out-of-phase loading being the worst case.

Fig. 5-90 In-phase and 90 degree out-of-phase stress-strain curves



In the software this is handled by defining different cyclic stress-strain curves for uniaxial and 90 degree out-of-phase loading. See the sections on nPrimeRatio and KPrimeRatio below.

The degree of difference between the two extremes is strongly dependent on the material type and microstructure, in particular on how easily slip systems develop and interact.

We can characterize the difference between the curves by the ratio of the stresses R_σ at an arbitrary strain level (0.003 in the software). If n’90 = n’, this ratio is simply the ratio of the K’ values, i.e. is equal to KPrimeRatio.

To model the degree of non-proportional hardening, the Jiang-Sehitoglu plasticity model employs a fourth order tensor C, as proposed by Tanaka, which is defined through:

$$dC_{ijkl} = C_c\left(n_{ij}n_{kl} - C_{ijkl}\right)dp$$

...where C_c is a material constant, with an associated non-proportionality parameter A:

$$A = \sqrt{1 - \frac{n_{\alpha\beta}C_{\varsigma\xi\alpha\beta}C_{\varsigma\xi\gamma\eta}n_{\gamma\eta}}{C_{ijkl}C_{ijkl}}}$$

We can define another material parameter N_p which derives from R_σ.

$$N_p = \sqrt{2}\left(R_\sigma - 1\right) + 1$$

Then the evolution of the yield stress k and the parameters r(i) is given by:

$$dk = b(k_0 \cdot P - k)dp$$

$$dr^{(i)} = b\left(r_0^{(i)} \cdot P - k\right)dp$$

where

$$P = (N_p - 1)A + 1$$

k0 and r0(i) are the initial values of the k (yield stress) and r(i) (as defined in the next section) and b is a material parameter, corresponding to the analysis engine property, Beee. The parameter b controls the rate at which non-proportional hardening accumulates.

k_0 and $r_0^{(i)}$ are the initial values of the k (yield stress) and r(i) (as defined in the next section) and b is a material parameter, corresponding to the analysis engine property Beee. The parameter b controls the rate at which non-proportional hardening accumulates.

5.5.4 Wang-Brown Method

The Wang-Brown method has two parts: a multiaxial rainflow cycle counting algorithm, and a path-independent multiaxial fatigue damage parameter. The method requires as input the elastic-plastic strain and stress history, resolved to the plane of the surface.

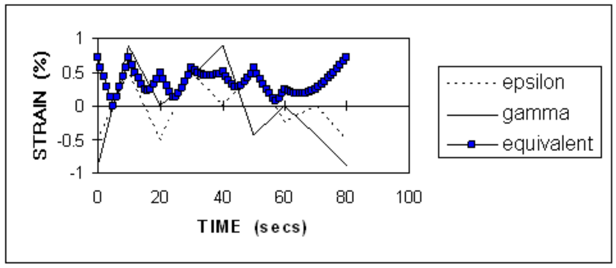
For more details, refer to the paper:

(Wang, C. H. and Brown, M. W., 1993, "A Path-Independent Parameter for Fatigue under Proportional and Non-Proportional Loading," Fatigue and Fracture of Engineering Materials and Structures, vol. 16, pp. 1285-1298.)

5.5.5 Multiaxial Rainflow Counting Method

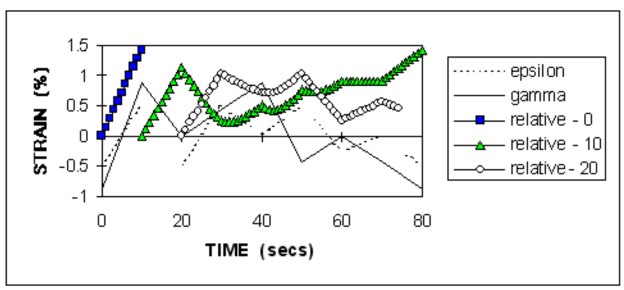
The multiaxial cycle counting method is based on strain hardening behaviour under non-proportional variable amplitude loading. The concept of relative stresses and strains is introduced so that a pair of turning points defines the start and end points of a reversal, where the equivalent relative strain rises monotonically to a peak value. Since plastic deformation generates the driving force for small fatigue cracks, hysteresis hardening provides a physical parameter for cycle counting, analogous to rainflow counting in the uniaxial case. Each reversal commences with elastic unloading, which is followed by reloading and plastic strain hardening up to the next turning point. The most significant turning point occurs at the highest value of equivalent strain. This is illustrated at time 0 in Figure 5-10, which shows a repeating block of a combined tension/torsion non-proportional load history. The equivalent strain is defined as the von Mises strain.

Fig. 5-91 Variable amplitude non-proportional strain history - applied tensile and torsional strains with absolute equivalent strain



Above is a variable amplitude non-proportional strain history, showing applied tensile (epsilon) and torsional (gamma) strains with the absolute equivalent strain The cycle counting method is illustrated by the following example. Starting from the most significant turning point, a graph is drawn for the loading block of relative equivalent strain, where relative strain $\varepsilon_{ij}^{*} = \varepsilon_{ij} - \varepsilon_{ij}^{A}$ represents the change of strain since time A. Figure 5-11 shows the relative equivalent strain, with respect to times 0, 10 and 20 seconds. Using the relative strain, a reversal can be defined starting from 0, up to the maximum value 10 seconds. To obtain the second reversal, the relative strain is re-plotted starting from the next turning point where unloading commences (at 10 seconds), and the portions of the strain-hardening curve for the reversal are selected by a traditional rainflow procedure [3]. The region of unloading and reloading within that reversal is counted in the next step.

Fig. 5-92 The variable amplitude history, showing relative equivalent strains plotted with respect to times 0, 10 and 20 seconds respectively



Using the next turning point, relative strain is re-plotted with respect to 20 seconds for the subsequent continuous fragment of strain history, yielding the third reversal in "Application of Neuber correction to cyclic loading" on page 192. This procedure is repeated for each turning point in chronological order, until every fragment of strain history has been counted. The method identifies a series of pairs of turning points, each of which represents the start and end point of a reversal (or half-cycle).

Damage Model

The counting method described above is independent of fatigue damage parameters, being based on hysteresis deformation behavior. Being unrelated to material properties, it can be integrated with any multiaxial fatigue damage model. If the counted reversals are non-proportional, a fatigue damage parameter that accounts for non-proportional straining effects is required. The path-independent damage parameter proposed by Wang and Brown has been shown to provide good correlation for several materials under proportional and non-proportional loading:

$$\hat{\varepsilon} \equiv \frac{\gamma_{\max} + S.\delta\varepsilon_n}{1 + \nu' + S(1 - \nu')} = \frac{\sigma'_f - 2\sigma_{n,mean}}{E}(2N_f)^b + \varepsilon'_f(2N_f)^c$$

...where γ_{max} is the maximum shear strain amplitude on a critical plane (proportional or non-proportional), $\delta\varepsilon_n$ is the normal strain excursion between the two turning points of the maximum shear strain (that is the range of normal strain experienced on the maximum shear plane over the interval from start to end of the reversal), and $\sigma_{n,mean}$ is the mean stress normal to the maximum shear plane. The term S is a material constant determined from a multiaxial test (typically between 1 and 2 for Case A and around 0 for Case B) and ν' is the effective Poisson's ratio. Case A loadings occur when the plane experiencing the maximum range of shear has a normal lying in the plane of the surface (negative biaxiality ratios), whereas in Case B loadings the normal to the plane of maximum shear is inclined at 45 degrees to the free surface.

The right side of the equation is the same as the uniaxial strain life equation, with a Morrow mean stress correction. Mean stress is measured as the average of the maximum and minimum stress values over the reversal. The total damage induced by a loading history is calculated using Miner's rule.

Note that under general non-proportional multiaxial loading conditions

1. Closed rainflow cycles may not occur.
2. Each reversal may have a different critical plane (plane of maximum shear strain amplitude).

5.5.6 Analysis Engine Properties

The following sections describe in detail the property settings of the Multiaxial EN Analysis Engine.

ENMethod

There are two options here:

- Standard
- MultiTemperatureCurve

Both methods use the same analysis procedures as described above but differ in how the material properties and obtained.

Standard

This option uses the standard strain life material properties as described above.

MultiTemperatureCurve

MultiTemperatureCurve datasets provide a set of E-N curves at different temperatures. The fatigue curves for intervening temperatures are determined by interpolation. Each dataset consists of a generic parent, and a number of children containing the temperature sensitive data.

For more detailed information, see "MultiTemperatureCurve" on page 178.

DamageMethod

There are two options:

WangBrown

The Wang-Brown method will be used as described above, but omitting the mean stress term ($-2\,\sigma_{n,mean}$).

WangBrownWithMean

The Wang-Brown method will be used, including the mean stress term.

PlasticityModel

There are two options:

- **JiangSehitoglu**: The Jiang-Sehitoglu plasticity model is used, in conjunction with the chosen notch correction.
  Note that this option requires 2-D elastic stress input.
- **None**: There will be no notch or plasticity correction. This option requires 2-D total strain and stress input. The Multiaxial EN Engine will assume that these are elastic-plastic strains and stresses.

NotchCorrection

If *PlasticityModel=JiangSehitoglu*, one of the previously described notch corrections is applied. They are:

- None
- Neuber
- Glinka
- Seeger-Heuler

Note that the Seeger-Heuler method requires the definition of a shape factor. If this is not defined on the material map a default value, typically 3.0, will be assumed.

CertaintyOfSurvival

The certainty of survival (%) is applied in the same way as in the Standard EN Analysis Engine. The standard error parameters (SEe and SEp) are used to determine the life associated with a particular design criterion (certainty of survival). Referring to Table 5-5 on page 133 (in the S-N analysis section), the desired number of standard deviations z from the mean is determined, based on the certainty of survival. For example, if a certainty of survival (design criterion) of 97.7% is desired, z = -2. The strain-life curve is then shifted downwards as follows:

$$\hat{\varepsilon} \equiv \frac{\gamma_{\max} + S.\delta\varepsilon_n}{1 + \nu' + S(1 - \nu')} = \frac{\sigma'_f - 2\sigma_{n,mean}}{E}(2N_f)^b \cdot 10^{z \cdot SEe} + \varepsilon'_f (2N_f)^c \cdot 10^{z \cdot SEp}$$

ScaleFactor

The scale factor is a multiplier applied to the stress or stress and strain before processing through the analysis engine.

EventProcessing

A duty cycle is a sequence of events to be processed through the analysis engine. The creation and types of different duty cycle are described in the section on Loading. Duty cycles can be processed by the Multiaxial E-N Analysis engine in two different ways, which can be summarized as follows:

- **Independent** mode. Each unique event in the duty cycle is processed separately, and the damage from each is multiplied by the number of times it appears in the complete sequence. The total damage is the sum of the damage from all the different events. This is the faster method, but may miss significant reversals that begin and end in different events, as well as failing to capture the full stress-strain response of the material.
- **CombinedFull** mode. All the events of the duty cycle are concatenated, including all repeats of each event, and the resulting stress/strain history is processed as if it were one long event. This is more accurate, but due to the computationally intensive nature of the multiaxial fatigue calculations, is not advised for very long loading histories.

OutputEventResults

If this property is set to True, the accumulated damage (and other calculated results) associated with each individual event from the duty cycle will be output in addition to the total damage. If EventProcessing=CombinedFull, some reversals may cross events. The damage associated with these events is split equally between the two events with which it is associated.

DamageFloor

This parameter sets a lower limit on the calculated damage. If a reversal has less predicted damage, the damage value is set to zero. This parameter is somewhat redundant, as similar functionality may be accessed using the fatigue cutoff Nc in the material definition.

MaxDamage

This parameter sets a numerical upper limit on the damage that can be predicted for each cycle.

TimeHistoryProcessing

The stress-strain response to non-proportional multiaxial loading can be complex, including transient behavior and ratchetting, so that if a particular loading history is repeated several times, the material behavior may continue to evolve as the history is repeated. For this reason, three options are provided for processing the time history.

- **FromStart**: A single pass will be made through the loading history, starting at the first point. This option should be selected only if you are interested in the transient behavior during the first application of the loading history.
- **FromMaxVonMises**: This option finds the point in the loading history where the von Mises stress is a maximum, and the loading history is processed starting at this point, wrapping back around to the beginning of the loading to ensure that the entire history is processed. The rationale behind this is that by starting at the maximum von Mises stress, the plasticity model will be taken to its extreme loading, and the subsequent response will be more representative of what would happen were the loading to be repeated.
- **Stabilized**: The analysis engine will make several passes through the loading history until the stress response has stabilized. The criterion for this depends on the value of HardeningTolerance (see below). The fatigue calculation is made based on the final pass through the loading history, once the criterion for stabilization has been met.

CheckMaxStrain

This property determines the behaviour that occurs when MaxStrain is exceeded.

If this strain level is exceeded, further calculations on that node are invalid.

It has two options:

- **Warn** The damage on the given node is set to StrainLimitDamage, and the calculation continues, on the next node.
- **Stop** (recommended) The analysis aborts with an error.

StrainLimitDamage

This parameter sets a numerical value that may be set for the damage to a particular node or element in the event that the equivalent strain exceeds the strain limit defined by the property MaxStrain. This property applies only if the Jiang-Sehitoglu plasticity model is being used.

The following properties control the behavior of the Jiang-Sehitoglu plasticity model. While some of these properties may be thought of as material properties, they are not widely available or easily obtainable from standard material tests. For this reason, they have not at this point been included in the standard EN material definition, and are instead set as properties on the Multiaxial EN Engine, with reasonable defaults.

NumberOfSurfaces

This parameter controls the number of surfaces, and the corresponding number of components of backstress used in the Jiang-Sehitoglu plasticity model. The Jiang-Sehitoglu model approximates the shape of the stress-strain curve (Ramberg-Osgood formulation) using a number of increments. The number of surfaces corresponds to the number of increments. More increments will in principle result in a more precise calculation. The maximum allowable value is 21. Ten surfaces seem to be reasonable.

MaxStrain

The Jiang-Sehitoglu plasticity model requires that a maximum strain be set. If this strain level is exceeded, further calculations are invalid. This property defines that limit, in microstrain ($\mu\varepsilon$). If this happens during calculation, the calculation will stop and a damage value equal to StrainLimitDamage will be set.

YieldStrain

In the plasticity model, within the yield surface, the stress-strain response is linear elastic. By setting the yield strain in microstrain ($\mu\varepsilon$), this parameter effectively controls the size of the yield surface. Note that this should be thought of as a numerical parameter controlling the behavior of the plasticity model, not a physical yield strain or material property. Setting a value equivalent to the static yield or proof strain will be much too large. A value of 200 $\mu\varepsilon$ seems reasonable.

In the plasticity model, within the yield surface, the stress-strain response is linear elastic. By setting the yield strain in microstrain ($\mu\varepsilon$), this parameter effectively controls the size of the yield surface. Note that this should be thought of as a numerical parameter controlling the behaviour of the plasticity model, not a physical yield strain or material property. Setting a value equivalent to the static yield or proof strain will be much too large.

It is recommended that the YieldStrain is at least two orders of magnitude less than the MaxStrain.

IncrementMethod

This option determines how the time history is incremented.

It has two options:

- **Single** (recommended) The increment size parameter is used to set a uniform step size between strain values. When this method is chosen it is essential that the increment size is significantly less than the YieldStrain, to avoid anomalous results due to poorly sampled stress-strain curve.

- **Double** Two different step sizes are used across the time history. When the change in strain keeps the point within the yield surface then InternalIncrementSize is used, otherwise ExternalIncrementSize is utilised. When this method is used, the InternalIncrementSize can be of a dimension similar to that of the YieldStrain, provided that the ExternalIncrementSize is small. This method can provide a substantial speed increase, with some loss of accuracy.

IncrementSize

The Jiang-Sehitoglu plasticity model calculates the stress response to a strain increment and vice versa. Consideration of large increments can lead to a loss of accuracy in the response, so large increments are subdivided internally. This property defines the maximum allowable increment size in microstrain ($\mu\varepsilon$). A value of no more than 10 $\mu\varepsilon$ is suggested. Smaller increment sizes will result in increased precision at the expense of computation time.

ExternalIncrementSize

The Jiang-Sehitoglu plasticity model calculates the stress response to a strain increment and vice versa. Consideration of large increments can lead to a loss of accuracy in the response, so large increments are subdivided internally. This property defines the maximum allowable increment size, outside the yield surface in microstrain ($\mu\varepsilon$). Smaller increment sizes will result in increased precision at the expense of computation time.

For best results, it is recommended that the ExternalIncrementSize should be two orders of magnitude less than the YieldStrain, with a value of no more than 1 $\mu\varepsilon$ is suggested.

InternalIncrementSize

The Jiang-Sehitoglu plasticity model calculates the stress response to a strain increment and vice versa. Consideration of large increments can lead to a loss of accuracy in the response, so large increments are subdivided internally. This property defines the maximum allowable increment size) inside the yield surface, in microstrain ($\mu\varepsilon$). Smaller increment sizes will result in increased precision at the expense of computation time.

This parameter will be ignored if it is set to be smaller than the ExternalIncrementSize, otherwise it is suggested that the InternalIncrementSize is of the same order of magnitude as the YieldStrain, for an increase in computation speed.

Beee

This is a dimensionless scalar parameter, which is known as the non-proportional rate parameter. It controls the rate of non-proportional hardening in the Jiang-Sehitoglu plasticity model. In reality, this is a material dependent parameter. In this implementation, it is assumed to be constant for along the cyclic stress-strain curve. Positive values promote increased hardening of the material, whilst negative values lead to softening. A value of 5.0 appears to be reasonable. A value of zero is not recommended as it prevents any evolution of the yield surface, k or stress-strain response $r^{(i)}$.

Chi

This is a dimensionless parameter positive scalar parameter called the ratcheting rate parameter. It has two effects—controlling the rate of ratcheting, and also smoothing the transition between hardening rates as the different backstress components activate in the Jiang-Sehitoglu plasticity model. In this implementation, it is assumed to be constant for along the cyclic stress-strain curve. A large value will result in a stress-strain response that is effectively piecewise linear, and will also suppress ratcheting. By default, Chi is set to 100 so that ratcheting is negligible.

HardeningTolerance

This parameter is relevant only when TimeHistoryProcessing=Stabilised. In this case, the software will make several passes through the time history until the stress/strain response has stabilized.

This parameter determines the tolerance that will be applied. This condition allows for the occurrence of both hardening and softening. The default value of 0.05 corresponds to a difference of ±5%, between the yield stress from the current and previous passes.

KPrimeRatio

The Jiang Sehitoglu plasticity model can predict the extra hardening that occurs in some materials as a result of non-proportional loading. This effect is captured in the material properties by having different values of K' and n' for proportional and 90-degree out-of-phase non-proportional loadings. K'90 and n'90 are material properties. If K'90 is not defined in the material dataset it is calculated from:

$K'90 = K' \cdot K'Ratio$ (KPrimeRatio)

The default value is 1.0.

nPrimeRatio

If n'90 is not defined in the material dataset it will be calculated from:

$n'90 = n' \cdot n'Ratio$ (nPrimeRatio)

The default value is 1.0.

If KPrimeRatio = nPrimeRatio = 1.0, no non-proportional hardening will be predicted.

Temperature Selection

This parameter is only used if the ENMethod is set to MultiTemperature Curve, the temperature data is supplied by a temperature load provider and the Mode property of the temperature load provider is set to All. It has three options:

- **Overall Max** - An isothermal calculation is done using the maximum value of the temperature data.
- **Overall Min** - An isothermal calculation is done using the minimum value of the temperature data.
- **Overall Median** - An isothermal calculation is done using the median value of the temperature data. The median is defined as (Max + Min)/2.

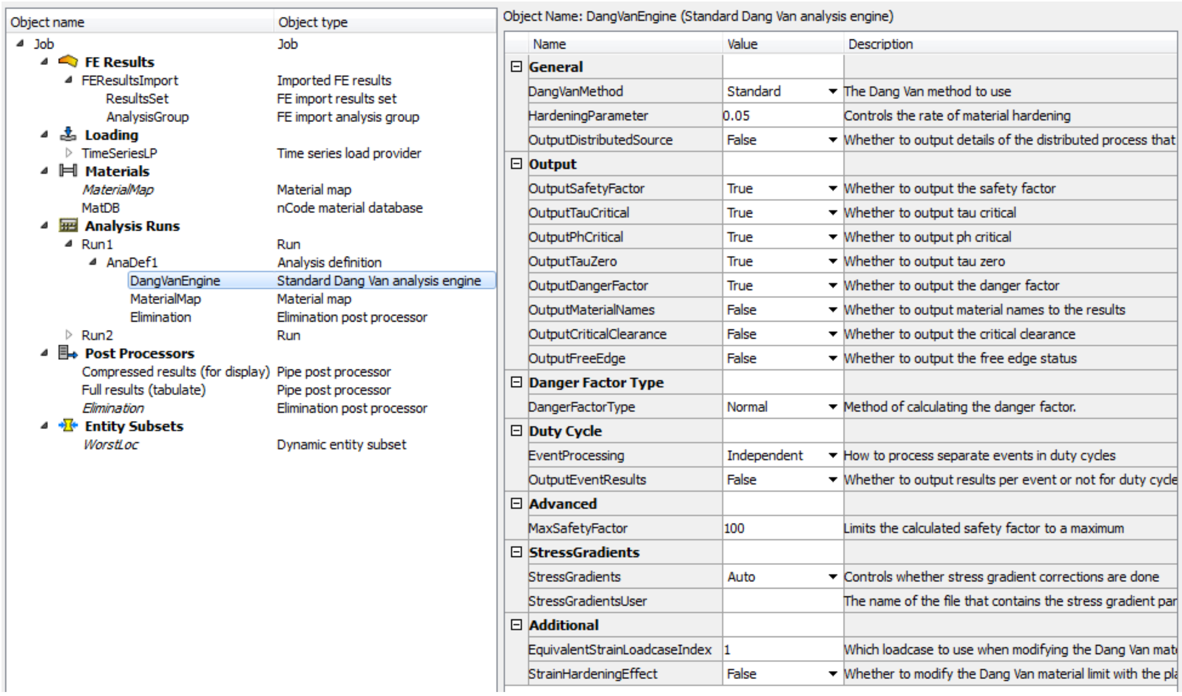
Temperature Interpolation Limit

This parameter is only used if the ENMethod is set to MultiTemperatureCurve for an isothermal or cycle-by-cycle temperature corrected calculation. If the temperature value is above the highest curve temperature of the material data set, then this parameter determines whether to extrapolate or not. It has two options:

- **UseMaxCurve** - No extrapolation is done and the highest temperature curve is used.
- **Extrapolate** - The two highest temperature curves are used to extrapolate to the required temperature value.

5.6 Standard Dang Van Analysis Engine

Fig. 5-93 Standard Dang Van analysis engine properties



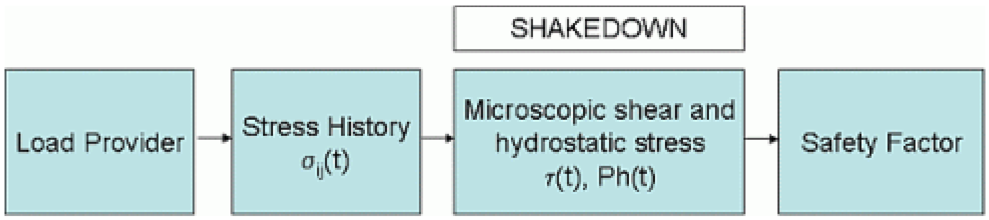
5.6.1 Summary

The Dang Van method is designed to provide a safety factor calculation for multi-axially loaded components in the endurance regime (large number of cycles to failure). The theoretical basis can be summarized as follows:

- Fatigue crack initiation normally occurs due to repeated plasticity on shear planes in individual grains.
- The most important factor as to whether a crack will propagate past the first grain and go on to cause fatigue failure is therefore the microscopic shear stress in critical grains.
- The ability of a shear crack to propagate is modified by the hydrostatic stress (which can increase damage by opening existing cracks).
- The state of stress in a grain where repeated plasticity is occurring will be affected by a process of shakedown.

The Dang Van analysis process is summarized in Figure 5-94:

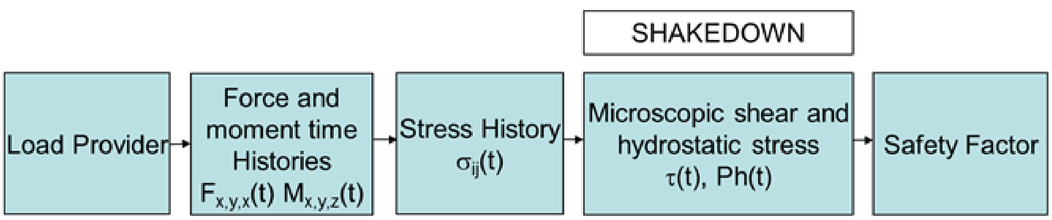
Fig. 5-94 Dang Van analysis process summary



The Dang Van analysis engine requires an FE-derived stress-time history as input. It does not support the Vibration Load Provider.

The Spot Weld Dang Van analysis process is summarized in Figure 5-95:

Fig. 5-95 Dang Van analysis (spot weld) process summary



The Dang Van Analysis method "Spotweld" calculates the stress history from spot weld forces and therefore requires FE-derived Force/Moment time histories as input.

This analysis calculates the stress based on the forces in the spot weld and a set of calculation parameters which are in a file *spotweld_DV.sys* which can be found in the *nssys* directory of the DesignLife installation. The calculation is only carried out for 2 and 3 sheet welds. For 3 sheet welds a calculation of the alignment of the elements representing the two weld nuggets is carried out and if they are mis-aligned by more than 5 degrees then no fatigue calculation is carried out.

This Dang Van spot weld calculation can be carried out using any of the currently supported FE modelling methods but is only validated for directly connected beam elements in Nastran and Abaqus.

5.6.2 Dang Van Material parameters

The material parameters required for a Dang Van analysis are as follows:

Material Parameter	Description
MaterialType	Numeric code defining the type of material
UTS	Ultimate tensile strength
E	Modulus of elasticity
TAFE	Type A fatigue endurance (MPa). This is the shear stress amplitude at the fatigue limit under pure shear (torsion) loading, usually denoted "b" in discussions of the Dang Van method.
HSS	This is the hydrostatic sensitivity factor, usually denoted "a" in discussions of the Dang Van method.
LHF	The limiting hardening factor–used to adjust the fatigue endurance to take into account the forming strain accumulated in the manufacturing process. This parameter, usually denoted "C" in discussions of the Dang Van method, may be left undefined if the effect of forming strain is not to be included in the calculation.
SHF	The strain hardening factor, usually denoted k in discussions of the Dang Van method. This may be left undefined unless the effect of forming strain is to be taken into consideration.
Comments	
References	

The Dang Van parameters a and b (TAFE and HSS) define a fatigue threshold condition which is described by the equation:

$$\tau + a.Ph = b$$

...where τ is the microscopic shear stress and Ph is the hydrostatic pressure.

The material parameters *a* and *b* can be determined if the fatigue stress limit amplitude is known under two different loading conditions, with the most common being uniaxial tension-compression or bending and pure shear, both at stress ratio R = -1.

DesignLife uses the parameters TAFE and HSS, which represent respectively a torsional fatigue limit and a hydrostatic stress sensitivity factor.

In the case of a pure shear test, at the fatigue limit, Ph = 0 and τ = b = TAFE.

For a tension-compression or bending test, at the fatigue limit f, Ph = f/3 and τ = f/2. This leads to: $a = \frac{3(TAFE - 0.5f)}{f} = HSS$

Fig. 5-96 Dang Van diagram example definition

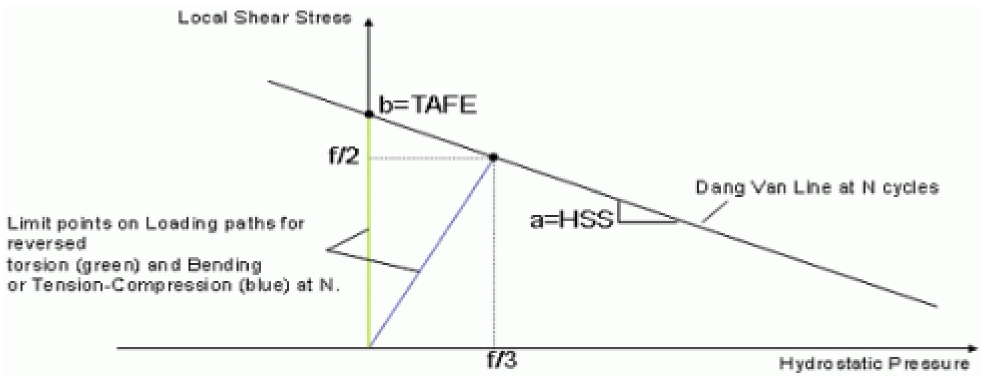
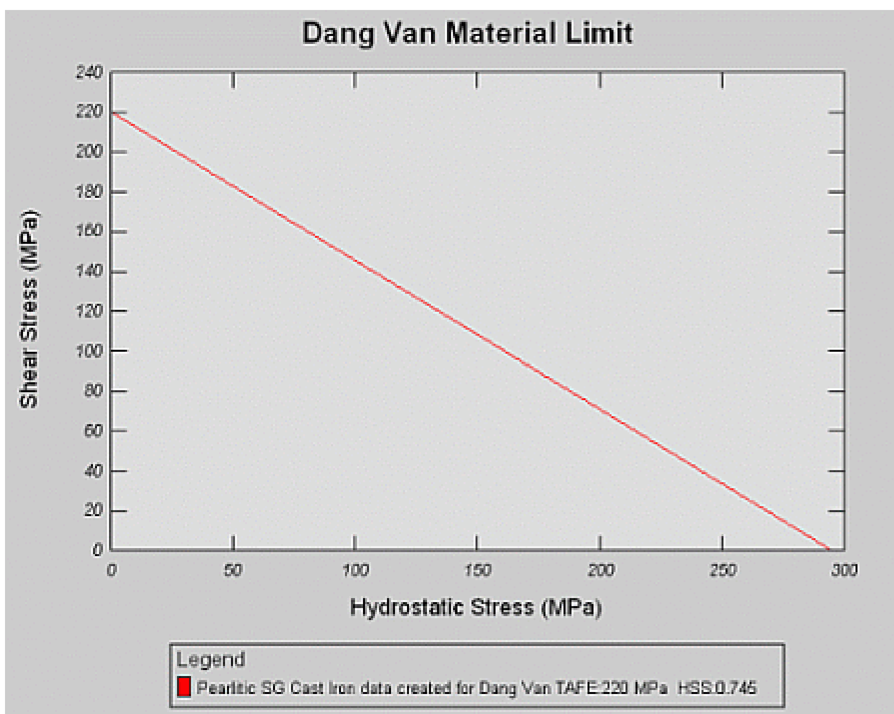


Fig. 5-97 Dang Van example plot from nCode Materials Manager



5.6.3 The Dang Van criterion

The Dang Van criterion is a three-dimensional multiaxial fatigue limit criterion dealing with high-cycle fatigue conditions where the crack initiation occurs at a microscopic level. The method assumes that around the fatigue limit, cyclic plasticity will occur in critically oriented individual grains, and therefore local stress tensors need to be considered. These microscopic stresses differ from the macroscopic ones by the presence of a microscopic residual stress field and near the fatigue limit tend towards a pseudo shakedown state.

The Dang Van method states that, at the stabilized state (shakedown state), crack initiation will happen whenever a function of the microscopic stresses exceeds a threshold at any time in a stabilized cycle. The formulation uses a linear combination of the current local shear stress τ(t), creating plasticity, and hydrostatic pressure p(t), acting on opening of micro-cracks, to calculate an "equivalent" stress and compare it to a limit.

Crack initiation is predicted to occur if:

$$\tau(t) + a * p(t) \geq b$$

...where *a* and *b* are material parameters calculated from two simple fatigue tests. The relation between the macroscopic stresses $\Sigma_{ij}(t)$ and the microscopic ones

$\sigma_{ij}(t)$ is given by: $\sigma_{ij}(t) = \Sigma_{ij}(t) + \rho_{ij}^{*}$

ρ_{ij}^{*} being the stabilized local residual stress tensor. Dang Van demonstrated that this tensor is dependent on the microscopic plastic strains, so it is a deviatoric.

The microscopic hydrostatic pressure is then equal to the macroscopic one:

p(t) = P(t) = 1/3 * trace $[\Sigma_{ij}(t)]$

Considering the deviatoric stresses:

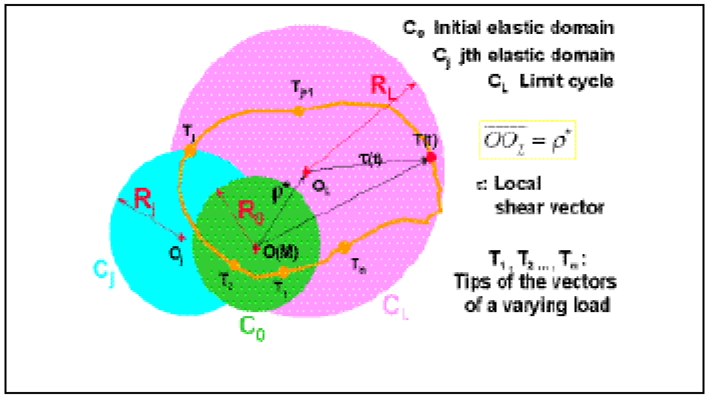
$$s_{ij}(t) = S_{ij}(t) + \rho_{ij}^{*}$$

Based on the work of Papadopoulos, Dang Van proposed in 1987 to take $\rho_{ij}^{*} = -z^{*}_{ij}$, z^{*}_{ij} being the center of the smallest hyper sphere that encompasses the path described by the macroscopic deviatoric stress tensor $S_{ij}(t)$. It is in fact the solution of the min-max problem:

$$z_{ij}^{*} : \min_{z_{ij}} \left\{ \max_{t} \sqrt{(S_{ij}(t) - z_{ij}) : (S_{ij}(t) - z_{ij})} \right\}$$

The microscopic residual stress is calculated using an iterative procedure that attempts to simulate the process of elastic shakedown. This is practically achieved by simulating a combination of isotropic and kinematic hardening, as illustrated below (in 2-D):

Fig. 5-98 2-D representation of hyper sphere evolution



A small initial yield surface is defined, denoted C_0. When the loading path contacts the yield surface, this surface is allowed to move (kinematic hardening) and expand (isotropic hardening) and the process continues cycling through the loading history until there is no further expansion or movement of the yield surface (convergence). The displacement of the center of the yield surface from the origin then defines the microscopic residual stress. A hardening parameter controls the rate of isotropic hardening and it is obvious that convergence will be more accurate if the sphere radius grows slowly. Therefore, for accuracy, the isotropic hardening coefficient should be as small as possible. On the other hand, smaller values mean longer calculation duration, so a compromise needs to be found. A value of 0.05 for this hardening parameter is sufficient for most loading histories; however, when the loadings are very short (only a few points), or constant amplitude loading is used, a smaller value will give greater accuracy, at some cost in processing time.

Once ρ_{ij}^* is obtained, the local shear stress is determined from the microscopic deviatoric stresses, using the Tresca criterion, by:

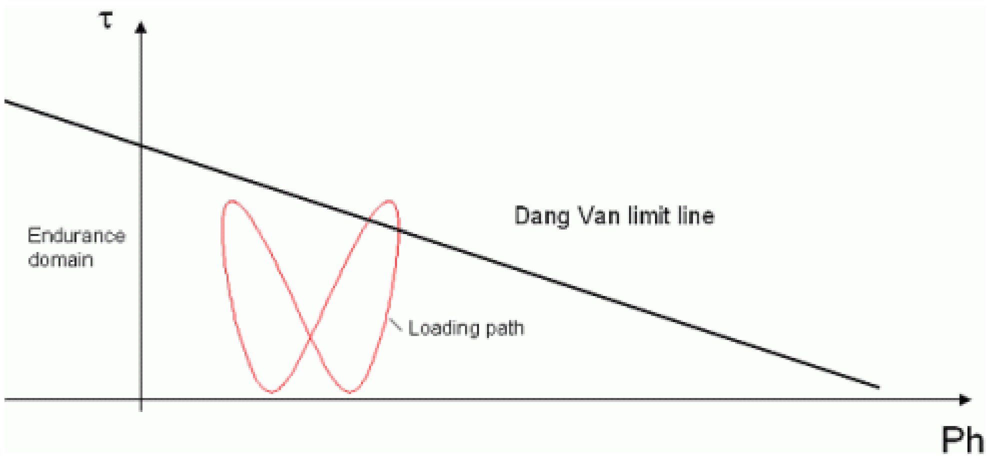
$$\tau(t) = \frac{1}{2} Tresca\{s_{ij}(t)\}$$

$$\tau(t) = \frac{s_1(t) - s_3(t)}{2}$$

...where $s_1(t)$ and $s_3(t)$ being respectively the largest and smallest principals of $s_{ij}(t)$.

The couples (P(t),τ(t)) define a loading path in the Dang Van diagram. Damage will occur only if this loading path crosses the Dang Van threshold line.

Fig. 5-99 Loading path in a Dang Van diagram



5.6.4 Determination of critical point

The critical point in the loading history is determined in different ways based on the Dang Van material type and the property DangerFactor.

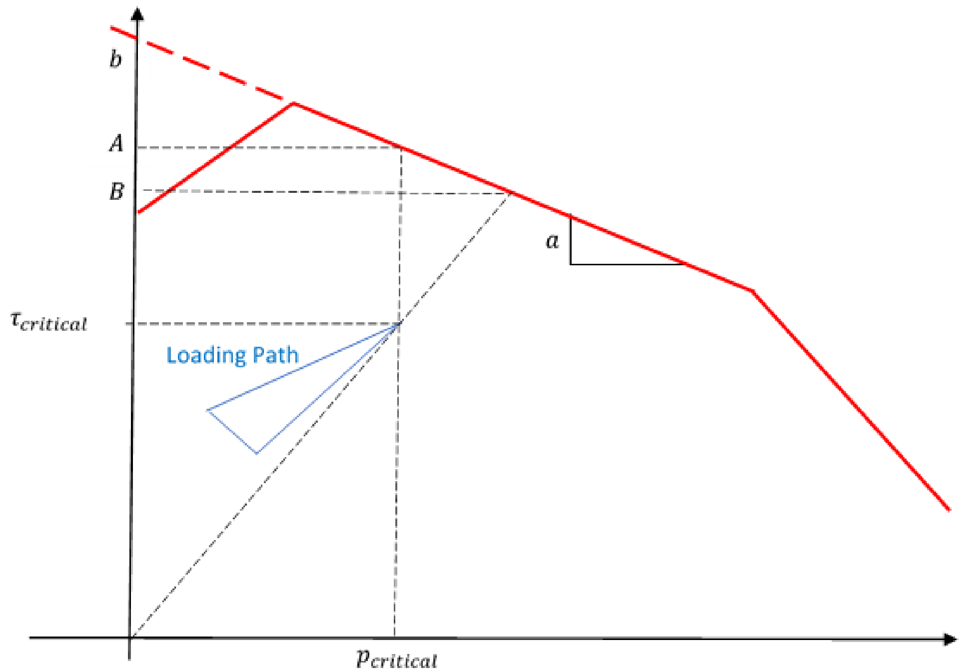
With a standard Dang Van material the safety factor is calculated for all points and the point of minimum safety factor is found. The values of Ph and Tau are taken from this point and the DangerFactor is then calculated (normal or oblique).

With a multi-segment Dang Van material two different methods of finding the critical point are used:

- DangerFactor = Normal: the point of minimum safety factor is found and this defines the critical point. The safety factor is calculated based on which material curve segment the Ph value falls under.
- DangerFactor = Oblique: the point of worst danger factor is found and this defines the critical point. The safety factor is then calculated for this point in the normal way and the danger factor is calculated using the oblique method. The DangerFactor is calculated based on the material curve segment intersected by the line from the origin through each point.

In both cases with multi-segment materials the first and last segments are extrapolated when required. The two methods are illustrated in Fig. 5-100 Determination of critical point:

Fig. 5-100 Determination of critical point



5.6.5 Sheet Cut Edge Effect

During manufacturing of sheet metal components the edges of the sheet will have their mechanical properties modified by the cutting process. This effect can now be taken into account in the Dang Van calculation. The free edges within the FE model are identified automatically (AnalysisGroup/IdentlfyCutEdges) and the side of the shell element that requires the correction is selected for each material group (Cut Edge location).

This correction is applied by reducing the material intercept a (TAFE) of the material.

$TAFE_{edge}$ = $TAFE_{material}$ * (1 - D)

...where:

D = (0.0004 * $R_{p0.2\%}$) - 0.0467

$R_{p0.2\%}$ in MPa.

Another parameter called the critical clearance $J_{c\%}$ is also calculated for the edge elements. Its derivation is:

$$Jc_{\%} = \max\left(Abs(-5.32.e^{-4} \times Rm + 0.471) * 100;13\right)$$

UTS are in MPa.

5.6.6 Outputs of a Dang Van analysis

The outputs of a Dang Van analysis are as follows:

Safety Factor

The Dang Van safety factor *SF* is defined by:

$$SF = \frac{b}{Max\left(\tau(t) + a \times p(t)\right)}$$

TauCritical

Critical microscopic shear stress. This is the value of the microscopic shear stress at the critical point of the loading path defined by the safety factor calculation.

PhCritical

Critical hydrostatic stress. This is the value of the hydrostatic stress at the critical point of the loading path.

CriticalClearance

The cut edge property *critical clearance* can be included in the results. See "Sheet Cut Edge Effect" for its derivation.

FreeEdge

A cut edge property, indicating whether the element or node is on a free edge.

TauZero

τ_0 is the projection of the critical point of the loading path on the axis P=0, parallel to the Dang Van line.

$$\tau_0 = \tau_{critical} + a \cdot p_{critical}$$

DangerFactor

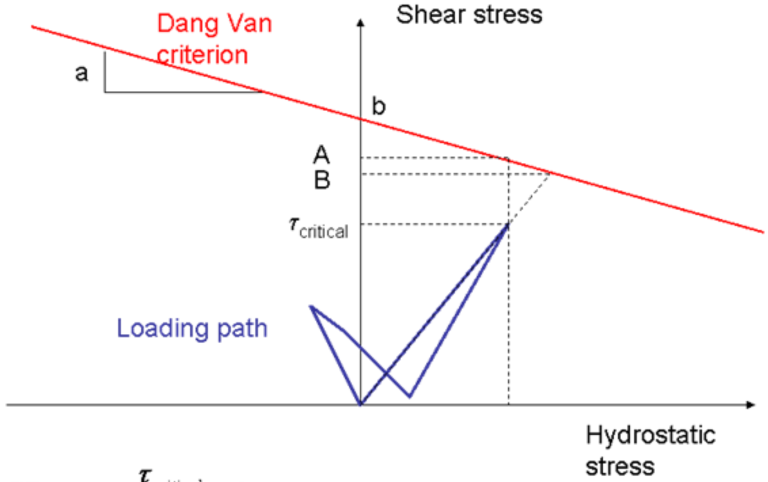
The two ways to calculate the Danger Factor are known as the Normal and Oblique methods. They are both calculated at the loading path critical point.

The "Normal" method... $DF\ (or\ CD_{Normal}) = \dfrac{\tau_{critical}}{b - a \cdot p_{critical}} - 1$

The "Oblique" method... $DF\ (or\ CD_{Oblique}) = \dfrac{\tau_{critical} + a \cdot p_{critical}}{b} - 1 = \dfrac{1}{SF} - 1$

They are illustrated in the figure below:

Fig. 5-101 Normal and Oblique definitions of the Dang Van Danger Factor (CD)



$$CD_{Normal} = \frac{\tau_{critical}}{A} - 1$$

$$CD_{Oblique} = \frac{\tau_{critical}}{B} - 1$$

5.6.7 Properties of the Dang Van analysis engine

The properties of the Dang Van analysis engine (see Fig. 5-93 Standard Dang Van analysis engine properties) are:

HardeningParameter

This parameter controls the rate of isotropic hardening, as described above. The default value is 0.05. Decreasing the value of the hardening parameter will reduce the rate of growth of the yield surface used to determine shakedown. This will lead to slower calculation, but increased accuracy.

OutputSafetyFactor

True/False: Defines whether or not to include the Dang Van safety factor in the results. See also “Safety Factor” on page 235.

OutputTauCritical

True/False: Defines whether or not to include the critical microscopic shear stress in the results. See also “TauCritical” on page 235.

OutputPhCritical

True/False: Defines whether or not to include the critical hydrostatic stress in the results. See also “PhCritical” on page 235.

OutputTauZero

True/False: Defines whether or not to include the τ_0 in the results. See the previous section for a definition of τ_0.

OutputDangerFactor

True/False: Defines whether or not to include the danger factor in the results. SSee also "DangerFactor" on page 235.

OutputCriticalClearance

True/False: Defines whether or not to include the critical clearance in the results.

See the previous section for a definition of critical clearance.

OutputFreeEdge

True/False: Defines whether or not to include the cut edge status in the results.

DangerFactorType

Normal/Oblique: Determines whether the **"**Normal**"** or **"**Oblique**"** formulation of the Danger Factor is to be calculated. This property is active only if *OutputDangerFactor=True*.

EventProcessing

This property controls the way that the Dang Van analysis engine handles duty cycles. The options here are Independent and Combined Full.

> **Independent:** Dang Van safety factor calculations are made for each event separately. Note that because of the nature of the calculations, it is only necessary to consider each unique event in the duty cycle once. In fact, the repeat count is ignored for Dang Van calculations. The overall result is the worst safety factor taken from the individual events.
>
> **CombinedFull:** Dang Van safety factor calculations are made for the complete duty cycle. Note that because of the nature of the Dang Van calculation, neither the number of repeats nor the exact sequence of events should have any significant effect on the final result (in practice they may have a minor effect on the shakedown part of the calculation).

OutputEventResults

True/False: This controls the output of results when processing duty cycles. It determines whether or not individual event results are output as well as the overall result. When using the CombinedFull option, this should be set to False.

MaxSafetyFactor

For convenience when viewing and plotting results, this property allows an upper limit to be set on the Safety Factor.

EquivalentStrainLoadcaseIndex

This property is relevant only if *StrainHardeningEffect = True*. In that case, this index tells the software from which of the available loadcases or loadsteps the equivalent plastic strain should be recovered.

StrainHardeningEffect

Many metallic components experience significant plastic strains during the manufacturing process, e.g., pressing of steel sheets. The work hardening that occurs during such a process can result in significantly improved fatigue strength, and if this effect is ignored in a fatigue calculation, the durability prediction may be unnecessarily pessimistic. If *StrainHardeningEffect = True*, the Dang Van analysis engine will modify the Dang Van parameter b (corresponding to TAFE in the material database—the shear fatigue strength at the endurance limit) to simulate the fatigue strength improvement due to work hardening.

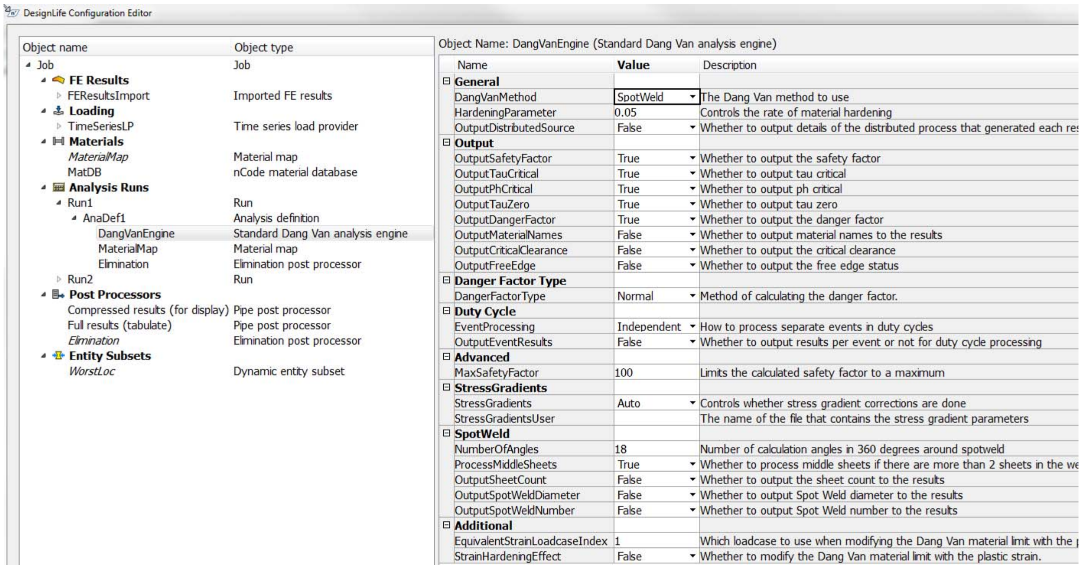
In order to use this feature, the equivalent plastic strain should be available as part of at least one of the load steps or load cases in the FE results set, and the property "*IncludeEquivalentPlasticStrain*" should be set to "True" on the FE Results Set properties.

The parameter b is modified as follows:

$$b = b_0\left(1 + C\left(1 - e^{-k\varepsilon_p^{VM}}\right)\right)$$

where b_0 is the shear fatigue strength corresponding to zero equivalent plastic strain, C is a material constant called the limiting hardening factor (representing the maximum possible improvement to the fatigue strength due to work hardening) and k is another material constant, the strain hardening factor.

Fig. 5-102 Spot Weld Dang Van Analysis engine properties



The properties of the Dang Van analysis engine when the DangVanMethod=Spotweld (see Figure 5-102 above) also include:

NumberOfAngles

In spot weld calculations, predictions of life are made at intervals around the periphery of the weld in the two sheets and in the weld nugget. This property controls the number of values of the angle ? at which the fatigue calculations are made. For example, if NumberOfAngles = 18, calculations will be made at 20 degree intervals.

ProcessMiddleSheets

True/False: The Rupp method for spot weld life prediction has not been validated for the prediction of fatigue damage occurring at the middle sheets for spot welds joining more than two sheets. It should also be noted that such failures are relatively rare, difficult to reproduce in the laboratory, and difficult to detect in practice. For this reason, the software provides an option (by setting this property to False) to omit any fatigue calculation on the middle sheets, which might otherwise give a false positive failure prediction.

OutputSheetCount

True/False: If True the count of sheets analysed with each weld group will be output to the results. For example, the weld group of a two sheet weld typically consists of two sheet plus a weld nugget. The sheet count column will have a value of 2 in this case.

OutputSpotWeldDiameter

True/False: If true, the spot weld diameter will be output to the results.

OutputSpotWeldNumber

True/False: If true, the spot weld number will be output to the results. The spot weld number is an internally generated index that can be used to identlfy which results are related to the same physical spot weld. The number does not correspond to any ID from the finite element model.

StressGradients

This property has three options::

Table 5-10 Stress gradient options

Auto	A correction for stress gradient is applied if stress gradients are present (i.e., if they were requested at the translation stage in the Analysis Group properties).
User	A user-defined stress gradient correction is applied (if stress gradients are available) based on a lookup table provided in a file. A valid file must be specified using the StressGradientsUser property.
Off	No stress gradient correction is applied.

The stress gradient correction works in exactly the same way as for the Standard S-N Analysis engine, with the correction factor being applied to the stress before stress combination, cycle extraction, rainflow cycle counting, etc.

StressGradientsUser

If StressGradients is set to User, the name of a valid file must be provided here (see Standard S-N Analysis Engine for more information).

StressGradientMethod

See 5.3.26 StressGradientMethod in “Standard SN Analysis Engine”.

OutputStressGradientFactors

See 5.3.27 OutputStressGradientFactors in “Standard SN Analysis Engine”.

5.7 Spot Weld Analysis Engine

Fig. 5-103 Properties of the Spot Weld analysis engine

Object Name: AnalysisEngineSpotWeld1 (Spot Weld analysis engine)

Name	Value	Description
General		
CombinationMethod	Standard	The method used to calculate the local stresses from joint cross sectional forces and moments
CustomCombinationMethod	<Not Available>	Specifies the customised combination method to use
SNMethod	Standard	The method used to calculate damage from a stress cycle
CustomSNMethod	<Not Available>	Specifies the customised SN method to use
MeanStressCorrection	Simple	The method used to correct the damage calculation for mean stress
NumberOfAngles	18	Number of calculation angles in 360 degrees around spotweld
ProcessMiddleSheets	False	Whether to process middle sheets if there are more than 2 sheets in the weld
CertaintyOfSurvival	50	Required confidence level on damage results
ScaleFactor	1	The scale factor to apply prior to damage calculation
CalculateTorsion	False	Whether to calculate torsion
OutputMaxMin	False	Whether to output max and min stresses
OutputMaterialNames	False	Whether to output material names to the results
OutputSheetCount	False	Whether to output the sheet count to the results
OutputSpotWeldDiameter	False	Whether to output Spot Weld diameter to the results
OutputSpotWeldNumber	False	Whether to output Spot Weld number to the results
OutputDistributedSource	False	Whether to output details of the distributed process that generated each result
BackCalculation		
BackCalcMode	None	Whether to perform a back-calculation or not
TargetDamage	1	Target damage for back calculation
BackCalcAccuracy	1	The accuracy of the back calculation
BackCalcMaxScale	5	The max scale factor to allow in back calculation
BackCalcMinScale	0.2	The min scale factor to allow in back calculation
DutyCycle		
EventProcessing	Independent	How to process separate events in duty cycles
OutputEventResults	False	Whether to output results per event or not for duty cycle processing
Advanced		
CheckStaticFailure	False	The action to take on static failure
DamageFloor	0	The calculated damage value below which the damage is set to zero
MaxDamage	1E30	The maximum damage value
StaticFailureDamage	1E10	The damage value to insert on static failure

5.7.1 Background

Introduction and Modeling Guidelines

The method implemented in nCodeDT for the fatigue analysis of spot welds is based closely upon the work of Rupp, Störzel and Grubisic [*Rupp, A., Störzel, K. and Grubisic, V. (1995) "Computer Aided Dimensioning of Spot-Welded Automotive Structures". SAE Technical Paper 950711.*].

The method as originally conceived requires each spot weld to be represented by a CBAR element in NASTRAN, joining two sheets of shell elements. The forces (Element Forces) transmitted through these CBAR elements are used to calculate the structural stresses in the weld nugget and in the adjoining sheet metal at intervals around the perimeter of each weld. These stresses can then be used to make fatigue life predictions on the spot weld using a S-N (total life) method.

In recent times, the method has been adapted to allow the use of different FE analysis codes, and different mesh joining techniques (mainly so-called "Area Contact Methods") to represent the spot welds. In these cases the principles remain the same, i.e., cross-sectional forces and moments are used to derive stresses for use in a stress-life fatigue calculation.

Using NASTRAN element types, if CBAR (or CBEAM) elements are being used, each spot weld should be represented by a single stiff CBAR element joining 2 sheets of shell elements, and perpendicular to both. The CBAR elements should be sufficiently stiff that the stiffness of the joint is not sensitive to the CBAR properties. In practice, this can be readily achieved by giving the CBAR element the same modulus of elasticity as steel (for welds in steel), and dimensions of the cross section approximately equal to the weld nugget it represents. The shells are positioned at the mid-planes of the sheet metal and the length of the CBAR element. The separation of the shells should therefore be half the sum of the sheet thicknesses. There is no need for any refinement of the mesh around the spot welds. The only requirement for the shell elements used to model the sheets is that they transmit the correct loads to the bar elements. In fact it seems that best results (i.e., most realistic joint stiffnesses) are achieved when the dimensions of the shell elements are quite large, more than twice the diameter of the weld nuggets, or around 10 mm. Due to the single point connection, progressive refinement of the mesh will result in joints that are more and more compliant, which is counterproductive.

Three sheet spot welds may be modeled as if they were two spot welds, with the CBAR elements sharing a common node with the middle sheet.

It is a good idea to group the CBAR elements representing the spot welds into different property sets according to their diameters. This isn't essential if you want to use the option that allows spot weld diameters to be defined automatically based on sheet thickness, but is otherwise sensible.

Equivalent formulations to the CBAR element in other FE codes can also be used; for example, ABAQUS B31 elements. Note that in addition to the forces transmitted through each spot weld, it is also necessary that the sheet thicknesses be available in the results file. It may be necessary to specifically request in the FE input deck that these form part of the results set. Also note that when using LS-DYNA with beam elements, the connectivity between the weld elements and the shell elements is not included. For further information on how the connecting shells may be found see *DesignLife Programmers Manual* > *Spot Weld Spatial Searching*.

Details of the supported FE analysis codes and joint formulations for spot welds are provided elsewhere, but some comment is required here concerning so-called "Area Contact Methods".

The simple CBAR approach to modeling spot welds has a couple of drawbacks:

- It tends to produce structural models with global stiffnesses that are a little too low. This leads to errors in vibrational modes, etc.
- The requirement that the CBAR elements be perpendicular to the sheets makes mesh generation more difficult, because of the need for more or less congruent meshes on opposing flanges.

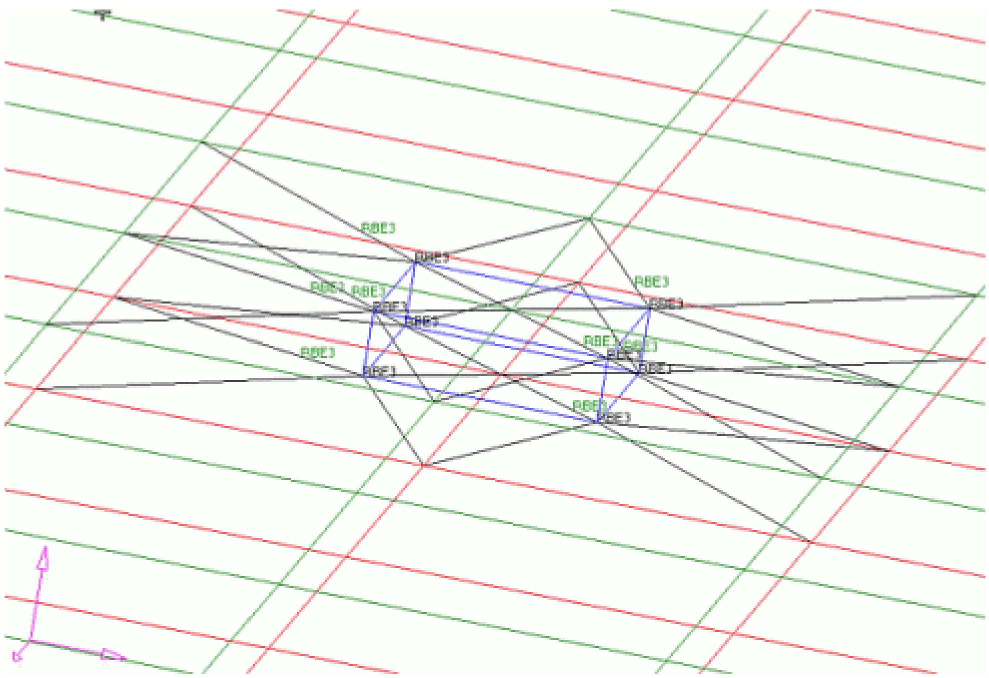
A number of so-called "Area Contact Methods" have been tried in order to address one or both of these issues, including:

- Adding a “spider” of rigid elements to distribute the load to the mesh
- Using NASTRAN CWELD elements
- Using a single hexahedral element to represent the weld nugget, connected to the sheets by means of multi-point constraints or equivalent elements.

Note	When defining a CWELD element in NASTRAN, set the PWELD property MSET to OFF and use SPOT for TYPE. If MSET is ON or if TYPE is blank, then DesignLife won’t consider the CWELD element to be a spot weld and the analysis will not continue.

The single hexahedral element option is the most popular and this will be discussed in more detail. A joint of this type (an “ACM”) can be realized for example in NASTRAN using CHEXA and RBE3 elements, or in ABAQUS using C3D8 and DCOUP3D elements. An example is illustrated in Figure 5-104 below:

Fig. 5-104 ACM representation of spot weld



Note that the hexahedral (e.g., CHEXA) element must be created in such a way that if the node numbers are G1, G2 ...G8 in the input deck, G1-G4 are associated with sheet 1, and G5-G8 with sheet 2. When a three-sheet spot weld is to be defined, the two hexahedral elements should share four nodes at the middle sheet. To correctly identify the middle sheets the element on multi-sheet welds should be in the same orientation with the bottom face of the first element being common with the top face on the second element and continuing this throughout the each individual weld. This means that nodes G5-G8 on the first element are common with G1-G4 on the second.

Either grid point forces from the nodes of the hexahedral element, or MPC forces (which will be equivalent) can be used. Either way, the forces are transformed to give forces and moments on an equivalent bar element joining the centroids of the faces G1-G4 and G5-G8, in the coordinate system of the equivalent bar. The coordinate system of the bar element is defined so that the z-axis is defined by the centroids of the faces G1-G4 and G5-G8, and the x-axis is as close to parallel to G1-G2 as possible.

Note that if MPC forces are used with three-sheet spot welds, the life prediction for the weld nugget will not be correct. It is in general better to use Grid Point Forces calculated by the FE solver for the hexahedral elements.

Fig. 5-105 Hexahedral element and equivalent bar element

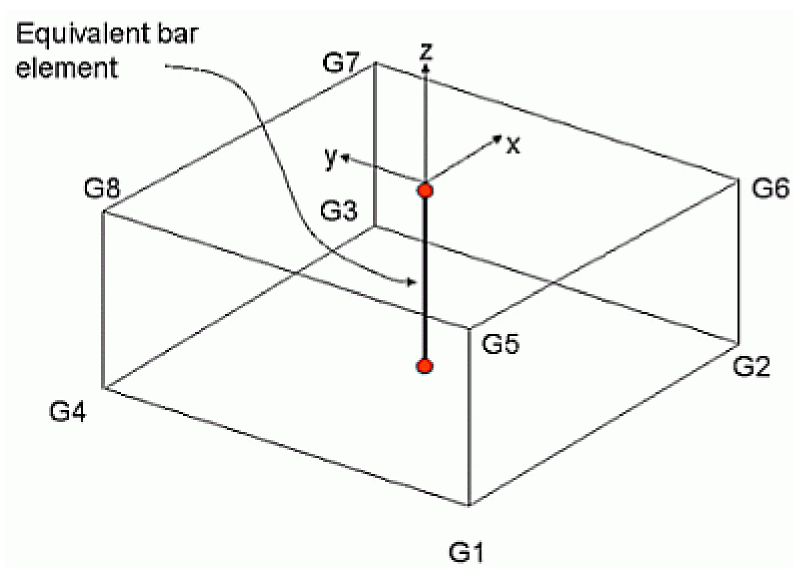
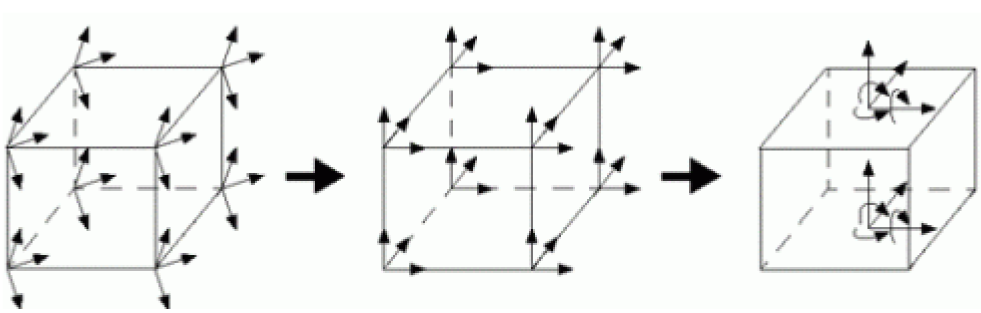


Fig. 5-106 Forces are transformed from global to local coordinates and then transferred to the centroids of the opposing element faces.



The advantages of this method are:

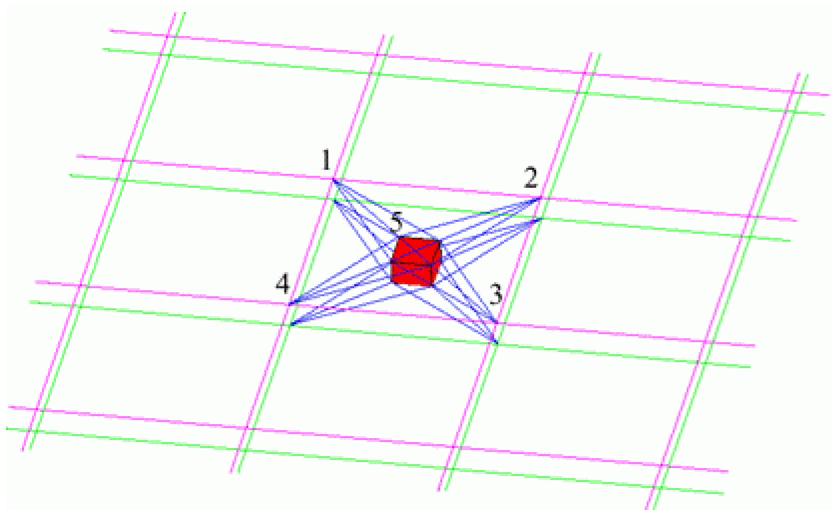
- Joints of this type can be readily created automatically by commercial meshing tools.
- Joints can be created without the need to have congruent meshes on the flanges. This means that sheet metal parts can be meshed independently and SpotWeld can be easily moved.
- The global stiffness of models created in this way is more realistic, and can be "tuned" by adjusting the cross-sectional area of the spot weld elements. (The ratio of the element cross-sectional area to the actual spot weld cross-sectional area is known as the area contact factor.)
- Such methods meet the requirements of other disciplines (e.g., NVH) and therefore promote use of common models.

The disadvantage of this method is that although it may be easier to implement than using CBAR elements, and the global stiffness may be better, there is much more variability in local stiffness, which can give rise to quite large inconsistencies in fatigue:

- Application of the joints may be "mesh independent" but the results are very much mesh dependent. The position of the joint relative to the attached shell elements can have quite significant effects on the forces and moments and hence on the fatigue life.
- Results may be very poor if the MPC or RBE3 connection connects to nodes that are in the angle of a flange.
- Results may be very poor if the MPC or RBE3 connections share independent nodes with adjacent spot welds.

ACM methods can still be used successfully if these factors are taken into account. For example, at least three elements should be used across flanges, and best results are achieved when the connection to each sheet lies within a single shell element. Note also that due to the different characteristics of the different modelling strategies, different S-N curves may be required for best results.

Fig. 5-107 ACM connection lying within a single element



Note When using ABAQUS (.ODB) FE results using C3D8 and DCOUP3D elements, the connectivity between the weld elements and the shell elements is not included, so either the ABAQUS input file needs to be used along with the .ODB file, or alternatively Spot Weld Spatial Searching can be used as discussed earlier.

With LS-DYNA the spot weld nugget can also be modelled as a single hexahedral element but in this case tied contact must be used to connect this element to the sheets. Then the contact forces are recovered from the *ncforc* file and used in the same way as the grid point forces described above.

Two types of tied contact can be used:

1. **CONTACT_TIED_SHELL_EDGE_TO_SOLID*
2. **CONTACT_TIED_SURFACE_TO_SURFACE*

In both cases, the option to include master or slave side forces in the **DATABASE_NCFORC* interface force file needs to be set (MPR=1 or SPR=1, depending on whether the weld elements are defined as the master or slave for the interface).

Note When using *CONTACT_TIED_SHELL_EDGE_TO_SOLID, the spot weld hexahedral elements must be defined as the master in the contact definition in order to get the hexahedral node forces in the *ncforc* file (nodal contact forces are available only for the master side with node to surface contact types).

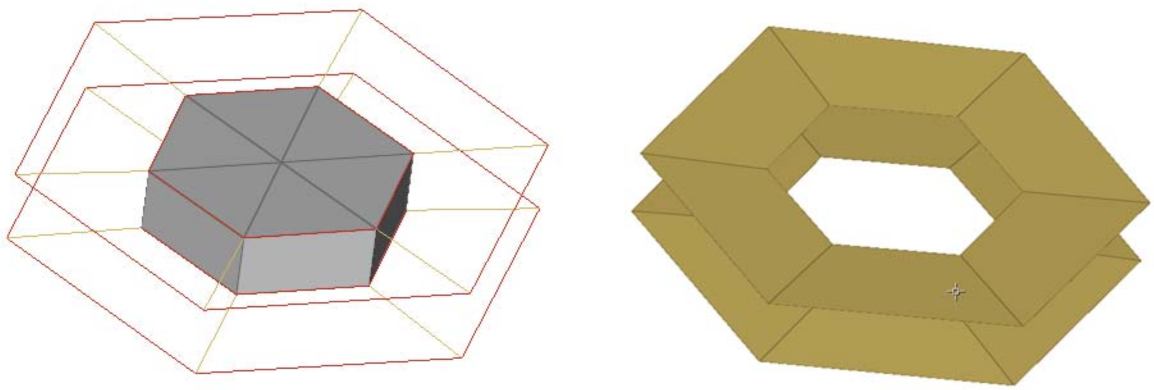
A single interface can be used to connect all the appropriate shell elements (using part ID or part set ID) to the weld elements (part ID) or multiple interfaces can used as long as each weld element only takes part in a single interface. With 3 sheet welds, each pair of welds needs to be in the same interface.

The keyword **DATABASE_NCFORC* is used with *BINARY* field set to 3 to write an ASCII *ncforc* file. Note that the time interval between outputs to the *ncforc* file can be different to any other solution time intervals.

To use the interface forces from LS-Dyna in a spot weld analysis, the *SolutionLocation* is set to *SpotWeld*, and *EntityDataType* set to *ContactForce*. Then if there is an *ncforc* file present in the same folder as the *d3plot* files, contact forces will be picked up and translated for the nodes on the hexahedral weld elements. Note that the *d3plot* files are used in the normal way as input to the analysis and the ncforc is then automatically read.

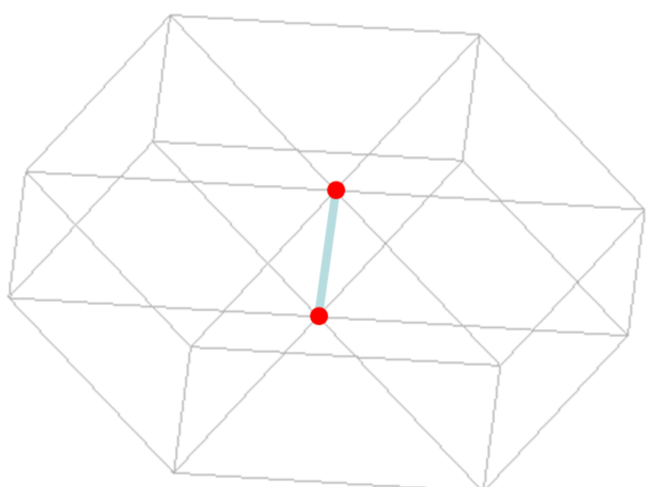
The spot weld nugget can also be modelled as a group of 6 *penta* elements connected directly to the sheets. A joint of this type can be realised in NASTRAN using *CPENTA* elements, in ANSYS using *SOLID 185* or in ABAQUS using *C3d6* elements. The forces at the corners of the nugget would be recovered with GPFORCE (NASTRAN), standard force/moment output (ANSYS) and NFORC (ABAQUS).

Fig. 5-108 PENTA element connection showing surrounding shells (no shells on the penta element faces)



The grid point forces are transformed in a similar way to the ACM method to give forces and moments on an equivalent bar element jointing the central nodes of the two faces of the PENTA elements.

Fig. 5-109 PENTA Elements and equivalent bar element.

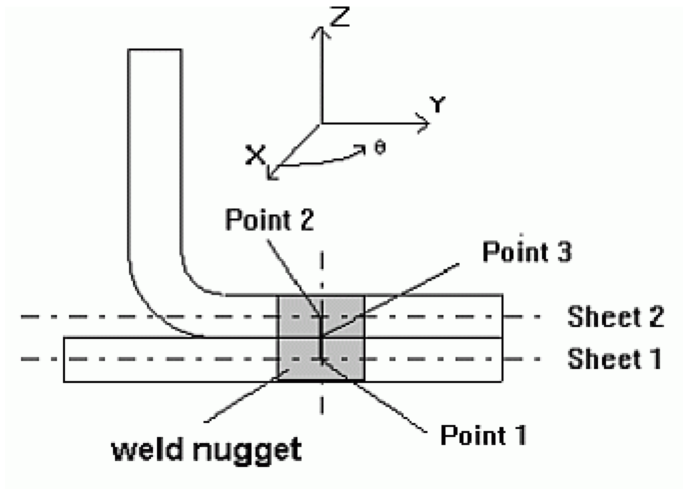


Note, as with the ACM method, due to the different characteristics of the various modelling strategies, different S-N curves may be required for best results.

The steps in the fatigue calculation are now described in more detail.

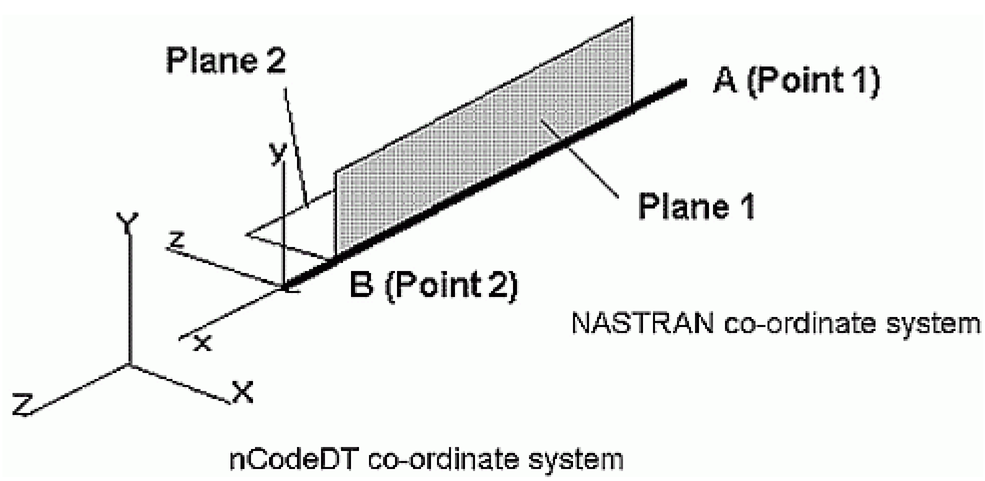
Force and Moment Recovery

Fig. 5-110 Cross-section of typical spot weld



The shaded part in Figure 5-110 represents the spot weld “nugget”. Using NASTRAN as the example finite element solver, the weld is modelled as a stiff CBAR element joining the mid-planes of two sheets of thin shell elements. The length of the bar element is 0.5(s1+s2) where s1 and s2 are the thickness’s of sheets 1 and 2 respectively. Point 3 is on the axis of the weld nugget and at the interface of the two sheets, i.e., 0.5 s1 from Point 1. All forces and moments are taken to be in the coordinate system illustrated in Figure 5-111. This is a Cartesian system with the Z axis going from Point 1 to Point 2. This is different both from the arrangement used by Rupp et al [1] and that used in NASTRAN for CBAR elements, but a little simpler. The relationship between the nCodeDT and NASTRAN CBAR coordinate systems is illustrated in Figure 5-111.

Fig. 5-111 Spot weld coordinate system



The correspondence between the spot weld forces and the CBAR forces from NASTRAN is as follows:

	nCodeDT Spot Weld	**NASTRAN CBAR**
Point 1	FX1	-Z force (plane 2 shear)
	FY1	Y force (plane 1 shear)
	FZ1	X force (axial)
	MX1	-Z moment 1 (plane 1 bending moment, end A)
	MY1	-Y moment 1 (plane 2 bending moment, end A)
	MZ1	X moment (torque)
Point 2	FX2	Z force (plane 2 shear)
	FY2	-Y force (plane 1 shear)
	FZ2	-X force (axial)
	MX2	Z moment 2 (plane 1 bending moment, end B)
	MY2	Y moment 2 (plane 2 bending moment, end B)
	MZ2	-X moment (torque)
Point 3	FX3=FX1	
	FY3=FY1	
	FZ3=FZ1	
	MX3 = (MX1*s2 – MX2*s1)/(s1 +s2)	
	MY3 = (MY1*s2 – MY2*s1)/(s1 +s2)	
	MZ3 = (MZ1*s2 – MZ2*s1)/(s1 +s2)	

Where s1 and s2 are the thickness's of sheets 1 and 2 respectively.

Stress Calculations

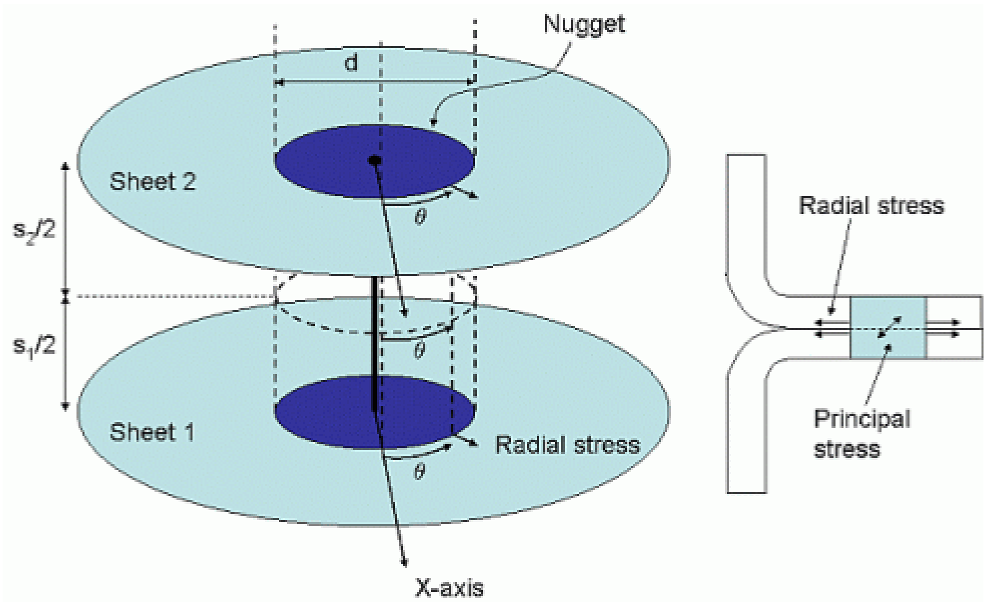
In order to make fatigue calculations, we need stress-time histories, which in turn are derived from time histories of the forces and moments at each of the three calculation points. The time histories of force and moment (FX, FY, FZ, MX, MY, MZ) at the three calculation points can be derived from Time Series, Constant Amplitude or Time Step load providers (see the section on load providers) or from Duty Cycles based on these load provider types.

The required stresses are the structural stresses in the two sheets adjacent to the spot weld and in the weld nugget.

The structural stresses in the sheets are the radial stresses from the inner surfaces of the two sheets around the perimeter of the weld nugget.

The structural stresses in the nugget are determined from the principal stresses around the perimeter of the nugget at a section representing where the two sheets meet, based on the nominal shear and direct stresses in the nugget at that point.

Fig. 5-112 Spot weld force calculation—schematic



The structural stresses in the sheets are a function of the shear (FX, FY), axial (FZ), and bending moments (MX, MY) transmitted by the structural element (e.g., CBAR) to the sheet, and varying with the angle θ. For example, for sheet 1:

$$\sigma_{sheet1} = -\sigma_{\max}(FX1)\cos\theta - \sigma_{\max}(FY1)\sin\theta + \sigma(FZ1) + \sigma_{\max}(MX1)\sin\theta - \sigma_{\max}(MY1)\cos\theta$$

...where:

$$\sigma_{\max}(FX1) = \frac{FX1}{\pi d s_1} \times SFFXY \times d^{DEFXY} \times s_1^{TEFXY}$$

$$\sigma_{\max}(FY1) = \frac{FY1}{\pi d s_1} \times SFFXY \times d^{DEFXY} \times s_1^{TEFXY}$$

(Contributions from the shear forces. Note that these are adjusted by an empirical factor, $SFFXY \times d^{DEFXY} \times s_1^{TEFXY}$ which adjusts the contributions of the shear forces relative to those of the axial force and bending moments, and allows a size effect in terms of weld diameter and sheet thickness to be taken into account.)

$$\sigma(FZ1) = \frac{1.744 \cdot FZ1}{s_1^2} \times SFFZ \times d^{DEFZ} \times s_1^{TEFZ}$$ if FZ1 > 0 or 0 if FZ1 ≤ 0

(Contributions from the axial force; note that only the contribution from tensile loads is considered. Note also the empirical factor in the same form as before.)

$$\sigma_{\max}(MX1) = \frac{1.872 \cdot MX1}{ds_1^{\,2}} \times SFMXY \times d^{DEMXY} \times s_1^{\,TEMXY}$$

$$\sigma_{\max}(MY1) = \frac{1.872 \cdot MY1}{ds_1^{\,2}} \times SFMXY \times d^{DEMXY} \times s_1^{\,TEMXY}$$

(Contribution from the bending moment, again including an empirical factor.)

A similar set of equations is used for the stresses in sheet 2.

The empirical factors are calculated from a set of nine material properties. The following values are typical for spot welds in steel:

Component	**Factor**	**Diameter Exponent**	**Thickness Exponent**
FX, FY	SFFXY = 1.0	DEFXY = 0.0	TEFXY = 0.0
MX, MY	SFMXY = 0.6	DEMXY = 0.0	TEMXY = 0.5
FZ	SFFZ = 0.6	DEFZ = 0.0	TEFZ = 0.5

The following values are more typical for spot welds in aluminium:

Component	**Factor**	**Diameter Exponent**	**Thickness Exponent**
FX, FY	SFFXY = 0.4	DEFXY = 0.5	TEFXY = -0.25
MX, MY	SFMXY = 0.4	DEMXY = 0.5	TEMXY = -0.25
FZ	SFFZ = 1.0	DEFZ = 0.0	TEFZ = 1.0

The stress around the periphery of the nugget at Point 3 is calculated as follows:

Shear stress:

$$\tau = \tau_{\max}(FX3)\sin^2\theta + \tau_{\max}(FY3)\cos^2\theta$$

Direct stress:

$$\sigma = \sigma(FZ3) + \sigma_{\max}(MX3)\sin\theta - \sigma_{\max}(MY3)\cos\theta$$

...where:

$$\tau_{\max}(FX3) = \frac{16 \cdot FX3}{3\pi d^2}$$

$$\tau_{\max}(FY3) = \frac{16 \cdot FY3}{3\pi d^2}$$

$$\sigma(FZ3) = \frac{4 \cdot FZ3}{\pi d^2}$$ if FZ3 > 0 or 0 if FZ3 ≤ 0

$$\sigma_{\max}(MX3) = \frac{32 \cdot MX3}{\pi d^3}$$

$$\sigma_{\max}(MY3) = \frac{32 \cdot MY3}{\pi d^3}$$

Then, from the shear and direct components of stress, we can calculate the Absolute Maximum Principal stress in the nugget as follows:

$$\sigma_{AMP\,nugget} = \frac{\sigma}{2} + \sqrt{\left(\frac{\sigma}{2}\right)^2 + \tau^2} \text{ if } \sigma \geq 0 \text{ or...}$$
$$\sigma_{AMP\,nugget} = \frac{\sigma}{2} - \sqrt{\left(\frac{\sigma}{2}\right)^2 + \tau^2} \text{ if } \sigma < 0$$

Although most spot welds in a well-designed structure will not experience significant torsional loads under normal conditions, occasionally, spot welds will fail early due to torsional fatigue. Torsional fatigue is modelled separately as a distinct failure mode, determined by the circumferential shear stress in each sheet. This is calculated as follows:

$$\tau_1 = \frac{2 \cdot MZ1}{\pi S_1 d^2}$$

$$\tau_2 = \frac{2 \cdot MZ1}{\pi S_2 d^2}$$

Note that this torsional calculation is optional (based on the CalculateTorsion property) and requires additional material properties.

When using a vibration load provider complex forms of the above equations are used to calculate the structural stress PSD.

Material Data

The material data for spot welds consists of S-N curves, based on the Standard nCode formulation, with some additional parameters which are specifically for spot welds. Curves can be for failure in the sheet metal, through the nugget, or for torsional failure—the data format is the same:

Parameter Name	**Description**
MaterialType	nCode material type number
YS	Material Yield Stress
UTS	Ultimate Tensile Strength
E	Elastic Modulus
me	Elastic Poisson Ratio
mp	Plastic Poisson Ratio
SRI1	Stress range intercept
b1	First fatigue strength exponent
Nc1	Transition life
b2	Second fatigue strength exponent
SE	Standard error of $\log_{10}(N)$. This is used to calculate the life adjusted to a certain probability of failure/survival.
RR	R-ratio (ratio of minimum to maximum load) of the tests used to define the S-N curve
Nfc	Numerical fatigue cutoff life. Beyond this life, damage will be assumed to be zero.
MSS	Mean stress sensitivity factor for “simple” method
SF-FXY	Scale factor for stress due to FX or FY (shear forces)
DE-FXY	Diameter exponent for stress due to FX or FY (shear forces)
TE-FXY	Thickness exponent for stress due to FX or FY (shear forces)
SF-MXY	Scale factor for stress due to MX or MY (bending moments)
DE-MXY	Diameter exponent for stress due to MX or MY (bending moments)
TE-MXY	Thickness exponent for stress due to MX or MY (bending moments)
SF-FZ	Scale factor for stress due to FZ (axial force)
DE-FZ	Diameter exponent for stress due to FZ (axial force)
TE-FZ	Thickness exponent for stress due to FZ (axial force)
SF-MZ	Scale factor for stress due to MZ (torsion) – not used

Parameter Name	Description
DE-MZ	Diameter exponent for stress due to MZ (torsion) – not used
TE-MZ	Thickness exponent for stress due to MZ (torsion) – not used
M1	Mean stress sensitivity when R > 1 for FKM mean stress correction method
M2	Mean stress sensitivity when -∞ < R < 0 for FKM mean stress correction method
M3	Mean stress sensitivity when 0 ≤ R < 0.5 for FKM mean stress correction method
M4	Mean stress sensitivity when 0.5 ≤ R < 1 for FKM mean stress correction method
Comments	
References	

As mentioned, the S-N data is Standard nCode S-N data, and is handled in the same way as for a normal S-N calculation. The parameters SF-FXY...TE-MZ are used in the calculation of stress, as described in the previous section. M1-M4 are the FKM mean stress sensitivity factors. The FKM mean stress correction is described in the section on the Standard SN Analysis Engine.

The parameter MSS is used for a simple mean stress correction. It is basically a simplified version of the FKM mean stress correction where:

M1 = M2 = M3 = M4 = -MSS

Material Data - Coefficient values used in MDB databases

If an MXD database is used to supply the materials data then the coefficients used to convert forces and moments into stresses are also defined with the chosen material's data.

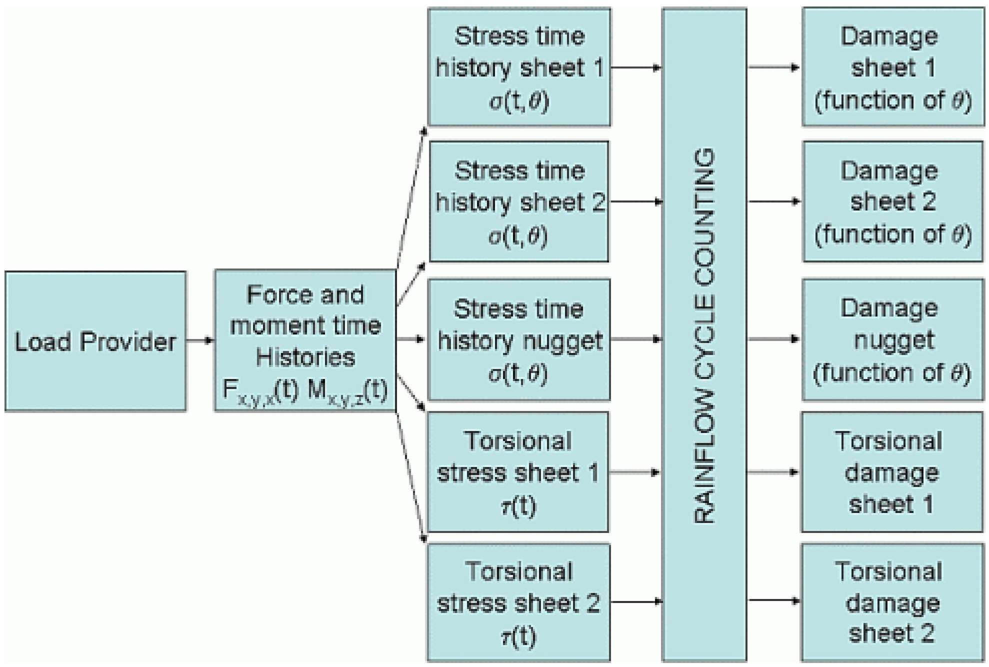
If the older MDB format database is used (as may be used by FE-Fatigue) then the required coefficients are not included in the materials data. For MDB databases the following values are used:

Parameter Name	Description	Value
For Spot Weld material = Steel		
Scale Factor	SF-FXY	1.0
Diameter Exponent	DE-FXY	0.0
Thickness Exponent	TE-FXY	0.0
Scale Factor	SF-MXY	0.6
Diameter Exponent	DE-MXY	0.0
Thickness Exponent	TE-MXY	0.5
Scale Factor	SF-FZ	0.6
Diameter Exponent	DE-FZ	0.0
Thickness Exponent	TE-FZ	0.5
Scale Factor	SF-MZ	0.0
Diameter Exponent	DE-MZ	0.0
Thickness Exponent	DE-MZ	0.0
For Spot Weld material = Aluminium		
Scale Factor	SF-FXY	0.4
Diameter Exponent	DE-FXY	0.5
Thickness Exponent	TE-FXY	-0.25
Scale Factor	SF-MXY	0.4
Diameter Exponent	DE-MXY	0.5
Thickness Exponent	TE-MXY	-0.25
Scale Factor	SF-FZ	1.0
Diameter Exponent	DE-FZ	0.0
Thickness Exponent	TE-FZ	1.0
Scale Factor	SF-MZ	0.0
Diameter Exponent	DE-MZ	0.0
Thickness Exponent	TE-MZ	0.0

Analysis Process Summary

The spot weld analysis process is summarized in Figure 5-113 below:

Fig. 5-113 Spot weld analysis process summary



Based on the information provided by the load provider, the analysis engine creates time histories of force and moment (FX, FY, FZ, MX, MY, MZ) at the three calculation points (Sheet 1, Sheet 2, Nugget). These can be derived from Time Series, Constant Amplitude, Time Step or Vibration load providers, or from Duty Cycles based on these load provider types.

From the force and moment time histories, stresses are calculated as described above, before rainflow counting and damage accumulation using a standard S-N approach.

The properties of the spot weld analysis engine are discussed below.

5.7.2 CombinationMethod and SNMethod

These properties (and their associated properties CustomCombinationMethod and CustomSN method) allow the Spot Weld SN engine to be customized, for example to allow life prediction of self-piercing rivets. Otherwise they should be left set to “Standard”.

5.7.3 MeanStressCorrection

There are three mean stress correction options for spot weld fatigue analysis. Note that the application of these is slightly different for torsional fatigue analysis.

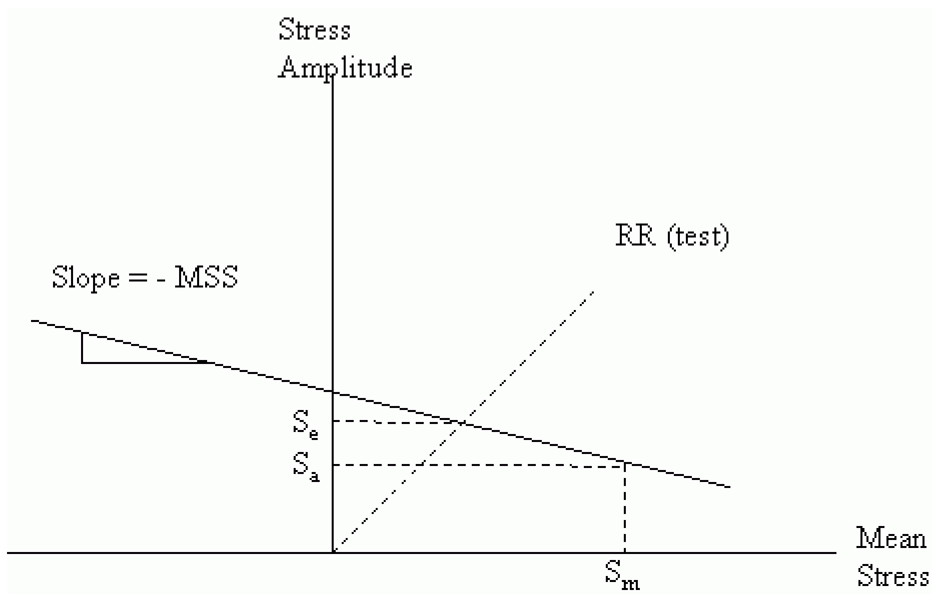
None

No mean stress correction is carried out.

Simple

This method uses a simplification of the FKM mean stress correction, defined by a single mean stress sensitivity factor MSS. It allows an equivalent stress amplitude S_e at the material R-ratio (RR) to be calculated based on the stress amplitude S_a and mean S_m of each cycle.

Fig. 5-114 Simple mean stress correction method



FKM

The FKM mean stress correction is described in detail in the section on the Standard SN Analysis Engine.

5.7.4 NumberOfAngles

In spot weld calculations, predictions of life are made at intervals around the periphery of the weld in the two sheets and in the weld nugget. This property controls the number of values of the angle θ at which the fatigue calculations are made. For example, if NumberOfAngles = 18, calculations will be made at 20 degree intervals.

5.7.5 ProcessMiddleSheets

True/False: The Rupp method for spot weld life prediction has not been validated for the prediction of fatigue damage occurring at the middle sheets for spot welds joining more than two sheets. It should also be noted that such failures are relatively rare, difficult to reproduce in the laboratory, and difficult to detect in practice. For this reason, the software provides an option (by setting this property to False) to omit any fatigue calculation on the middle sheets, which might otherwise give a false positive failure prediction.

5.7.6 CertaintyOfSurvival

The certainty of survival allows statistical variations in material behavior to be taken into account. It works in the same way as for Standard S-N calculations (see the section on the Standard SN Analysis Engine for details). The default value (50%) will result in the mean life being predicted.

5.7.7 ScaleFactor

This scale factor is applied to the stresses before the fatigue calculation. The default value is 1.0.

5.7.8 CalculateTorsion

Setting this option to True enables fatigue calculations to be made based on the shear stress around the periphery of the spot weld in each sheet, due to the torsional load on the spot weld.

Please note the following:

1. This option requires that suitable S-N curves be defined for the torsion case on the Material Map form.
2. Do not switch this option on unless your modeling strategy can generate realistic torsional loads in the spot welds. For example, this is not true of the simple CBAR modeling approach.
3. The mean stress correction is applied a little differently, because the sign of a mean shear stress cannot influence the fatigue damage. For this reason, the ABSOLUTE value (i.e., the magnitude) of the mean stress is used when applying any mean stress correction for the torsion case.
4. The calculation is a completely separate operation—there is no combination of the torsional stresses with the other stresses on the spot weld.
5. This option has had little validation, and no suitable S-N curves are provided with the software. However, it has been successfully used in practice, using S-N curves deduced from practical experience with car body structures, and has proved a useful tool for predicting premature failure in the small number of spot welds that may experience significant torsional loads.
6. The results of the torsional fatigue calculation are identified in the results by being associated with an angle of -1.

5.7.9 Output...

The user can choose to send any of these to the results:

Table 5-11 Possible spot weld result outputs

OutputEventResults	Whether to output *per event results* or not, when in duty cycle mode.
OutputMaxMin	Whether to output *max and min cycle stresses*.
OutputSheetCount	Whether to output the *sheet count* to the results
OutputSpotWeldDiameter	Whether to output *Spot Weld diameter* to the results
OutputSpotWeldNumber	Whether to output *Spot Weld number to* the results

5.7.10 BackCalcMode

The options here are **None** or **ScaleFactor**. When **ScaleFactor** is selected, the Spot Weld Analysis engine iteratively searches for a Scale Factor, which, when applied to the stresses before the fatigue calculation, will produce the **Target-Damage** within a reasonable tolerance set by **BackCalcAccuracy**. This setting overrides the ScaleFactor property setting on the analysis engine (see above). It provides a convenient way of determining a safety factor for complex or variable amplitude loadings. When this option is selected, the damage reported in the results will approximate to the target damage, and the resulting scale factor will be reported, too.

None	Static failure checks are not carried out during spot weld back calculations.

5.7.11 Target Damage

Sets the target damage value to be used for back calculations. Note that it is not always possible to make a successful back calculation. For example, if your loading is constant amplitude and the target damage corresponds to a flat part of the S-N curve, the back calculation will fail.

5.7.12 BackCalcAccuracy

This defines a percentage error on the target damage. The back calculation will iterate until the calculated damage matches the target damage within this tolerance.

5.7.13 BackCalcMaxScale

This defines the maximum value of the scale factor that will be reported in the analysis results.

EventProcessing

This property controls the processing of duty cycles. There are three options:

- Independent
- Combined Full
- Combined Fast

These operate in the same way as for a standard S-N calculation. See "EventProcessing" on page 144 for details.

5.7.14 CheckStaticFailure

Static failure checking compares the maximum value of the CombinedStress, as cycle counted, with the material UTS. It also compares the Equivalent stress amplitude (after any mean stress correction) with the UTS. There is currently no criterion applied for compressive failure.

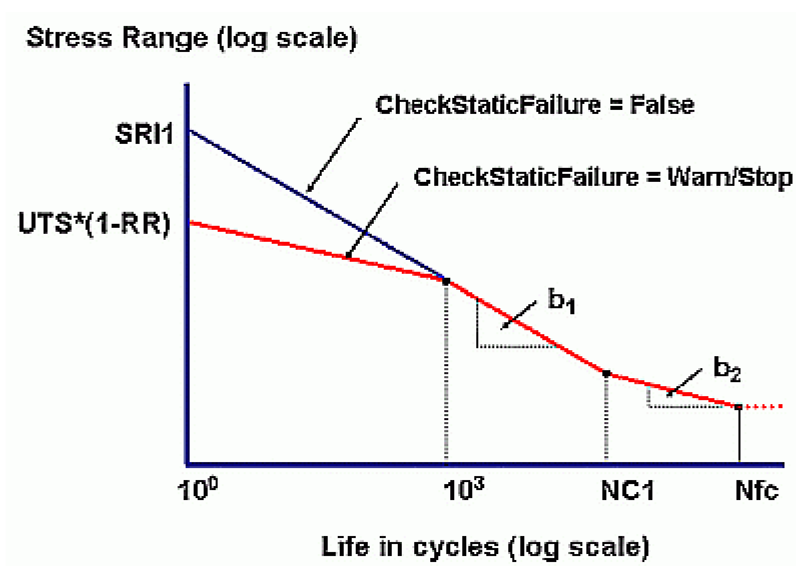
There are three options:

- **Warn**—The calculation continues, but a warning message is issued. The damage for this entity is also assigned a numeric value equal to the value of the StaticFailure Damage property, typically 2.0.
- **Stop**—The analysis job stops with a message.
- **False**—(Recommended) No static failure check is made. Damage will be calculated up to the numerical limit defined by the MaxDamage property. Note that this may be using an extrapolation of the S-N curve and may predict damage values per cycle greater than 1.0. Setting CheckStaticFailure to False has an additional effect if the SNMethod property is set to Standard. In this case, the adjustment of the S-N curve below 1000 cycles to take into account the static strength of the material is disabled.

Please note the following:

The static failure check has not had any validation as regards spot welds, and any prediction of static failure should be treated with caution. Also, setting CheckStaticFailure to Warn or Stop will result in the S-N curve being adjusted in the low cycle region (< 1000 cycles) in the same way as for the Standard S-N method, as illustrated in Figure 5-115. This is not desirable for spot weld fatigue. It can be avoided, either by setting CheckStaticFailure = False (recommended), or setting a value of the UTS such that UTS*(1 – RR) = SRI1 or greater.

Fig. 5-115 Effect of CheckStaticFailure setting on low cycle portion of S-N curve



5.7.15 DamageFloor

This parameter sets a lower limit on calculated damage. If a cycle has less predicted damage, the damage value is set to zero. This parameter is somewhat redundant as similar functionality may be accessed using the fatigue cutoff Nfc in the material definition.

5.7.16 MaxDamage

This parameter sets a numerical upper limit on the damage that can be predicted for each cycle.

5.7.17 StaticFailureDamage

This parameter sets a numerical value that may be set for the damage to a particular node or element in the event of the stress exceeding the UTS. (See "CheckStaticFailure" on page 147 above.)

5.8 Seam Weld Analysis Engine

Fig. 5-116 Seam Weld analysis engine

Object Name: SeamWeldEngine (Seam Weld analysis engine)

Name	Value	Description
General		
SNMethod	Standard	The method used to calculate damage from a stress cycle
CombinationMethod	AbsMaxPrincipal	The method used to combine component stresses/strains
MeanStressCorrection	FKM	The method used to correct the damage calculation for mean stress
MultiAxialAssessment	Standard	Whether to perform assessment of the multi-axial stress state
CertaintyOfSurvival	50	Required confidence level on damage results
ScaleFactor	1	The scale factor to apply prior to damage calculation
ThicknessCorrection	True	Whether to perform thickness corrections
OutputMaxMin	True	Whether to output max and min stresses
OutputMaterialNames	False	Whether to output material names to the results
OutputDistributedSource	False	Whether to output details of the distributed process that generated each result
OutputVibrationStats	False	Whether to output Vibration PSD parameters such as ExpectedZeroUpcrossings, ExpectedPeakCount and spectral moments
BackCalculation		
BackCalcMode	None	Whether to perform a back-calculation or not
TargetDamage	1	Target damage for back calculation
BackCalcAccuracy	1	The accuracy of the back calculation
BackCalcMaxScale	5	The max scale factor to allow in back calculation
BackCalcMinScale	0.2	The min scale factor to allow in back calculation
DutyCycle		
EventProcessing	Independent	How to process separate events in duty cycles
OutputEventResults	False	Whether to output results per event or not for duty cycle processing
Advanced		
CheckStaticFailure	False	The action to take on static failure
DamageFloor	1E-30	The calculated damage value below which the damage is set to zero
MaxDamage	1E30	The maximum damage value
StaticFailureDamage	2	The damage value to insert on static failure

5.8.1 Introduction

The seam weld fatigue analysis method implemented in this engine is based on that originally proposed by Volvo Car Corporation and Chalmers University of Technology, and developed in cooperation with nCode. The method was developed particularly for automotive components welded from thin steel sheets (typically 1-3 mm), and most of the industrial applications of this method have been to this type of structure. However, the structural stress-based approach it uses is of quite general applicability and an approach based on the ASME Boiler & Pressure Vessel Code VIII, Division 2 for thicker parts using solid elements has also been included.

The method is basically quite similar to the standard S-N method, but with some special features to deal with welds.

Please note the following:

1. The method was originally developed for thin sheet welded structures, modeled and analyzed in NASTRAN using predominantly CQUAD4 elements. (The original work is described in "Fatigue Life Prediction of MAG-Welded Thin-Sheet Structures", by Mikael Fermér, Magnus Andréasson and Björn Frodin. SAE Technical Paper 982311.)
2. As originally described by Fermér et al, the method used grid point forces from the FE solution to calculate structural stresses at the weld toe. This approach was considered to provide a good level of mesh insensitivity compared to standard stress recovery methods, which rely on extrapolation from gauss points. Subsequently, it was realized that a good level of mesh-insensitivity could also be achieved by using the CUBIC stress recovery option in NASTRAN, and implementation of the method in software relied on CUBIC stress recovery. (The CUBIC method

derives stresses at each node in each element from the displacements and rotations of the nodes.) Although use of stresses from other FE codes is quite possible, significant differences in the results may be caused by the differences in stress recovery methods. If it is necessary to use stresses from other FE codes (e.g., ABAQUS), better results may be obtained through the use of quadratic elements.

3. From version 6.0, optionally grid point forces may be used to determine the stresses to be used for fatigue analysis. The details of the method have been refined from that described in SAE 982311 and are described in detail later. To use grid point forces, they must be present in the FE results file, and EntityDataType = ForceMoment must be set on the Analysis Group properties.

4. From version 6.0, optionally nodal displacements and rotations may be used to determine the stresses in the elements around the weld. To use this option, nodal forces and displacements must be available in the FE results file, and EntityDataType = Displacement must be set on the Analysis Group properties. Details of this method are described later.

5. There are some specific guidelines for meshing the welds, which will be discussed shortly, but essentially the structure must be meshed predominantly with CQUAD4 elements (or equivalent) representing the mid-planes of the metal sheets, and the weld bead represented by a single or double row of shells. The mesh around the weld should be regular, with elements of around 5 mm in size if possible (in practise the size of the elements tends to be dictated by the dimensions of the weld and the need for good-shaped elements). Triangular elements in the vicinity of the weld should be avoided where possible.

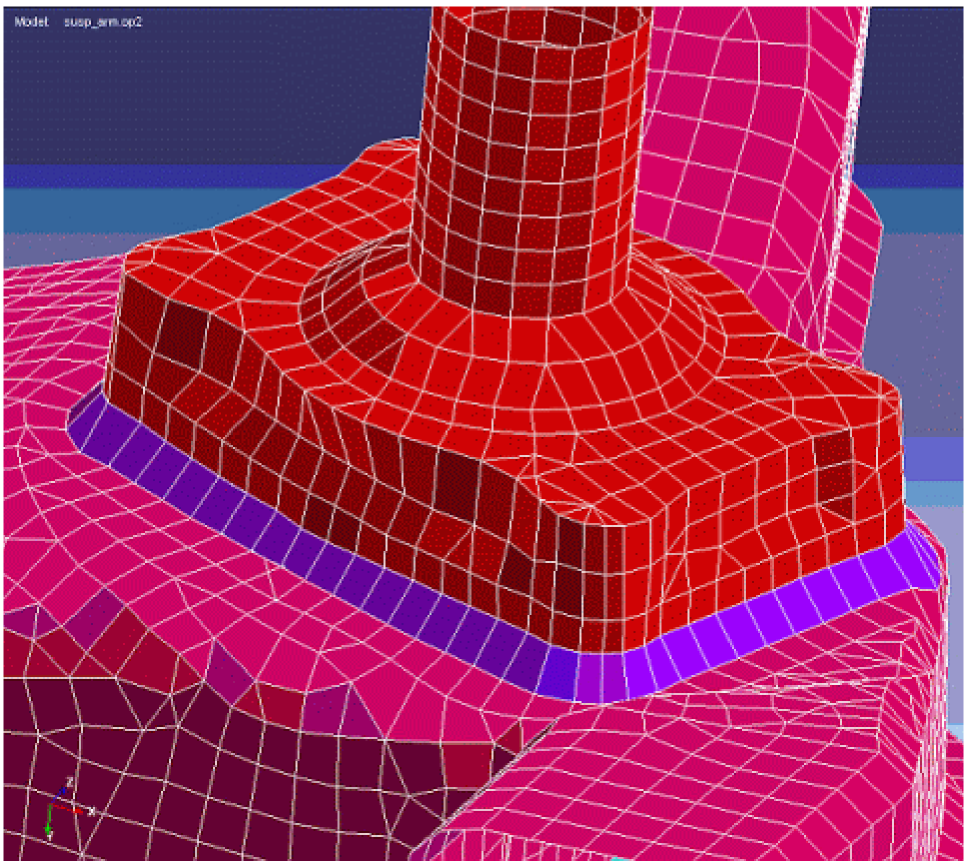
6. If Stresses (preferably CUBIC) or Displacements are being used, unaveraged node-at-element stresses are recovered for the weld toes and weld roots, based on the elements adjacent to the weld, and used to make the fatigue calculations. Where weld throat calculations are made, the centroid stress from the weld throat element is used.

7. If grid point forces are being used, average membrane and bending stresses at the mid-point of the element edges at the weld toes and weld roots are derived from the grid point forces, and this allows the stress normal to the weld toe at the top and bottom surfaces of the weld toe and root elements to be calculated. These stresses are then used to make the fatigue calculations in the normal way. Where weld throat calculations are made, the stresses are calculated from the two “welded” edges of the weld element and averaged to the centroid. The grid point force approach requires linear elements to be used.

8. From version 7.0, the option to calculate stresses at the mid-point of the element edges at the weld toes and roots is extended to FE stresses and stresses derived from displacements. These methods either recover or derive unaveraged (node-on-element) stresses. If the mid-element edge stress is used (by setting WeldLocation = MidElementEdge on the analysis group properties) the stresses for each element edge will be averaged in a common coordinate system and then translated to a local coordinate system based on the element edge.

9. The method determines the contribution of bending to the total stress, and from this determines whether the weld is essentially “stiff” or “flexible”. A different SN curve may be required for these cases, and interpolation is made between the curves based on the degree of bending.
10. The Vibration Load Provider is supported for stress inputs from shell and solid elements. In this case complex forms of the structural stress equations are used to calculate the stress PSD.

Fig. 5-117 FE model of typical automotive welded component



The following sections describe the essential features of the method in more detail.

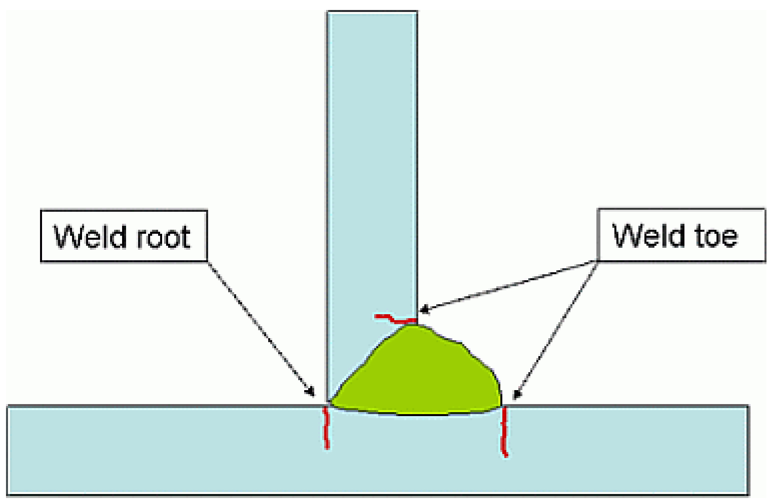
5.8.2 Grouping and Weld Types

Different types of welds have to be analyzed differently, depending upon their geometry, the process by which they are created, and the failure modes to be considered. In nCodeDT, welds must therefore be collected into different groups for analysis according to which category they fall into. In practice, this means that welds have to be grouped into separate FE Import Analysis Groups, each of which has the property SeamWeldType set to an appropriate value. The supported weld types (SeamWeldType) are as follows.

Fillet

A fillet weld is a Metal Inert Gas (MIG) or Metal Active Gas (MAG) weld joining two sheets at an angle. The weld illustrated is a single-sided weld. The most likely failure (fatigue crack) locations at the weld toe or weld root are marked. These are the locations where the fatigue damage will be evaluated. Note that weld throat failure is considered unlikely unless a weld is of poor quality or the weld bead under-dimensioned. The analysis of the weld throat as a failure location is a beta feature and can be activated by the preference /BetaFeatures/DesignLife/SeamWeldThroatFailure.

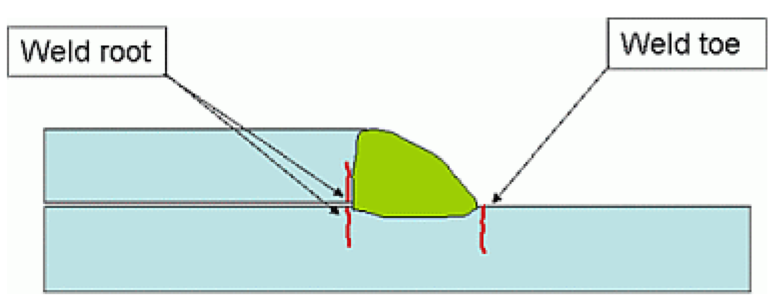
Fig. 5-118 Fillet weld cross-section showing likely failure locations



Overlap

An overlap weld is similar to a fillet weld, except that the plates being joined are parallel. Note the marked possible failure locations – these are the locations where the analysis engine will predict the fatigue damage.

Fig. 5-119 Overlap weld cross-section showing likely failure locations

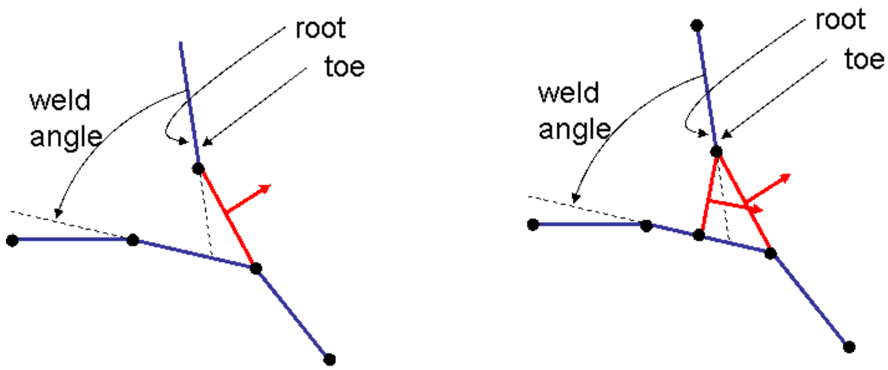


CombinedFilletOverlap

The difference between fillet and overlap welds is the angle between the vertical plate 'a1' (Fig 5-120) and the base plate 'b'. As this angle reduces from 90° the weld transitions from fillet to overlap. Two properties control the calculation of the weld type when the *SeamWeldType = CombinedFilletOverlap*, these are *LowerFilletOverlapThreshold* and *HigherFilletOverlapThreshold*.

The angle of the first element representing sheet *a* relative to sheet *b* is first determined:

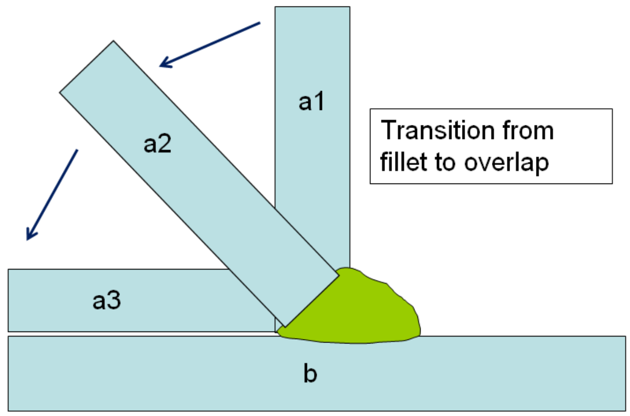
Fig. 5-120 Angle calculation method.



- If this angle is less than or equal to the *LowerFilletOverlapThreshold* then this section of the weld is defined as a an overlap weld.
- If this angle is greater than or equal to the *HigherFilletOverlapThreshold* then this section of the weld is defined as a fillet weld.
- If the angle is > *LowerFilletOverlapThreshold* and < *HigherFilletOverlapThreshold* then the weld is defined as a combined *FilletOverlap,* and both fillet and overlap calculations are carried out.

- If both angle properties are left blank then no assessment is made and all welds are defined as combined fillet and overlap.

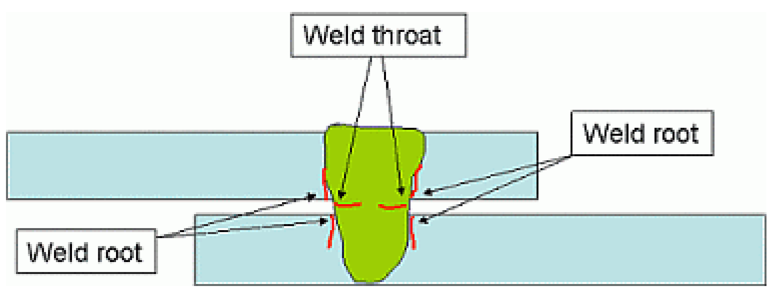
Fig. 5-121 Transition of weld from fillet to overlap.



Laser Overlap

In the case of the Laser Overlap weld, failures initiating and propagating through the sheets are classed as weld root failures. The other possible failure mode is cracking through the weld itself. This is classified as a weld throat failure. The possible failure locations are illustrated in Figure 5-123. Note that the sheet separation has been exaggerated for the sake of clarity.

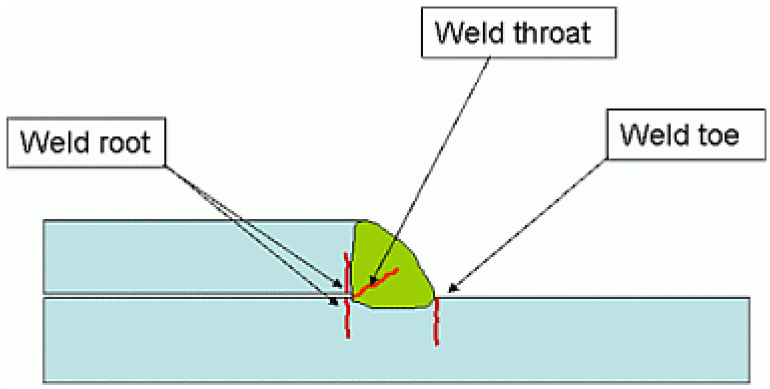
Fig. 5-122 Laser overlap cross-section showing likely failure modes (sheet separation exaggerated)



Laser Edge Overlap

Laser edge overlap welds are handled in a very similar way to overlap welds, except that because of the small dimensions of the weld bead, failure through the weld throat becomes a possibility.

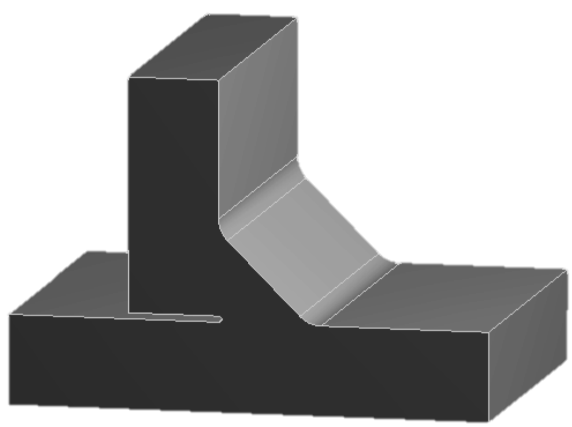
Fig. 5-123 Laser edge overlap weld cross-section and possible failure locations



Solid Weld

The *SolidWeld* type is provided for cases where it is not appropriate to use thin-shell elements because of the relative thickness of the structure and solid elements have instead been used. Below is shown a weld modeled with solid elements, however it should be noted that the actual modelling of the weld throat may not be essential to assess the overall structural stress at the toe locations.

Fig. 5-124 SeamWeldType = SolidWeld



When SeamWeldType = SolidWeld, three additional properties are used on the FE import analysis group

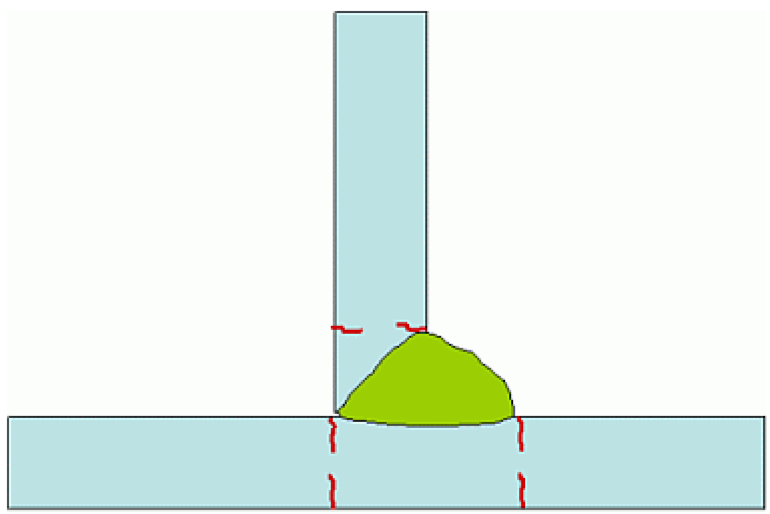
- WeldDefinitionFilename - the XML formatted file that defines the weld location and orientation.
- MaxWeldDepth - the maximum distance to recover stresses through the structure at the weld location. This distance can also be overridden on a per location basis in the weld definition file.
- NumWeldLayers - the number of points through the structure to recover stresses.

For more details on this method, see "Calculation of Stresses from Solid Elements" on page 294.

Generic

The "Generic" option is provided so that some users may reproduce a conservative option previously available in FE-Fatigue. In this option, damage is calculated for the top and bottom surfaces of all shell elements attached to the weld elements. All these locations are treated as weld toes. For example:

Fig. 5-125 Generic option "failure" locations. All positions are treated as weld toes.



5.8.3 Modeling Guidelines and Calculation Points

In order to obtain reasonably accurate fatigue analysis results, it is necessary to adhere to certain FE modelling guidelines. There are some general guidelines, and some that are specific to the different weld types when using thin-shell elements.

General Guidelines

The structures to be analysed should be meshed predominantly with 4-noded quadrilateral elements (CQUAD4 or equivalent) representing the mid-planes of the metal sheets. The weld bead should be represented by a single or double row of shells. The mesh around the weld should be regular, with elements of around 5 mm in size, and triangular elements in the vicinity of the weld should be avoided.

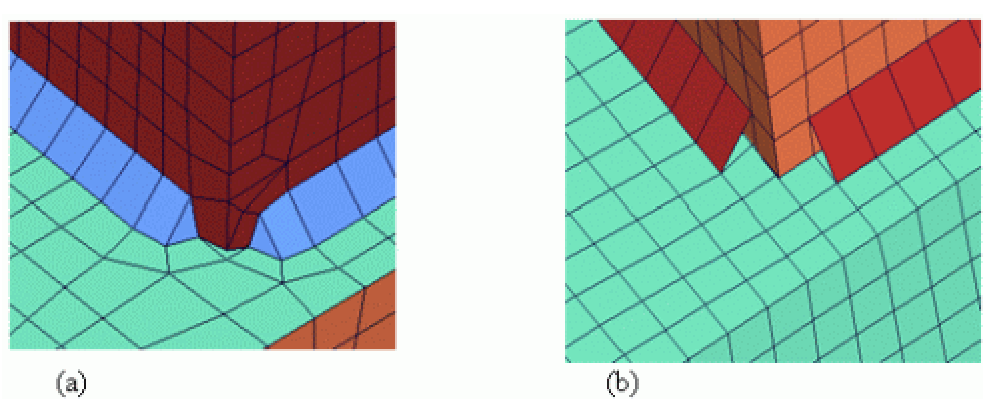
The elements representing the weld bead are used to identify the weld, and these elements must be placed in a unique property set for each weld or weld type. In addition, the weld element normals must be aligned so that the normals point outwards, that is towards the welder (except for Laser Overlap welds). The orientation of the sheet element normals is not important, except that within a single sheet they should be aligned in compliance with normal good practice.

Unaveraged node-at-element stresses are recovered for the weld toes and weld roots, based on the elements adjacent to the weld and used to make the fatigue calculations. Alternatively, the stresses may be averaged or calculated at the mid-point of the element edge, shared with the weld bead. Where weld throat calculations are made, the centroid stress from the weld bead element is used (averaged from the nodes or element edges).

Note All seam weld results for the root, toe and throat locations identified must have a non-zero thickness or the translation to FEI file will fail. Both displacement and grid point force methods use thicknesses to calculate stress or strain results that are written to the FEI file. These methods only use thicknesses from identified results locations.
All elements identified as part of results locations must be connected directly to the weld elements defined by the user in the FE Import Analysis Group objects. If you ensure that all weld elements and all shell elements connected directly to these have none zero thickness, you will not see a "zero thickness" error message. This applies to grid point force, displacement and stress methods.

In the past it has been recommended to "close" the ends of welds as illustrated in (a) of Figure 5-126. However, when using FE stresses, a simplified approach such as that illustrated in (b) appears to give quite good results while being significantly easier to build. When grid point forces are used, it may be beneficial to close the weld ends.

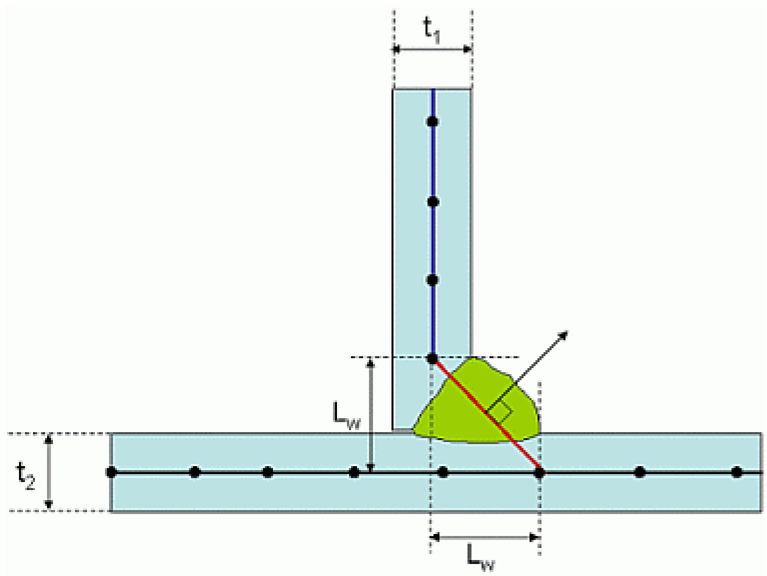
Fig. 5-126 Simplified modeling of weld ends. Approach (b) is adequate.



Fillet

Fillet welds may be modelled either by a row of inclined shell elements, as illustrated in Figure 5-127, or by two rows, as illustrated in Figure 5-128.

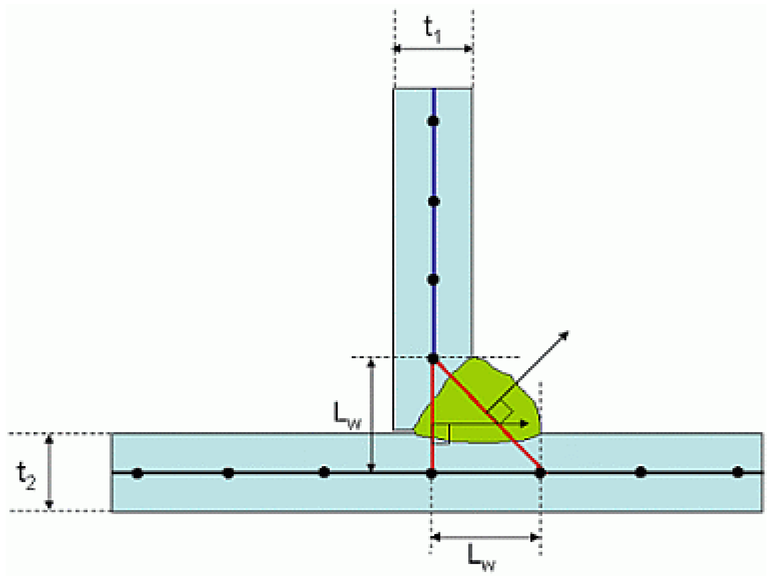
Fig. 5-127 Cross-section of fillet weld modeled with single inclined element



Please note the following:

1. The weld elements are in a different property set.
2. The weld element normal points “towards the welder”.
3. The nodes of the weld elements are in line with the weld toe.
4. The length L_w should be determined by the actual dimensions of the weld toe. A typical recommendation might be $L_w = t_1 + t_2$
5. The thickness of the weld element should represent the weld throat thickness, typically $L_w/\sqrt{2}$

Fig. 5-128   Cross-section of fillet weld modeled with two rows of elements



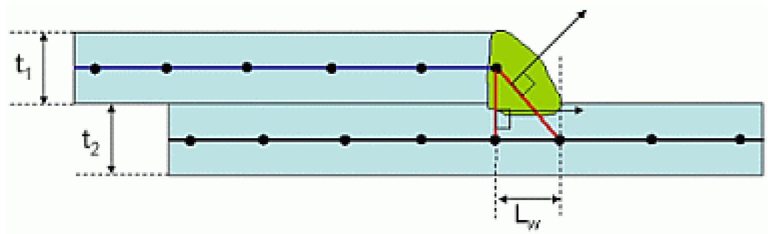
Please note the following:

1. Make sure that the weld element normals are aligned as shown. This orientation is required for the correct identification of weld toe and weld root.
2. A weld element thickness of $L_w/(2\sqrt{2})$ or in other words about $0.35(t_1 + t_2)$ is recommended, although the stiffness of this joint formulation will not be very sensitive to the weld element thickness.

Overlap and Laser Edge Overlap Welds

Overlap and Laser Edge Overlap welds are modelled in the same way, except that the dimensions of the weld bead will be smaller for a Laser Edge Overlap weld. The principles are the same as for fillet weld modelling; a row of nodes should be created along the line of the weld toe and one or two rows of elements used to model the weld, with the thickness of these weld elements representing the weld throat.

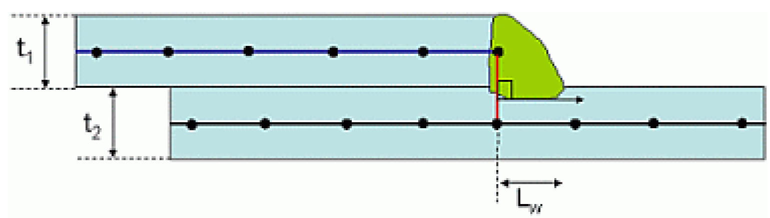
Fig. 5-129 Overlap weld modeling—two-element approach



For Overlap welds, typical leg length $L_w = t_1 + t_2$ and the weld element thickness should then be around $0.27(t_1+t_2)$.

The approach can be simplified, replacing the double row of elements by a single row, as illustrated in Figure 5-130. This method is more suitable for Laser Edge Overlap welds, because it allows weld throat fatigue calculations to be made; reasonable weld throat stresses require a single row of elements to be used.

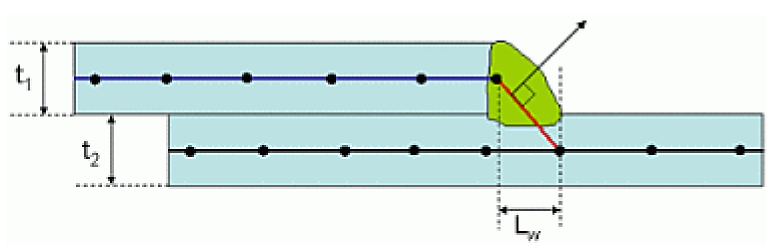
Fig. 5-130 Simplified modeling approach for overlap or laser edge overlap welds



In this case, for Overlap welds, the weld element thickness should be set to 2x the thickness of the thinner of the two sheets, but in any case not less than 3 mm.

For Laser Edge Overlap welds, the thickness of the weld element should correspond to the weld throat, which will be about 0.7x the upper sheet thickness.

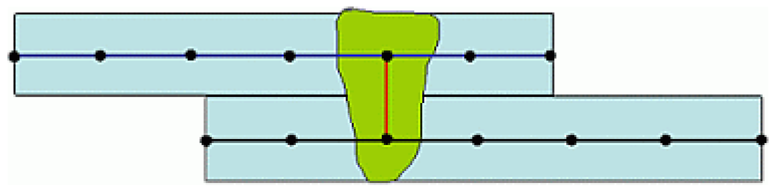
Fig. 5-131 Alternative method for overlap and laser edge overlap welds



Laser Overlap Welds

For Laser Overlap welds, the thickness of the weld element should represent the weld width. The weld width should be 90% of the thickness of the thinner sheet, but in any case not less than 1 mm.

Fig. 5-132 Laser overlap weld



5.8.4 Calculation Points and Stress Recovery

This section describes how the correct calculation points and stresses are identified for each weld type.

In general, the process is as follows, based on the weld elements and weld element normals:

- Identify weld toe and root elements.
- Identify the nodes on those elements that are shared with the weld elements.
- Identify which surface (Z1[bottom] or Z2[top]) of each element is the "weld top" surface, i.e., the surface at which crack development is predicted. This depends upon the direction of the weld toe or root element normal direction.

What follows next depends on whether stresses are being used directly or if grid point forces and moments or displacements are being used to derive the stresses for the fatigue calculation. This is controlled by the property "EntityDataType" in the Analysis Group properties. It also depends on whether the calculation is to be made at nodes (WeldResultLocation = NodeOnElement on the analysis group properties) or the mid-point of the element edges (WeldResultLocation = MidElementEdge).

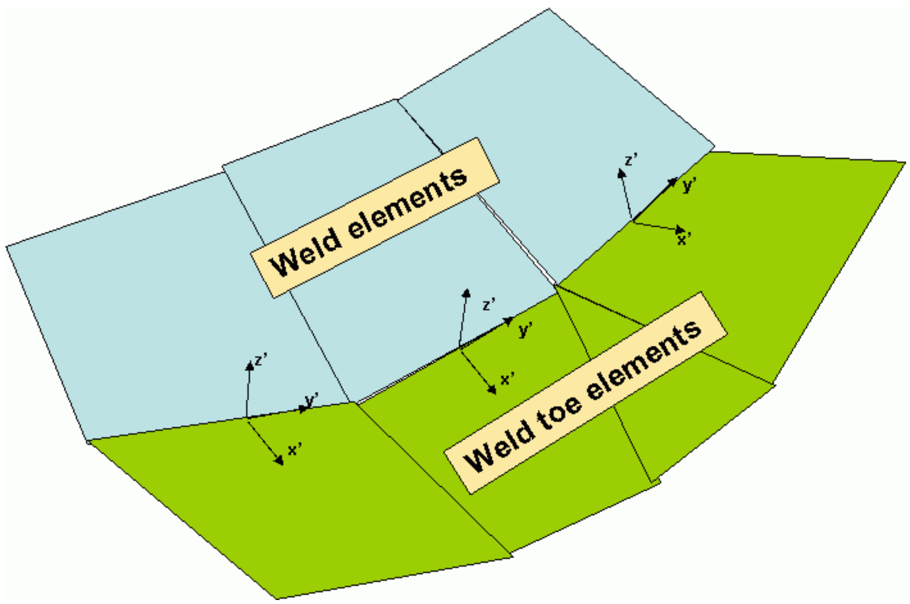
If stresses are being used directly:

- Recover stresses in element coordinates, for both surfaces of each element, with the "weld top" surface first.
- For weld toe and root elements, if WeldResultLocation = NodeOnElement, the stresses are unaveraged nodal stresses in local (element) coordinates.
- For weld toe and root elements, if WeldResultLocation = MidElementEdge, the stresses from the two nodes adjoining the weld are translated to a common local coordinate system based on the element edge and averaged. The local coordinate system has its z' axis as the upward surface normal at the weld toe and its y' axis along the edge such that the

x' axis is at right angles to and pointing outward from the weld toe, as illustrated in Figure 5-133 below.

- For weld throat elements, the stresses are averaged at the centroid in local (element) coordinates.
- For SeamWeldType = SolidWeld, the WeldResultLocation property is ignored and the stress recovery is defined by the weld definition file.

Fig. 5-133 Local coordinate systems for weld toe elements



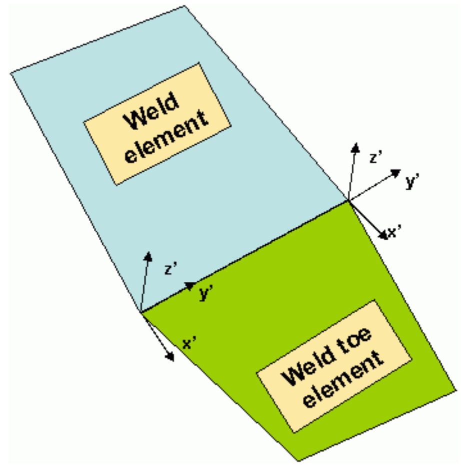
If grid point forces and moments are being used:

- The direct stress normal to the weld edge is recovered, for both surfaces of each element, with the "weld top" surface first. Note that the direct stress and shear stress components parallel to the weld are not calculated.
  - For weld toe and root elements, the stresses represent an average stress at the mid-point of the edge of the element adjoining the weld, normal to the edge. The method used to calculate these stresses is detailed later.
  - For weld throat elements, the stresses are averaged from the two edges of the element that run parallel to the weld.

If displacements (and rotations) are being used:

- The *strain* tensor at each weld toe or weld root node in each weld toe or weld root element is calculated based on the relative displacements and rotations of the nodes in each element. A local coordinate system is defined at each node, having its z' axis as the upward surface normal at the weld toe and its y' axis along the edge such that the x' axis is at right angles to and pointing outward from the weld toe, as illustrated in Figure 5-134 below:

Fig. 5-134 Local coordinate systems for displacement-based weld fatigue calculation



- For weld toe and root elements, if WeldResultLocation = MidElementEdge, the strains from the two nodes adjoining the weld are averaged. If the element is not significantly warped, the two coordinate systems will be the same, and identical with that in Figure 5-133.
- For weld throat elements, the strains from the corners of each element are averaged to the centroid in the element coordinate system.
- Within the analysis engine, the strain tensor is converted to stresses, assuming Hooke's law.

We will now look at the specific rules for each element type.

Fillet Welds

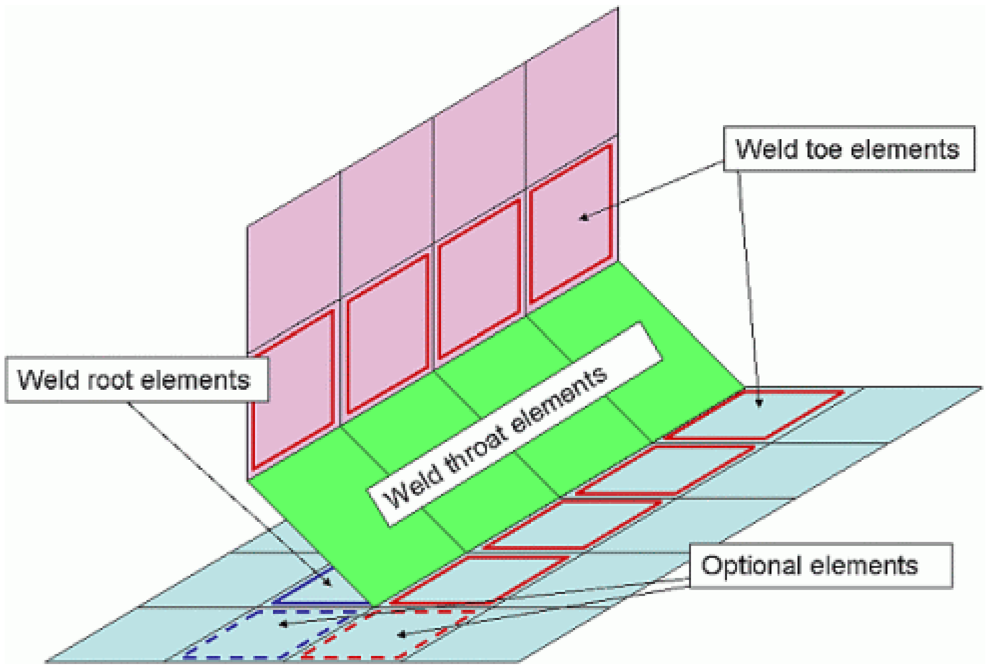
For fillet welds, the weld toe and root elements are outlined in Figure 5-135. The weld throat elements are not usually used for analysis but can be included; this functionality is currently a beta feature (see "Grouping and Weld Types" on page 266).

The weld toe elements are the elements in front of the weld that share nodes with the weld throat elements. The elements at the end of the weld line are optional. Whether or not they are used is controlled by the property WeldEndElements (Exclude or Include), which can be set on the FE Import Analysis Group. The effect of this property is slightly different depending on whether stresses, displacements, or grid point forces are being recovered from the FE results file, and if stresses or displacements are being used, if WeldResultLocation = MidElementEdge:

- If stresses are being recovered, or displacements are being used and WeldEndElements is set to Include, these elements will be included in the analysis, i.e., a fatigue calculation will be made for this element, unless WeldResultLocation = MidElementEdge.

- If WeldResultLocation = MidElementEdge, and/or if grid point forces are being used, no fatigue calculation is made in this element (because it doesn't share an edge with the weld), but the grid point forces in this element will contribute to the stress calculation. We recommend excluding these elements from the calculation (set WeldEndElements = Exclude on the Analysis Group properties).

Fig. 5-135 Weld toe and root elements for analysis for fillet welds (single element method)



The locations and "weld top" surfaces for calculation are indicated in Figure 5-136 and Figure 5-137 below.

Fig. 5-136 Weld toe and root calculation points and surfaces (weld top) for fillet weld—single-element method

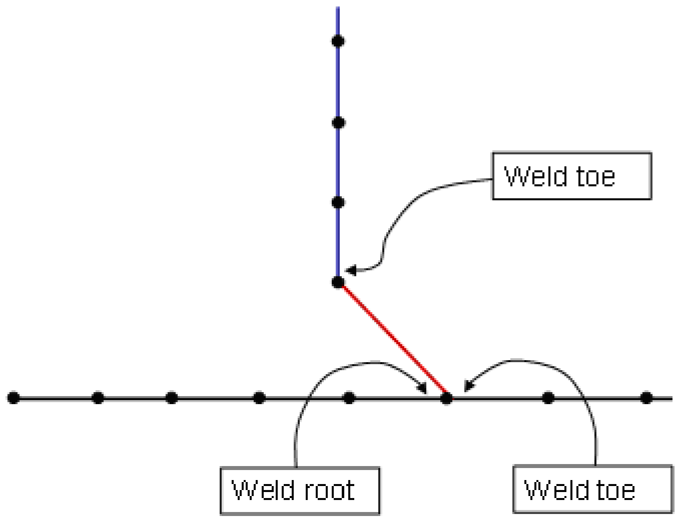
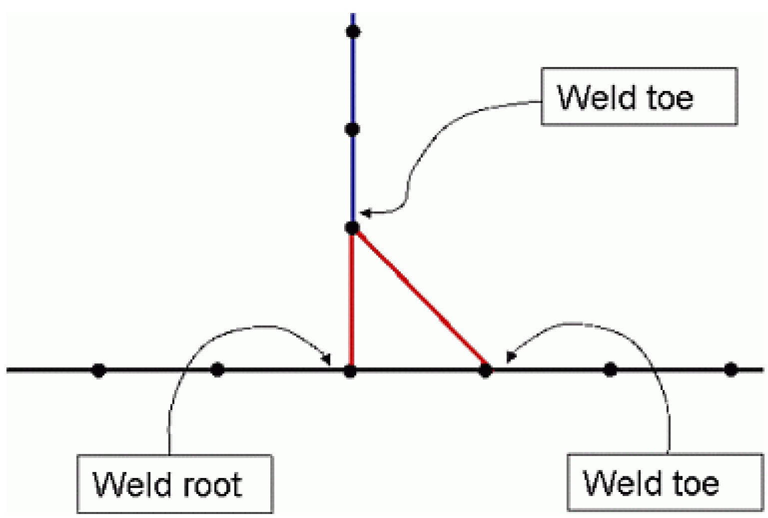


Fig. 5-137 Weld toe and root calculation points and surfaces (weld top) for fillet weld—two-element method

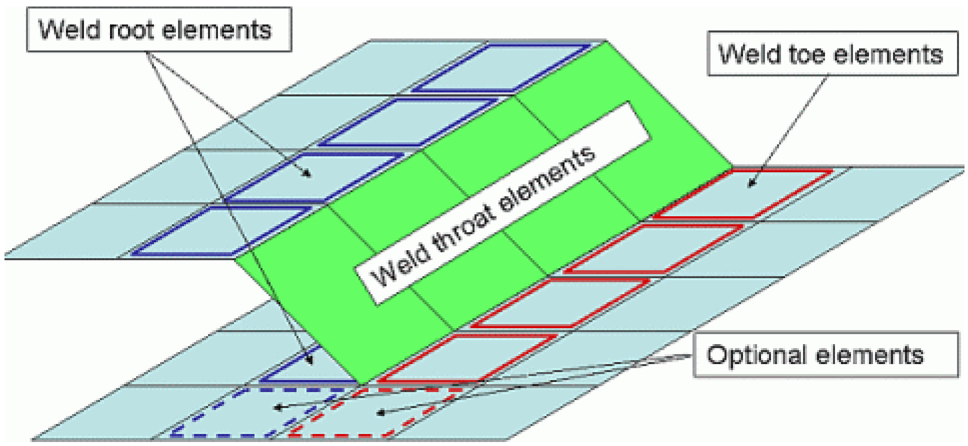


Overlap Welds

For overlap welds, the weld toe and root elements are outlined in Figure 5-138. The weld throat elements are not used for analysis.

Again, the elements at the end of the weld line are optional. Whether or not they are used is controlled by the property WeldEndElements (Exclude or Include) which can be set on the FE Import Analysis Group. We recommend excluding these elements.

Fig. 5-138 Weld toe and root elements for analysis for overlap welds (single-element method)



The locations and “weld top” surfaces for calculation are indicated in Figure 5-139 and Figure 5-141 below:

Fig. 5-139 Weld toe and root calculation points and surfaces (weld top) for overlap weld—two-element method

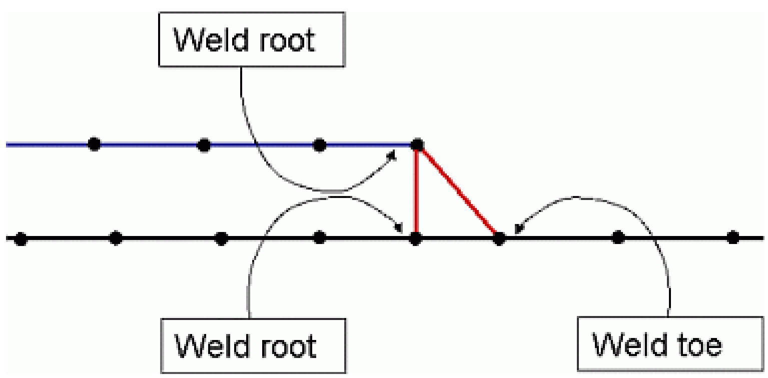


Fig. 5-140 Weld toe and root calculation points and surfaces (weld top) for overlap weld—single inclined-element method

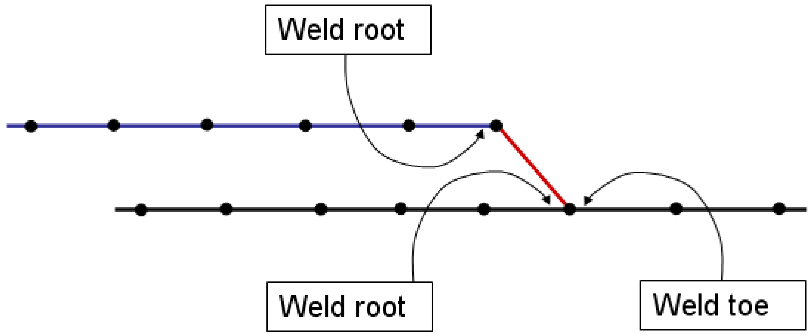
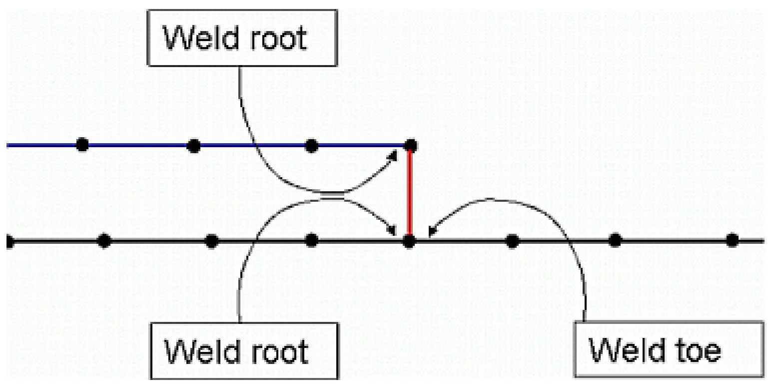


Fig. 5-141 Weld toe and root calculation points and surfaces (weld top) for overlap weld—single vertical-element method



Laser Overlap Welds

For Laser Overlap welds, the weld root and throat elements are outlined in Figure 5-142. Because a crack might initiate from either side of the weld throat, both sides of the weld throat element must be considered.

Again, the elements at the end of the weld line are optional. Whether or not they are used is controlled by the property WeldEndElements (Exclude or Include), which can be set on the FE Import Analysis Group. We recommend excluding these elements.

Fig. 5-142 Weld root and throat elements for analysis for laser overlap welds

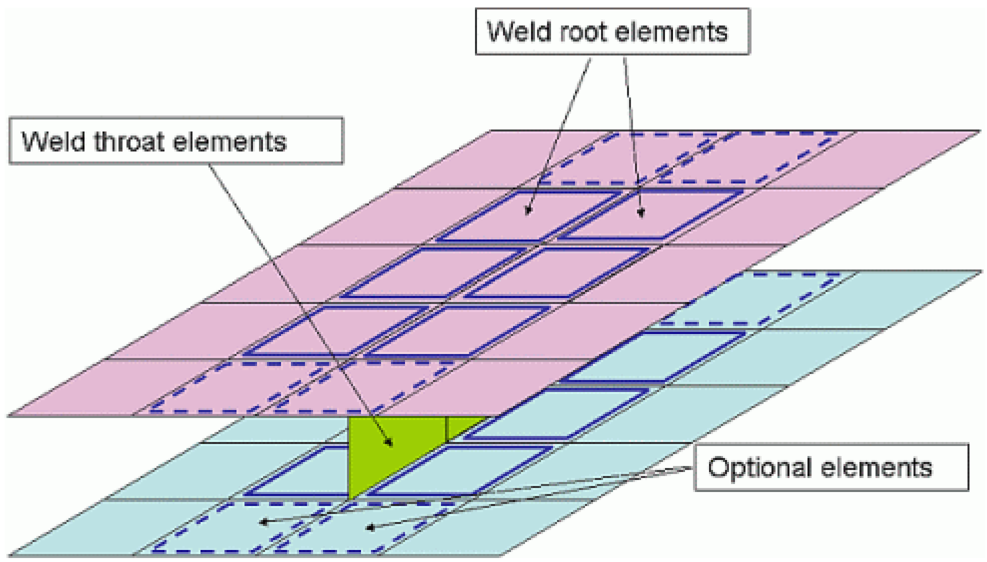
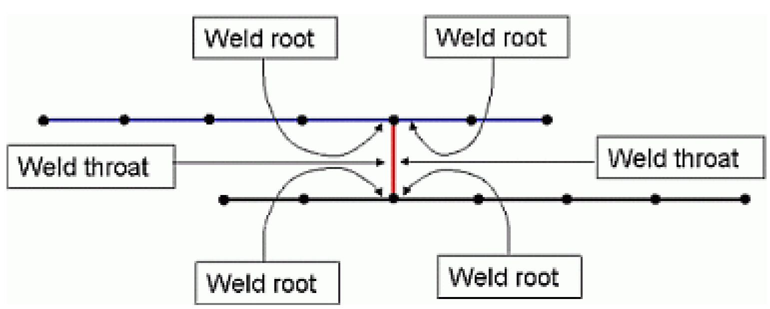


Fig. 5-143 Weld root and throat calculation points and surfaces for laser overlap weld—single vertical-element method

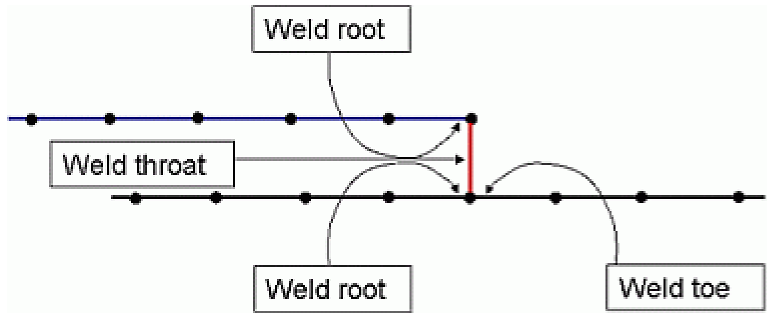


Laser Edge Overlap

Laser Edge Overlap welds are handled in the same way as Overlap welds, except that the weld throat may also be calculated. In this case, it is important to use a single row of elements to represent the weld. A double row of elements will not generate realistic weld throat stresses.

The calculation points and surfaces for a Laser Edge Overlap weld are illustrated in Figure 5-144.

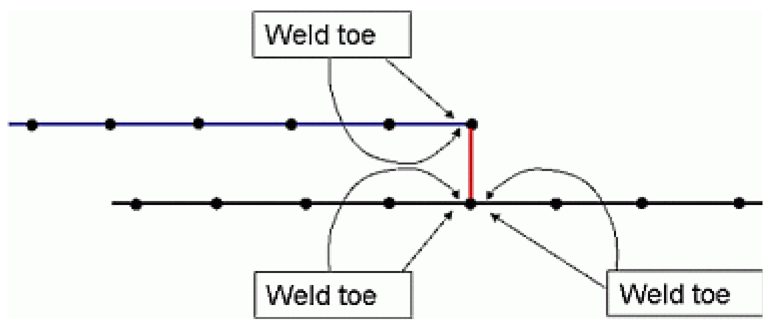
Fig. 5-144 Weld toe, root and throat calculation points and surfaces for laser edge overlap weld—single vertical-element method



Generic

If the WeldType is set to Generic, all surfaces of elements attached to the weld elements are used, and treated as weld toes. See the example in Figure 5-145:

Fig. 5-145 Example of calculation points and surfaces for generic method



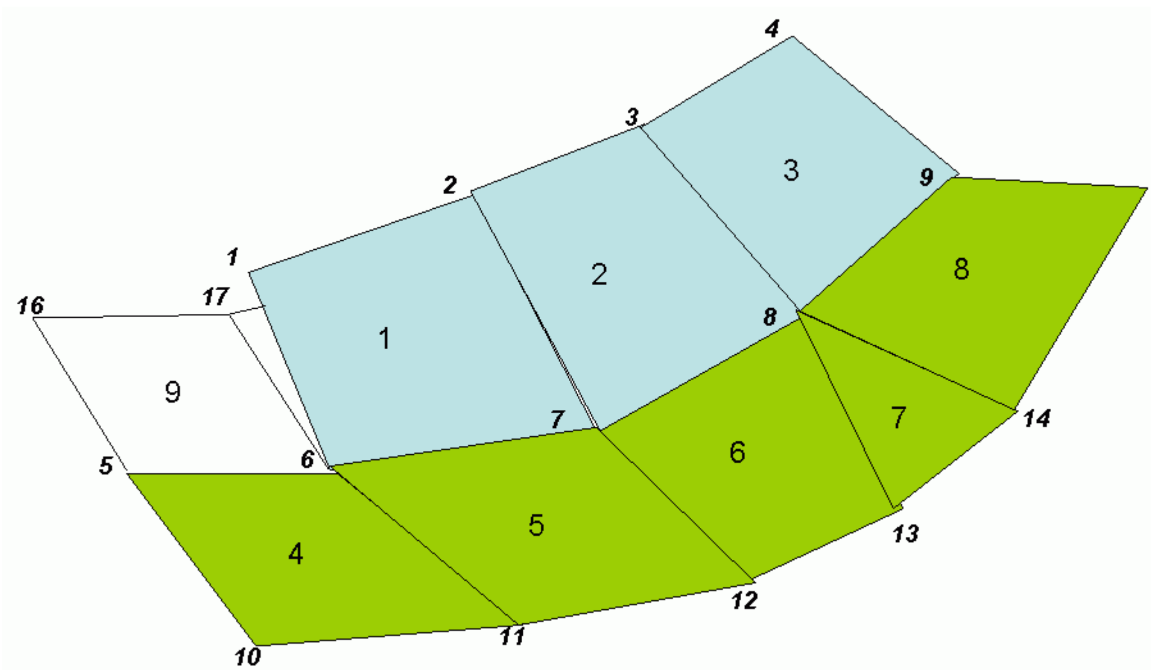
5.8.5 Calculation of Stresses from Grid Point Forces and Moments

As an alternative to using stresses directly from the FE results file, DesignLife also includes an approach in which the stresses required for a fatigue analysis are derived from nodal forces and moments. For each element for which a calculation will be made (weld toe, weld root, and possibly weld throat), we will require to know the grid point forces. These should be forces $F_{x,y,z}$ and moments $M_{x,y,z}$, in GLOBAL coordinates, for each node-on-element. Suitable output requests must be included in the FE analysis input decks to ensure that these results are available; for example, GPFORCE in NASTRAN and NFORC in ABAQUS. The method by which these forces and moments is used to calculate the stresses required for seam weld fatigue analysis is outlined below.

Consider the section of weld illustrated in Figure 5-146. The blue elements represent the weld and the green elements the weld toe elements. We are going to consider the calculation of stress in the weld toe elements.

Let us consider how we handle the weld toe elements, taking element 6 as an example. Before we can calculate the structural stresses, we need to determine the line forces and moments (force/moment per unit length) along the weld toe.

Fig. 5-146 Example weld



The basic information we need is the grid point forces and moments in the global coordinate system. We can think of these as the forces exerted by each element at each node to which it is connected.

We will denote these forces and moments as vectors, so the grid point forces and moments at node 7 in element 6 will be:

$$ {}_6^7F = \left({}_6^7F_x, {}_6^7F_y, {}_6^7F_z\right) \quad {}_6^7M = \left({}_6^7M_x, {}_6^7M_y, {}_6^7M_z\right) $$

The process then goes as follows:

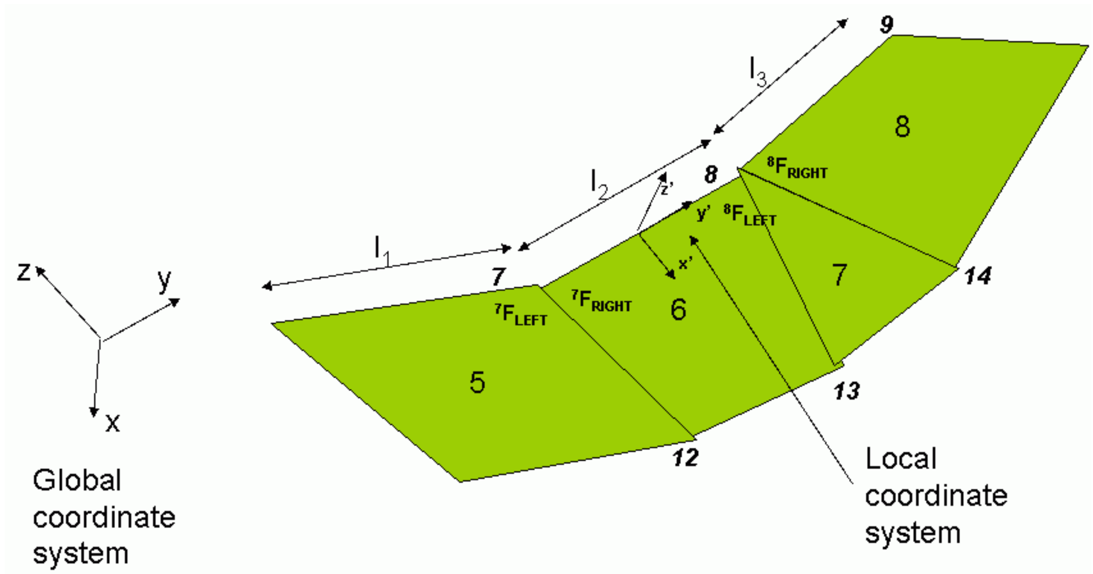
1. Identify weld toe elements and surfaces.

This works in the same way as if stresses are being used directly.

2. Define a local coordinate system.

A local coordinate system (x',y',z') is defined where the x' axis is normal to the element edge, and the z' axis is the average of the normals to element 6 at nodes 7 and 8, in an upward direction (relative to the surface at the weld toe), as illustrated in Figure 5-147.

Fig. 5-147 Local coordinate system for weld toe element



3. Calculate the total grid point forces.

Remaining with element 6, we want to know the total grid point forces exerted by this element at the weld toe at nodes 7 and 8.

At node 7 the total grid point forces and moments are:

$$ {}^{7}F = {}^{7}_{5}F + {}^{7}_{6}F \qquad {}^{7}M = {}^{7}_{5}M + {}^{7}_{6}M $$

At node 8 we also need to include the triangle element.

$$ {}^{8}F = {}^{8}_{6}F + {}^{8}_{7}F + {}^{8}_{8}F \qquad {}^{8}M = {}^{8}_{6}M + {}^{8}_{7}M + {}^{8}_{8}M $$

4. At each grid point, partition the forces and moments in proportion to edge length.

We are now going to share out the total grid point force at each of the nodes in proportion to the element edge lengths to left and right.

So at node 7, we want...

$$ {}^{7}F_{RIGHT} = {}^{7}F \cdot \frac{l_2}{l_1 + l_2} $$

and also

$$ {}^{7}M_{RIGHT} = {}^{7}M \cdot \frac{l_2}{l_1 + l_2} $$

And at node 8, we need....

$$^{8}F_{LEFT} = {}^{8}F \cdot \frac{l_2}{l_2 + l_3}$$

and

$$^{8}M_{LEFT} = {}^{8}M \cdot \frac{l_2}{l_2 + l_3}$$

5. Calculate the line forces and moments.

The line forces and moments f, m, are the forces and moments per unit length along the weld toe. The line forces and moments are distributed along the element edge, giving rise to the grid point forces. We will calculate first the values of the line forces at the nodes in each element, corresponding to the shares of the grid point forces calculated in the last step:

$$^{7}f_{RIGHT} = \frac{2}{l_2}\left(2 \cdot {}^{7}F_{RIGHT} - {}^{8}F_{LEFT}\right)$$

and of course

$$^{7}m_{RIGHT} = \frac{2}{l_2}\left(2 \cdot {}^{7}M_{RIGHT} - {}^{8}M_{LEFT}\right)$$

$$^{8}f_{LEFT} = \frac{2}{l_2}\left(2 \cdot {}^{8}F_{LEFT} - {}^{7}F_{RIGHT}\right)$$

and of course

$$^{8}m_{LEFT} = \frac{2}{l_2}\left(2 \cdot {}^{8}M_{LEFT} - {}^{7}M_{RIGHT}\right)$$

We then average these line forces and moments to give the value at the middle of the weld toe edge of element 6.

$$_{6}f = \frac{{}^{7}f_{RIGHT} + {}^{8}f_{LEFT}}{2}$$

...and:

$${}_{6}m = \frac{{}^{7}m_{RIGHT} + {}^{8}m_{LEFT}}{2}$$

6. Resolve to local coordinates.

The line forces are then resolved into the local coordinate system.

7. Calculate the stress normal to the weld toe.

The stress normal to the weld toe has a membrane and bending component. It can be calculated as follows:

$${}_{6}\sigma_{top,normal} = {}_{6}\sigma_{membrane} + {}_{6}\sigma_{bending} = \frac{{}_{6}f_{x'}}{t} + 6\frac{{}_{6}m_{y'}}{t^2}$$

$${}_{6}\sigma_{bottom,normal} = {}_{6}\sigma_{membrane} - {}_{6}\sigma_{bending} = \frac{{}_{6}f_{x'}}{t} - 6\frac{{}_{6}m_{y'}}{t^2}$$

...where t is the thickness of the weld toe elements.

8. Write results to the intermediate (FEI) file.

Stresses in the FEI file for weld toes normally look like this:

```
NodeID=31166,ElemID=27874,ElemType=Shell,ShellLoc=NotAShell,Coo
rdSys=2D,ECIdx=-
1,Thickness=10,SeamWeldLocation=Toe,PropID=22553
-3.9433088 -0.61234486 1.3359901 -2.3621747 0.70613515
0.73395562
```

comprising three stress components (x, y and xy) for “top” and “bottom” surfaces.

When grid point forces and moments are used to derive the stresses, the results are stored as element data, not node-at-element, and note also that only a single component of stress (the normal stress—in the x' direction) is written, for top and bottom surfaces. For example:

```
NodeID=-
1,ElemID=6,ElemType=Shell,ShellLoc=NotAShell,CoordSys=2D,ECIdx=
-1,Thickness=10,SeamWeldLocation=Toe,PropID=whatever
-3.9433088 0 0 -2.3621747 0 0
```

The rest of the fatigue calculation proceeds as normal, except that since there is only one stress component, there is not much point in doing critical plane analysis!

The process is very similar for weld root elements.

Additional Notes

This approach does not allow calculation of stresses for elements that share only a single node with the weld element, e.g., weld end elements. Considering the current example, no stresses can be calculated for element 4 or element 7 in Figure 5-146.

If we are calculating the stresses for an element such as element 5, which is the last element of the weld toe, we have the option of including or excluding the contribution from the element beyond the end of the weld toe. This is controlled by the "WeldEndElements" property (Include/Exclude) on the Analysis Group. If the element (like element 4) is present, and the option is True, the calculation can proceed as above, treating element 4 as if it were an adjacent weld toe element.

If WeldEndElements = Exclude, or if there is no available element, for example when the weld ends at an edge, as for element 8, then the step of accumulating the nodal forces and sharing them between adjacent elements can be omitted. In the case of element 8 for example, we would simply say:

$$^{9}F_{LEFT} = {}^{9}_{8}F$$

...and:

$$^{9}M_{LEFT} = {}^{9}_{8}M$$

The rest of the process is as described above.

For weld throats the basic calculation process is the same as for weld toe and root elements, except that the calculation is carried out at both the edges of the element parallel to the weld, and the resulting stresses averaged to the centroid before being written to the intermediate file.

5.8.6 Calculation of Stresses from Nodal Displacements and Rotations

The original implementation of the seam weld engine relied for accuracy on stresses calculated in NASTRAN using CUBIC stress recovery. CUBIC stress recovery calculates stresses at the grid points of CQUAD4 elements "using a strain gauge approach with cubic bending correction" rather than extrapolation from the integration points. Since this approach is not available in all FE codes, nCode-eDT offers a similar capability, whereby the strains (and hence the stresses) in each element are calculated based on the nodal displacements and rotations.

The calculation method applies to any linear quadrilateral element for which we require the stresses in the seam weld method. Triangular or quadratic elements are not supported. The information required for the calculation is the nodal displacements and rotations from the FE results file, calculated in the global coordinate system.

The calculation proceeds as follows:

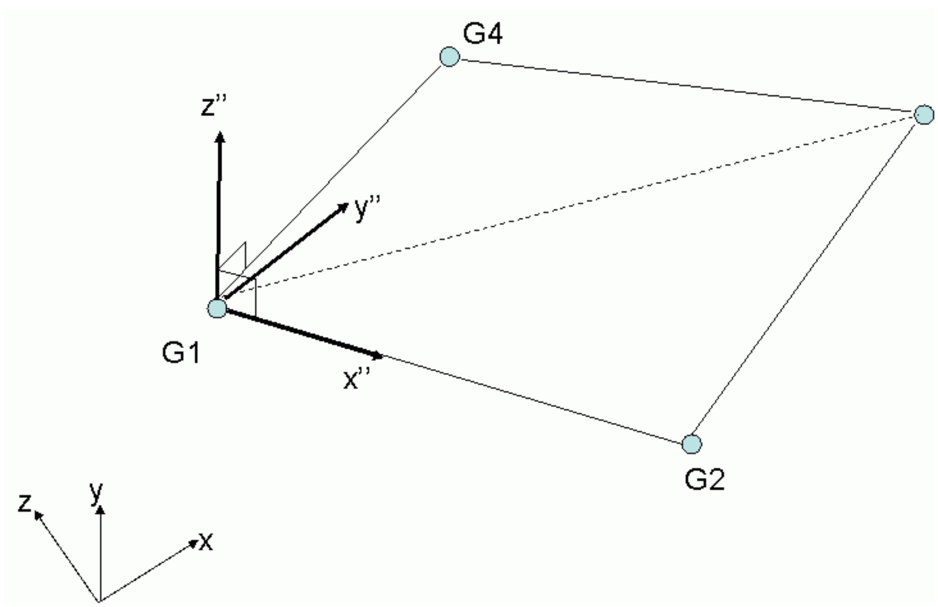
1. Identify weld toe elements and surfaces.

This works in the same way as if stresses are being used directly.

2. At each node, calculate the top and bottom surface strains in three directions.

Consider the quadrilateral element illustrated in Figure 5-148. Suppose that this is a weld toe element, and grid G1 is a node at the weld toe. The aim of the exercise is to determine the strain at G1 based on the displacements and rotations of all four nodes. The method is basically to determine the strains at G1 in three directions—G1-G2, G1-G3, and G1-G4, based on the displacements and rotations of the corresponding pairs of nodes. These three strains can then be used rather like the strains from a rosette to determine the strain tensor at G1.

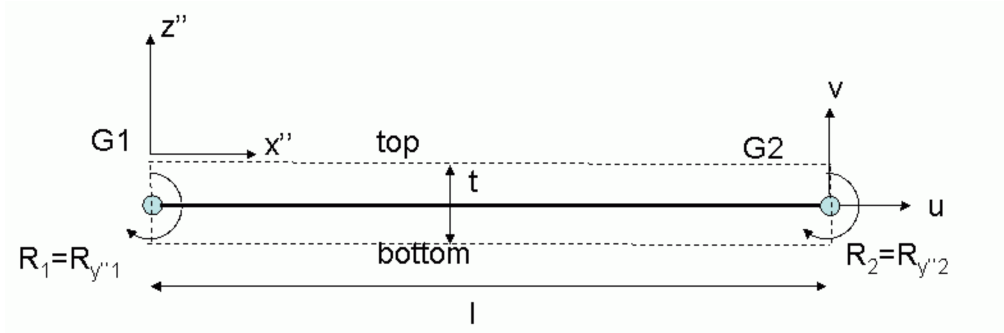
Fig. 5-148 Quadrilateral element, showing local coordinate system for strain calculation along G1-G2



First a local coordinate system is defined as illustrated, with the local x'' axis along G1-G2 and the local z'' axis vertical. If we transform the relative displacements and the rotations of G1 and G2 into this local coordinate system, we can look at the strain calculation in the G1-G2 direction as a 1-dimensional problem.

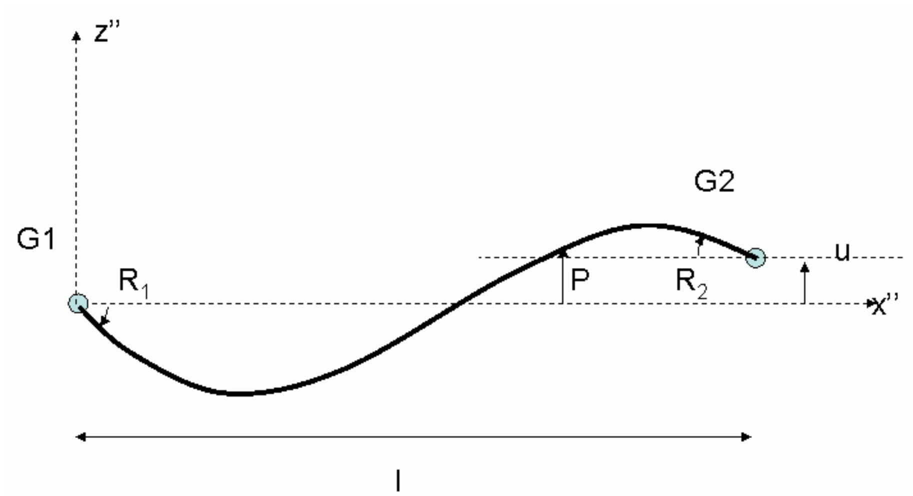
The problem is illustrated in Figure 5-149, where u and v are the relative displacements of the nodes in the local coordinate systems, and t is the sheet thickness.

Fig. 5-149 Calculation of strain between two nodes



The calculation assumes that the midsection of the element along G1-G2 deforms in the shape of a cubic, as illustrated in Figure 5-150 below.

Fig. 5-150 Cubic shape of deformed element



It can be shown that the top and bottom surface strains at G1 are given by:

$$\varepsilon_{1,top} = \frac{u}{l} - \frac{0.5t}{l}\left(\frac{\frac{6v}{l} + (4+F)R_1 + (2-F)R_2}{1+F}\right)$$

$$\varepsilon_{1,bot} = \frac{u}{l} + \frac{0.5t}{l}\left(\frac{\frac{6v}{l} + (4+F)R_1 + (2-F)R_2}{1+F}\right)$$

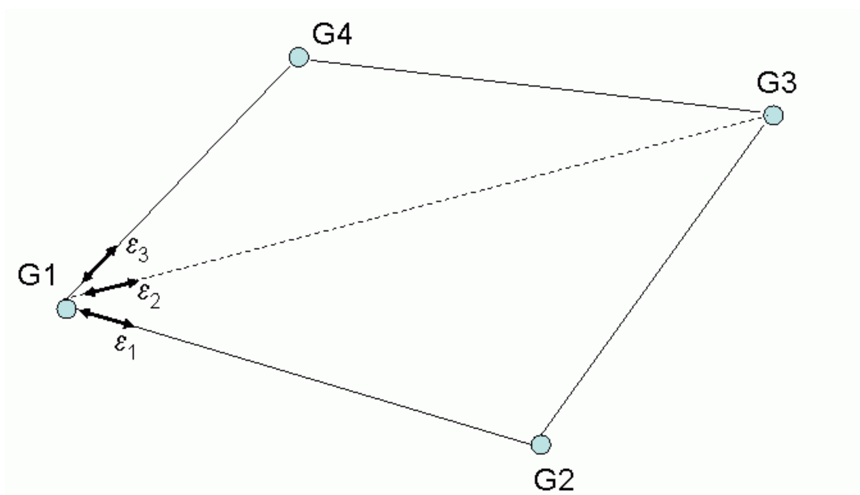
...where:

$$F = \frac{2t^2}{0.8333l^2(1-\nu_e)}$$

Because this calculation takes place at the translation stage of the calculation, where the material properties are not known, the elastic Poisson ratio e is assumed to be 0.3.

The process is repeated for the other pairs of nodes, G1-G3 and G1-G4, to give $\varepsilon_{2,top}$, $\varepsilon_{2,bot}$, $\varepsilon_{3,top}$ and $\varepsilon_{3,bot}$ as illustrated in Figure 5-151:

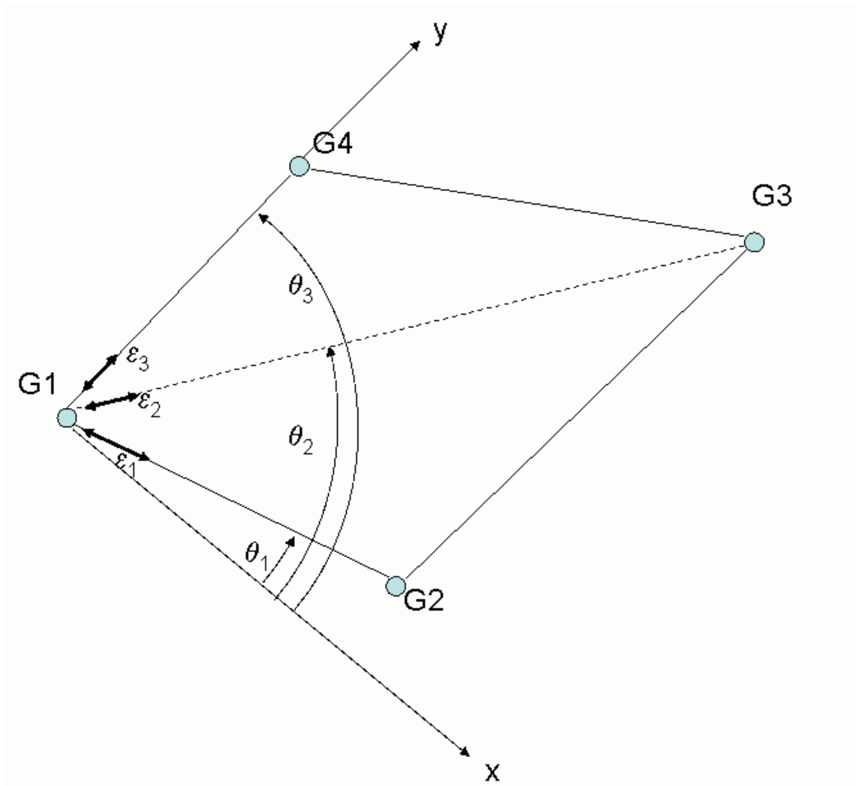
Fig. 5-151 Strains calculated in three directions at G1



3. Calculate the strain tensor.

From the three strains on the top or bottom surface, we can now calculate the top and bottom surface strain tensors at G1 by treating the strain values as if they came from the three legs of an arbitrary rosette. First a local coordinate system is defined, having one axis (y) parallel and one (x) normal to the weld toe. The coordinates are illustrated in Figure 5-152 assuming G1 and G4 are on the weld toe.

Fig. 5-152 Strain gauge rosette calculation to determine strain tensor in local coordinates



On top or bottom surfaces we can write the relationship between the strains in the three directions and the local strain tensor.

$$\begin{pmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \end{pmatrix} = \begin{pmatrix} \frac{1}{2}+\frac{1}{2}\cos 2\theta_1 & \frac{1}{2}-\frac{1}{2}\cos 2\theta_1 & \frac{1}{2}\sin 2\theta_1 \\ \frac{1}{2}+\frac{1}{2}\cos 2\theta_2 & \frac{1}{2}-\frac{1}{2}\cos 2\theta_2 & \frac{1}{2}\sin 2\theta_2 \\ \frac{1}{2}+\frac{1}{2}\cos 2\theta_3 & \frac{1}{2}-\frac{1}{2}\cos 2\theta_3 & \frac{1}{2}\sin 2\theta_3 \end{pmatrix} \begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{pmatrix}$$

Then:

$$\begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{pmatrix} = M^{-1} \begin{pmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \end{pmatrix}$$

When displacements are used, it is these derived strain tensors that are written to the intermediate (FEI) file instead of the usual stress tensors.

4. Convert to stresses in the fatigue engine.

The strains are converted to stresses inside the seamweld analysis engine (at which point the software does know what the correct modulus and Poisson ratio are):

$$\sigma_{xx} = \frac{E}{1-\nu^2}\left(\varepsilon_{xx} + \nu\varepsilon_{yy}\right)$$

$$\sigma_{yy} = \frac{E}{1-\nu^2}\left(\varepsilon_{yy} + \nu\varepsilon_{xx}\right)$$

$$\tau_{xy} = G\gamma_{xy} = \frac{E}{2(1+\nu)}\gamma_{xy} = \frac{E}{1+\nu}\varepsilon_{xy}$$

Additional Notes

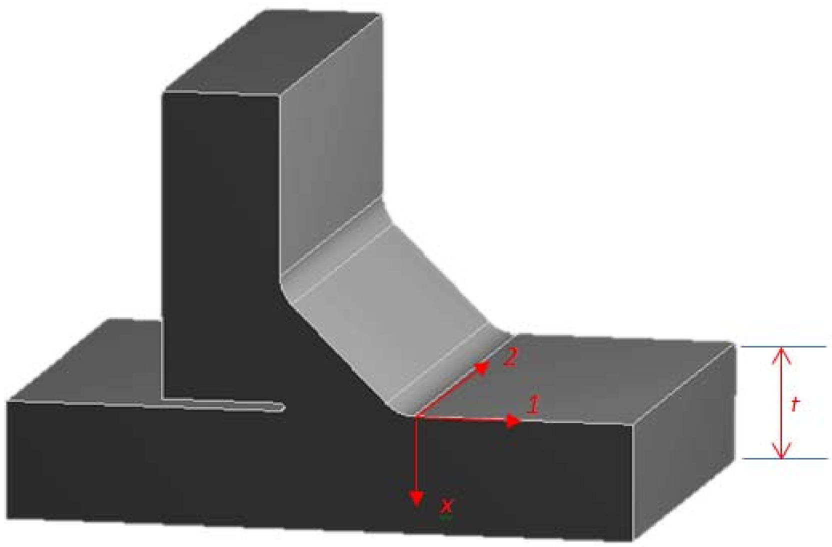
Please note that this method relies on relative displacements between nodes and on an assumption of small displacements. Its accuracy is likely to be compromised by:

- Numerical issues if the displacements are very small
- Large displacements (sufficient to require consideration of geometric non-linearity)
- Warped elements

5.8.7 Calculation of Stresses from Solid Elements

The seam weld fatigue damage calculation in DesignLife uses a structural stress at the location of interest, and the bending ratio associated with that stress. This methodology is also applicable to thicker structures that need to be modeled with solid elements.. With the goal of not requiring a specialized meshing scheme, the stress integration method outlined in the ASME Boiler & Pressure Vessel Code VIII, Division 2 has been adopted. This is detailed in 2008a Annex 5.A "Linearization of Stress Results for Stress Classification". This method integrates stress components along user defined Stress Classification Lines (SCL) to determine membrane and bending stresses.
In DesignLife, the 2D stress state perpendicular to the prescribed SCL is evaluated.

Fig. 5-153 Path of the SCL (Stress Classification Lines)



In Fig. 5-153 Path of the SCL (Stress Classification Lines), the x axis represents the path of the SCL. The plane defined by axes 1 and 2 contains the stress components that will be evaluated. The stresses in the model are first transformed into this local reference system. These stress components are then integrated using equation 1, through the model along the SCL.

$$\sigma_{ij,m} = \frac{1}{t}\int_0^t \sigma_{ij}\, dx \qquad (1)$$

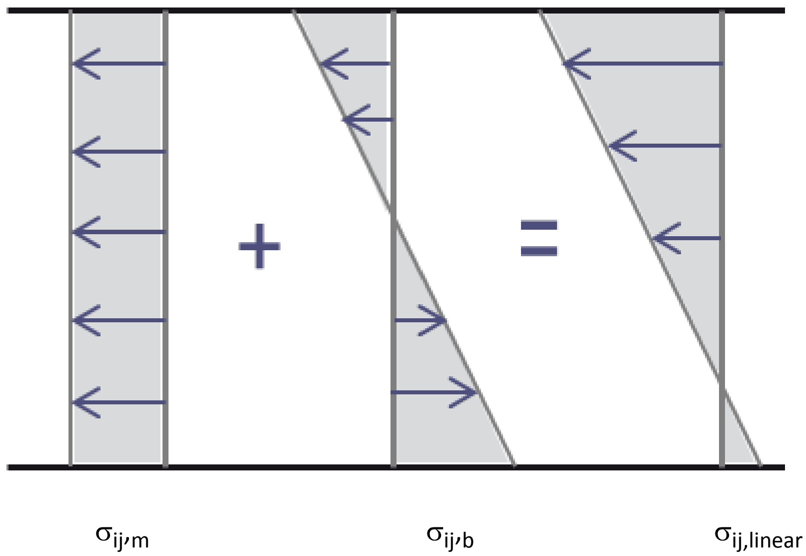
Where σ_{ij} is σ_{11}, σ_{22}, and σ_{12} in the local reference system, and $\sigma_{ij,m}$ is the membrane stress of each of the stress components. A second integration is performed using equation 2 to obtain the bending stress contribution for each of the stress components.

$$\sigma_{ij,b} = \frac{6}{t^2}\int_0^t \sigma_{ij}\left(\frac{t}{2} - x\right) dx \qquad (2)$$

The stress quantity used for the seam weld damage parameter is the sum of the membrane and bending stresses.

Note There is additionally a peak stress quantity defined by the ASME procedure but this is not used in the DesignLife calculation.

Fig. 5-154   Stress Classification Lines



In DesignLife the SCL is determined through the solid model using information from a weld definition file as shown in Fig. 5-155 Sample Weld Definition fileand selected via *WeldDefinitionFilename* on the FE import analysis group. This file contains a starting location, a vector to define the direction of the SCL, a vector to define the primary stress direction perpendicular to the SCL, and an optional max weld depth distance. The detail of this file format is described in File Formats.

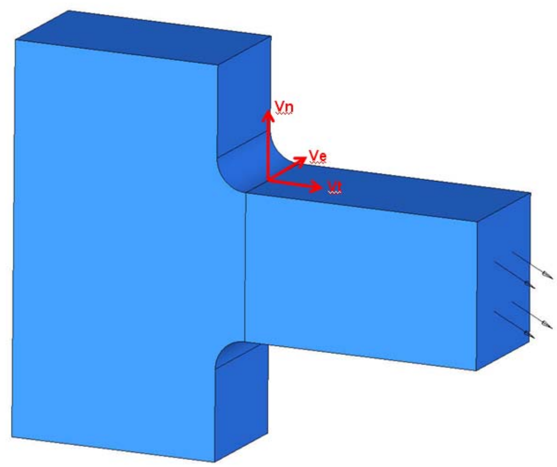
Fig. 5-155   Sample Weld Definition file

```
<WeldDefinitions>
<WeldLine name="Weld 1 - Root Top Sheet">
<WeldDef Location="-9.8, 0.78,  5.0" NormalVector="0.0, -1.0, 0.0" ToeVector="1.0, 0.0, 0.0"
id="11" MaxWeldDepth="1.4"/>
<WeldDef Location="-9.8, 0.78, 15.0" NormalVector="0.0, -1.0, 0.0" ToeVector="1.0, 0.0, 0.0"
id="12"/>
<WeldDef Location="-9.8, 0.78, 25.0" NormalVector="0.0, -1.0, 0.0" ToeVector="1.0, 0.0, 0.0"
id="13"/>
</WeldLine>

</WeldDefinitions >
```

The reference system defining a seam weld on a solid model is shown in Fig. 5-156 Seam Weld solid model referencing systembelow.

Fig. 5-156 Seam Weld solid model referencing system



Vn is defined as a unit vector in the direction opposite of the path of the SCL.

Vt defines a unit vector in the plane perpendicular to Vn, and perpendicular to the weld toe.

Ve defines a unit vector in the plane perpendicular to Vn, and perpendicular to Vt.

The first step is to determine the actual starting location of the SCL on the model (see also the note on Starting Locations in "Calculation Method" on page 300). This is accomplished by projecting the weld starting location onto the surface of the model along the vector Vn . The surface is chosen based on proximity to the defined starting location, and having an appropriate surface normal. An allowable surface normal must have an angle of less than 90 degrees between it and Vn. The location of starting point will be designated 'X_{start}'.

The second step is to determine the opposite end of the SCL. This is either the exit location out of the part, or a location inside the part if the max weld depth distance is reached prior to exiting the part. The max weld depth is initially defined as a property '**MaxWeldDepth**' in the FE Import group object, but may also be specified in the weld definition file, where it will override the property in the FE Import group object. The location of end point will be designated 'X_{end}'.

With the start point and the end point determined, a series of layers will be modeled along the path between them. The number of layers is specified as a property '**NumWeldLayers**' in the FE Import group object.

$$Dist = \text{distance between the } X_{start} \text{ and } X_{end}.$$

$$Thickness = Min(MaxWeldDepth, Dist)$$

$$Layer\_Thickness = \frac{Thickness}{NumWeldLayers}$$

The stress recovery points are at the center of each layer, and located at:

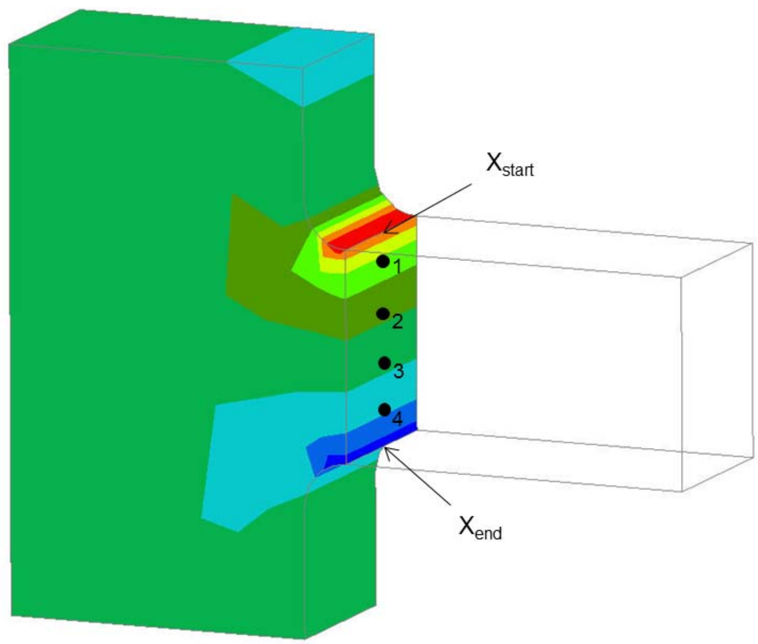
$X_i = X_{start} - (i-0.5)*Layer\_Thickness*Vn$

Where: X_{start} is the starting position of the SCL

X_i is the location of the center of the i^{th} layer

$i = 1$ to NumWeldLayers

Fig. 5-157 Stress recovery points



At each stress recovery point: (where i = 1 to NumWeldLayers):

Identify the element containing that point

Calculate the stress tensor at that point by interpolation of the element nodal stresses.

σxx_i, σyy_i, σzz_i, τxy_i, τyz_i, τxz_i

Transform the stress tensor to a reference system defined by Vt, Ve and Vn

The stress components in the plane perpendicular to Vt are σtt, σee, and τte

$\sigma tt_i = (\sigma xx_i * Vtx\text{^}2) + (\sigma yy_i * Vty\text{^}2) + (\sigma zz_i * Vtz\text{^}2) + 2*(Vtx*Vty*\tau xy_i + Vty*Vtz*\tau yz_i + Vtx*Vtz*\tau xz_i)$

$\sigma ee_i = (\sigma xx_i * Vex\text{^}2) + (\sigma yy_i * Vey\text{^}2) + (\sigma zz_i * Vez\text{^}2) + 2*(Vex*Vey*\tau xy_i + Vey*Vez*\tau yz_i + Vex*Vez*\tau xz_i)$

$\tau te_i = (\sigma xx_i * Vtx*Vex) + (\sigma yy_i * Vty*Vey) + (\sigma zz_i * Vtz*Vez) + (Vtx*Vey*\tau xy_i + Vty*Vez*\tau yz_i + Vtx*Vez*\tau xz_i + Vex*Vty*\tau xy_i + Vey*Vtz*\tau yz_i + Vex*Vtz*\tau xz_i)$

To calculate the membrane component of the stresses:

$$\sigma_{tt,m} = \frac{1}{NumWeldLayers} \sum_{1}^{NumWeldLayers} \sigma_{tt,i}$$

$$\sigma_{ee,m} = \frac{1}{NumWeldLayers} \sum_{1}^{NumWeldLayers} \sigma_{ee,i}$$

$$\tau_{te,m} = \frac{1}{NumWeldLayers} \sum_{1}^{NumWeldLayers} \tau_{te,i}$$

To calculate the bending component of the stresses

$$\sigma_{tt,b} = -\frac{6}{Thickness^2} \sum_{1}^{NumWeldLayers} \left(\sigma_{tt,i} - \sigma_{tt,m}\right) \cdot (i - 0.5) \cdot Layer\_Thickness^2$$

$$\sigma_{ee,b} = -\frac{6}{Thickness^2} \sum_{1}^{NumWeldLayers} \left(\sigma_{ee,i} - \sigma_{ee,m}\right) \cdot (i - 0.5) \cdot Layer\_Thickness^2$$

$$\tau_{te,b} = -\frac{6}{Thickness^2} \sum_{1}^{NumWeldLayers} \left(\tau_{te,i} - \tau_{te,m}\right) \cdot (i - 0.5) \cdot Layer\_Thickness^2$$

The seamweld engine is provided with a 2D stress tensor for the top and bottom of the weld section:

The stresses at the top surface are:

$$\sigma_{tt,linear} = \sigma_{tt,m} + \sigma_{tt,b}$$

$$\sigma_{ee,linear} = \sigma_{ee,m} + \sigma_{ee,b}$$

$$\tau_{te,linear} = \tau_{te,m} + \tau_{te,b}$$

The stresses at the bottom surface are:

$$\sigma_{tt,linear} = \sigma_{tt,m} - \sigma_{tt,b}$$

$$\sigma_{ee,linear} = \sigma_{ee,m} - \sigma_{ee,b}$$

$$\tau_{te,linear} = \tau_{te,m} - \tau_{te,b}$$

This generates top and bottom surface stresses in the same form as would be generated from a thin-shell model and so the calculation method used in the Seam Weld Analysis engine described below is equally applicable to stresses calculated from either solid or thin-shell models.

5.8.8 Calculation Method

Note Care should be taken when the weld *starting location* is defined near joints made using interface elements and contact definitions. The contact faces will be considered as free edges and therefore gaps will be present. This means that these free edges will be found in the projection of the starting location along the vector *Vn*, and therefore may be taken as the starting location for the SCL (Stress Classification Lines), depending on their surface normals and proximity to the defined starting location.

The result of the weld definition can be viewed in the *FEDisplay*. It shows the positions of each *location point* in the weld definition file, and the weld depth.

Having identified the FE entities (nodes/elements) from which stresses will be recovered for calculation, and the correct surfaces of each element to use, as described in the previous section, the calculation proceeds as follows:

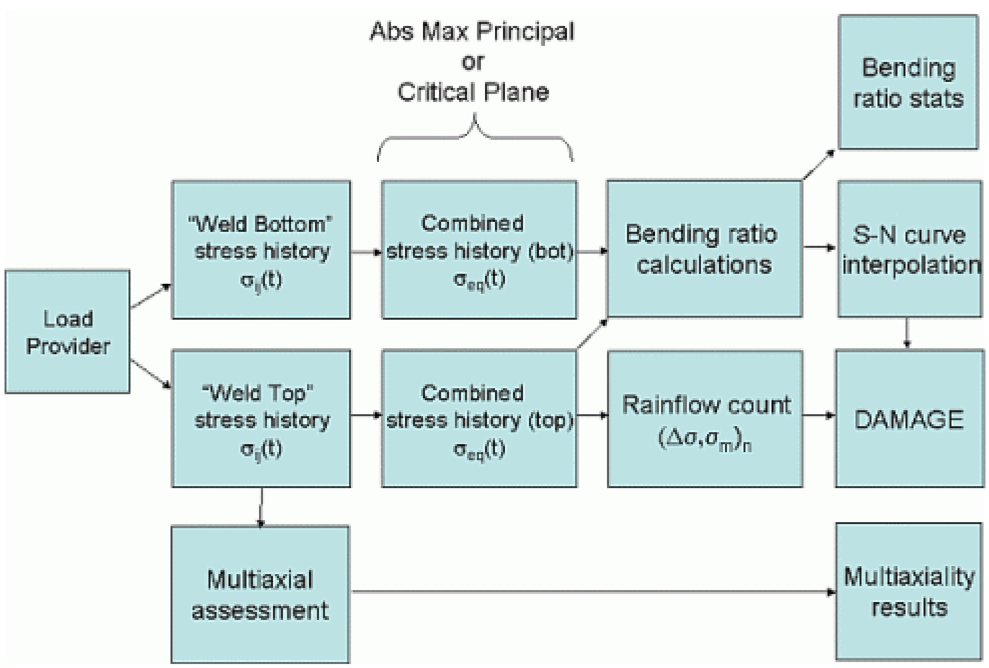
1. For each FE entity being analyzed, determine "weld-top" and "weld-bottom" surface stress tensor time histories and calculate "weld-top" and "weld-bottom" surface combined stress histories (Abs Max Principal and Critical Plane).
2. Where appropriate, use "weld-top" and "weld-bottom" surface stresses to determine the degree of bending in the loading on the weld at the calculation point in question.
3. Where appropriate, use the bending ratio to interpolate between "stiff" and "flexible" S-N curves.

4. Rainflow count the “weld top” stress history.
5. Carry out the damage calculation. The main features of the damage calculation are the same as for a normal S-N approach. A mean stress correction can optionally be applied using the FKM method. There is also an optional thickness (size effect) correction.

The main features of the S-N analysis process, the optional multiaxial assessment, and the FKM mean stress correction are described in the section “Standard SN Analysis Engine” on page 84.

The calculation process is summarized in Figure 5-158 below:

Fig. 5-158 Seam weld analysis process



Let us now examine each of these steps in more detail.

Stress Calculation and Combination

The stress tensor histories for the “weld top” and “weld bottom” surfaces are calculated in the usual way, based on the load provider.

For weld toe and root calculations, the stress may be an unaveraged node-at-element stress, or it may be averaged to the mid-point of the element edge at the weld toe or root. Mid-element-edge results are treated and postprocessed as if they were element results.

For weld throat calculations, the stress will be averaged to the centroid.

$$\begin{pmatrix}\varepsilon_1\\ \varepsilon_2\\ \varepsilon_3\end{pmatrix} = M\begin{pmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{pmatrix} \quad \text{where} \quad M = \begin{pmatrix} \frac{1}{2}+\frac{1}{2}\cos 2\theta_1 & \frac{1}{2}-\frac{1}{2}\cos 2\theta_1 & \frac{1}{2}\sin 2\theta_1 \\ \frac{1}{2}+\frac{1}{2}\cos 2\theta_2 & \frac{1}{2}-\frac{1}{2}\cos 2\theta_2 & \frac{1}{2}\sin 2\theta_2 \\ \frac{1}{2}+\frac{1}{2}\cos 2\theta_3 & \frac{1}{2}-\frac{1}{2}\cos 2\theta_3 & \frac{1}{2}\sin 2\theta_3 \end{pmatrix}$$

Abs Max Principal Method

If the Abs Max Principal stress is to be used, this is determined for the "weld top" surface in the usual way from the in-plane principals:

$$\sigma_{1,2} = \frac{\sigma_{xx} + \sigma_{yy}}{2} \pm \sqrt{\left(\frac{\sigma_{xx} - \sigma_{yy}}{2}\right)^2 + \sigma_{xy}{}^2}$$

$$\sigma_{eq,top} = \sigma_{AMP} = \sigma_1 \; if \; |\sigma_1| > |\sigma_2| \; otherwise \; \sigma_{AMP} = \sigma_2$$

The orientation ϕ_p of the abs max principal with respect to the element x-axis is determined from:

$$\tan 2\phi_p = \frac{2\sigma_{xy}}{\sigma_{xx} - \sigma_{yy}}$$

The bottom surface stress is calculated by determining the direct stress in the same direction as the top surface Abs Max Principal stress.

$$\sigma_{eq,bot} = \frac{\sigma_{xx} + \sigma_{yy}}{2} + \frac{\sigma_{xx} - \sigma_{yy}}{2} \cdot \cos\phi_p + \sigma_{xy}\sin\phi_p$$

Note that the "weld top" stress is the primary stress used for the fatigue calculation – the "weld bottom" surface stress is used only to determine the degree of bending.

Critical Plane Method

If the critical plane method is being used, the "weld top" and "weld bottom" surface stresses are both determined on a number of planes at 10-degree intervals:

$$\sigma_\phi = \frac{\sigma_{xx} + \sigma_{yy}}{2} + \frac{\sigma_{xx} - \sigma_{yy}}{2} \cdot \cos\phi + \sigma_{xy}\sin\phi$$

ϕ = 0, 10, 20 ...170 degrees

Weld Normal Method

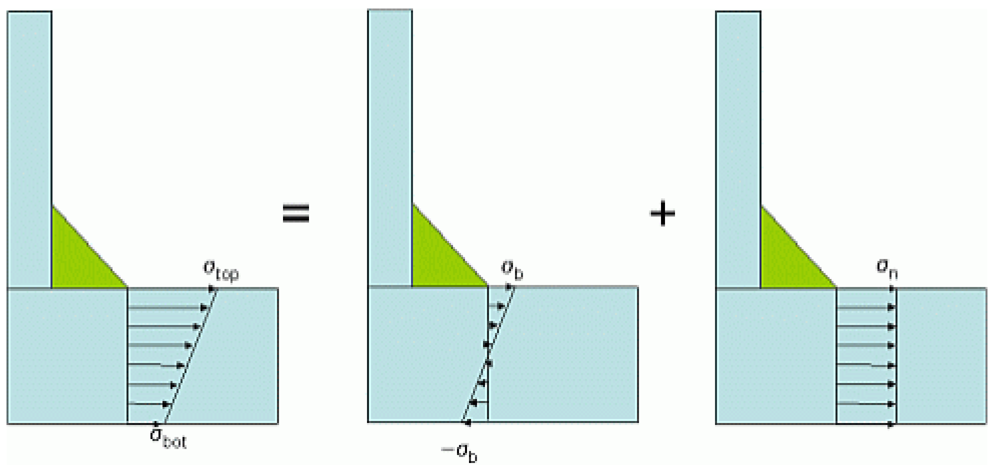
The weld normal method is available only if the calculation are being made at the mid-point of the element edges (i.e., if WeldResultsLocation = MidElementEdge).

In this case, the stresses are in local coordinate systems in which the direct stress normal to the weld toe is the x-component. The top and bottom surface stresses will therefore be the corresponding x-components. If grid point forces are being used, this is the option that should be selected—it will give the same results as the other two combination methods, but is more efficient.

Determine the Degree of Bending

The stress in a shell can be partitioned into two components—a bending stress σ_b and a direct or normal stress σ_n.

Fig. 5-159 Shell stress partitioned into bending and direct (membrane) components



The instantaneous degree of bending at a given calculation point is defined by the bending or flex ratio r, which always lies between 0 (no bending) and 1 (pure bending):

$$r = \frac{|\sigma_b|}{|\sigma_n| + |\sigma_b|}$$

$$r = \frac{|\sigma_{eq,top} - \sigma_{eq,bot}|}{|\sigma_{eq,top} + \sigma_{eq,bot}| + |\sigma_{eq,top} - \sigma_{eq,bot}|}$$

During the fatigue calculation, the minimum, maximum, average, and standard deviation of the bending ratio is calculated over all points in the loading history.

The average bending ratio is the most important value for the fatigue calculation; it is a weighted average calculated as follows:

$$\bar{r} = \frac{\sum_n r \cdot \sigma_{eq,top}^2}{\sum_n \sigma_{eq,top}^2}$$

Material Properties and Interpolation

The material dataset required for a seam weld calculation consists of a parent dataset of type “nCode SN Seam Weld” containing basic material properties, and either one or two children containing S-N curves. If two curves are present, one must represent “flexible” or bending loading conditions (r = 1) and the other “stiff” or membrane loading conditions (r = 0).

The generic (parent) data is as follows:

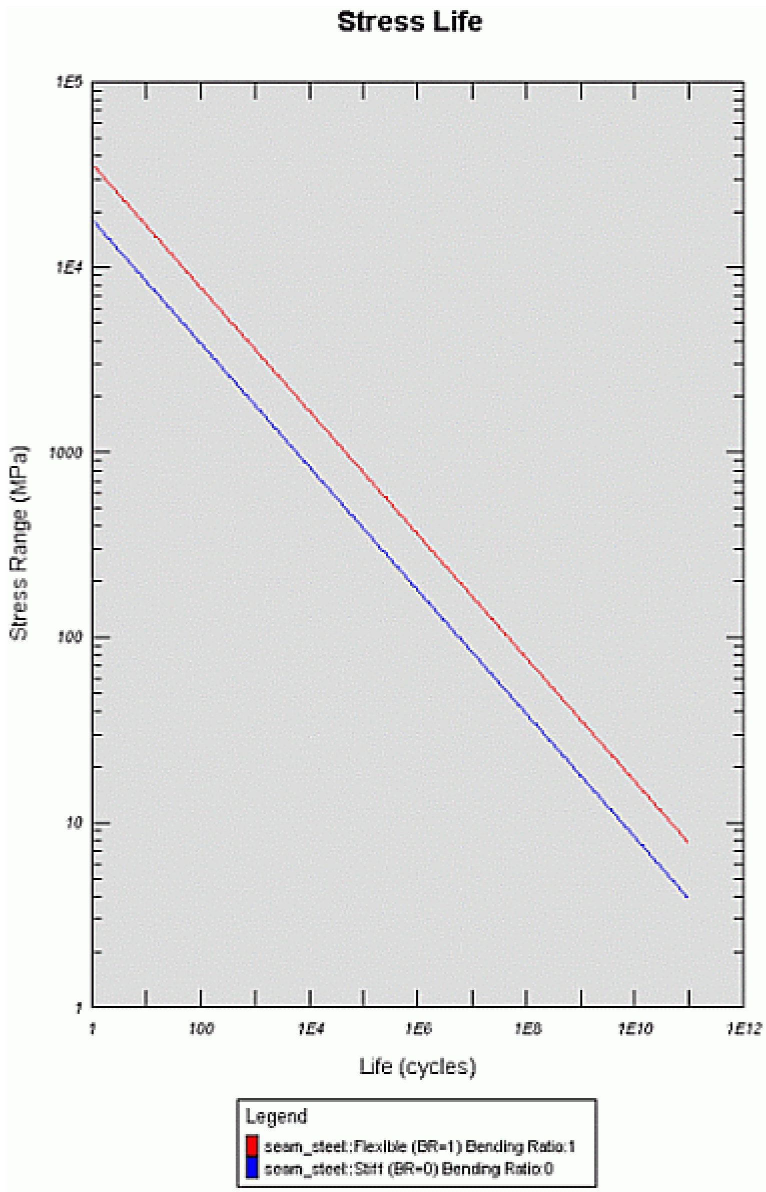
Parameter Name	Description
Material type	Numeric code defining the type of material
UTS	Ultimate tensile strength
E	Modulus of elasticity
M1	Mean stress sensitivity factors for FKM mean stress correction
M2	
M3	
M4	
TRef	Reference thickness/threshold for sheet thickness correction
n	Sheet thickness correction exponent
RR	R-ratio (ratio of minimum to maximum load) of the tests used to define the S-N curve
rth	Threshold value r_{th} of the bending (r) ratio, used in interpolation between “stiff” and “flexible” curves. If left blank, a value of 0.5 will be assumed.
Comments	
References	

In addition, each child dataset (there must be either one or two children) has the following additional properties:

Parameter Name	Description
BendingRatio	Bending or flex ratio of the curve. If two curves are present, BendingRatios must be 1 and 0 to identify “flex” and “stiff” curves.
SRI1	Stress range intercept
b1	First fatigue strength exponent
Nc1	Transition life
b2	Second fatigue strength exponent
SE	Standard error of $\log_{10}(N)$. This is used to calculate the life adjusted to a certain probability of failure/survival.
NFc	Numerical fatigue cutoff life. Beyond this life, damage will be assumed to be zero.

The formulation of the curve is the same as for the Standard S-N curve formulation. An example dataset is illustrated in Figure 5-160. Note that in this particular dataset, b1 and b2 are set to be the same, and that the curve should not be modified in the low-cycle regime to take into account the UTS. This can be ensured by setting the property CheckStaticFailure to False.

Fig. 5-160   Example S-N curve set for seam weld analysis



Some weld types may interpolate between the curves. The interpolation works as follows:

First, an interpolation factor is determined based on the mean bending ratio and the threshold bending ratio r_{th}:

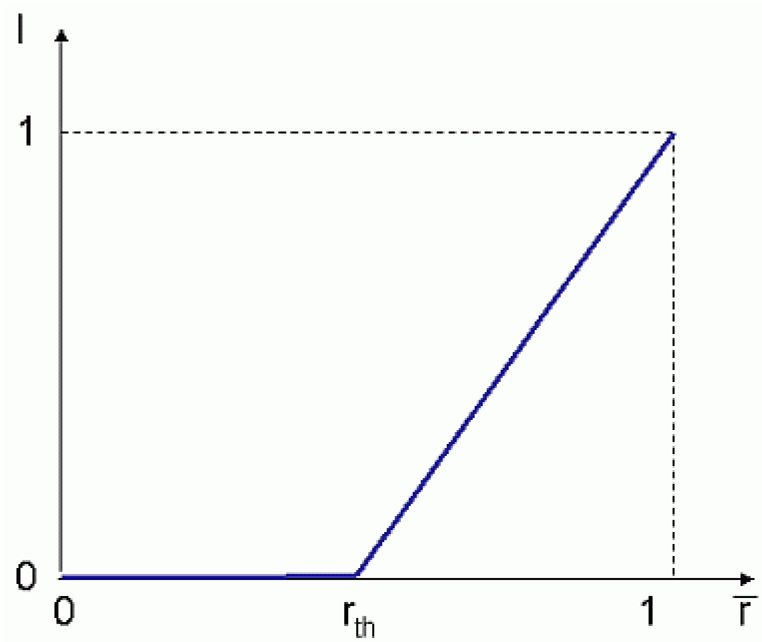
$$IF\ 0 \le \bar{r} \le r_{th} \quad I = 0$$

$$ELSE\ IF\ r_{th} < \bar{r} \le 1 \quad I = \frac{\bar{r} - r_{th}}{1 - r_{th}}$$

The default value of Bending Ratio Threshold r_{th} is 0.5, although the value can be changed in the materials data set. If no value for rth (r_{th}) is defined in the material database, this value will be assumed.

Graphically:

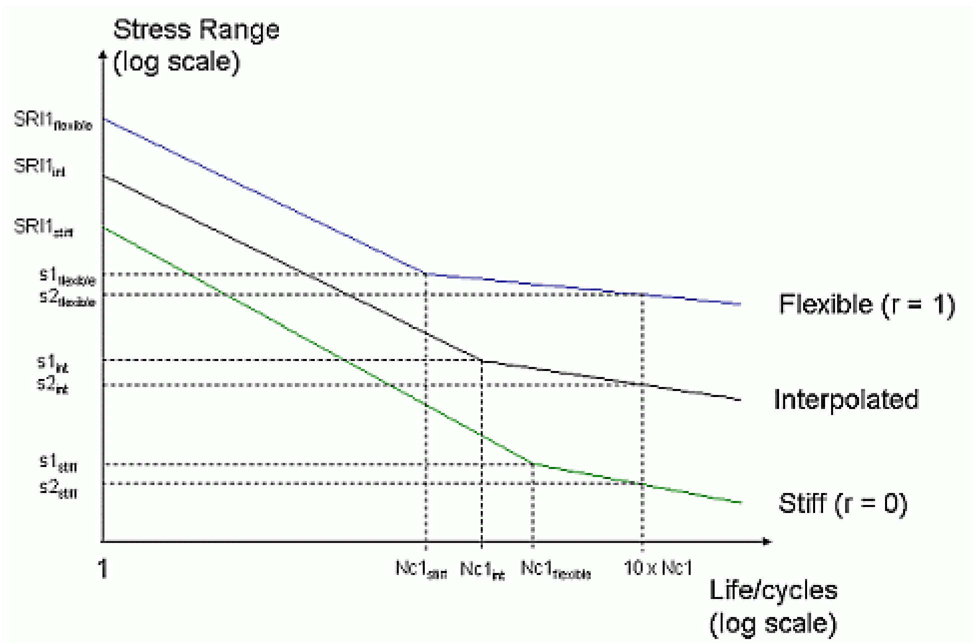
Fig. 5-161 Determination of interpolation factor



For values of the bending ratio up to the threshold, the “stiff” (r = 0) S-N curve will be used. Above that value, interpolation will be made between the “stiff” (r = 0) and “flexible” (r=1) curves.

The interpolation scheme is used to determine an interpolated set of parameters (SRI1, b1, Nc1, b2, SE), which will be used to predict fatigue damage. The scheme is illustrated in Figure 5-162 below.

Fig. 5-162 Interpolation between "stiff" and "bending" S-N curves



$$SRI1_{\text{int}} = SRI1_{stiff} + (SRI1_{flex} - SRI1_{stiff}) \cdot I$$

This defines the stress level at 1 cycle.

$$Nc1_{\text{int}} = 10^{(\log_{10} Nc1_{stiff} + (\log_{10} Nc1_{flexible} - \log_{10}(Nc1_{stiff}) \cdot I)}$$

$$s1_{\text{int}} = s1_{stiff} + (s1_{flexible} - s1_{stiff}) \cdot I$$

This defines the stress level at $Nc1_{int}$ cycles. These two points define the first section of the curve up to $Nc1_{in}$ cycles. The last section is defined by finding a third point as follows: A life value is defined, being 10 x the greater of the Nc1 values for the stiff and flexible curves. From these, we can calculate $s2_{flexible}$ and $s2_{stiff}$. From these, we can interpolate to get $s2_{int}$ which defines the high cycle part of the curve.

$$s2_{\text{int}} = s2_{stiff} + (s2_{flexible} - s2_{stiff}) \cdot I$$

Note that if only one S-N curve is present, it will be used directly without interpolation.

Thickness Correction

Optionally, a thickness (size effect correction) may be applied, based on the thickness t of each element. This works as follows:

If t ≤ T_{ref} (the reference thickness or threshold), there is no effect.

If t > T_{ref} the fatigue strength is reduced at all lifetimes by a factor $\left(\frac{T_{ref}}{t}\right)^n$ where n is the thickness exponent. (We use the factor to increase the stress.)

Fatigue Calculation

Once any interpolation and/or thickness correction has been carried out, the fatigue calculation proceeds as for a standard S-N calculation, based on the "weld top" stress. If a mean stress correction is to be applied, the FKM method is used. Multiaxial assessment may be carried out using the Standard or SimpleBiaxiality options. Auto mode is not available. These methods are all described in the section on the "Standard SN Analysis Engine" on page 84.

Note on Differences in Calculations for Different Weld Types

There are some differences between the way that different weld types are processed. For example, at release 5.0, interpolation between flexible and stiff S-N curves is applied only for fillet weld toes. For all other weld types and locations, the bending ratio r is set to 1.0 and only the "flexible" curve will be used.

The differences are summarized in Table 5-12 below:

Table 5-12 Calculation Method Differences Between Weld Types (Release 5.0)

Weld Type	Toe	Root	Throat	Interpolate	Notes
Fillet	✓	✓	×	✓	Interpolation is applied for the weld toe only. At the root, r is set to 1 and the "flexible" curve used.
Overlap	✓	✓	×	×	No interpolation. The bending ratio is set to 1 for all calculation points and the "flexible" curve is used.
Laser Overlap	N/A	✓	✓	×	
Laser Edge Overlap	✓	✓	✓	×	
Generic	✓	×	×	✓	Both surfaces of all elements attached to the weld elements are considered. All calculation points are handled as for fillet weld toes.

The software was made more flexible from release 5.1 as follows:

Table 5-13 Calculation Method Differences Between Weld Types (Release 5.1 onwards)

Weld Type	Toe	Root	Throat	Interpolate	Notes
Fillet	✓	✓	×	✓	Interpolation is allowed for all calculation points. The application (or not) of interpolation is determined by whether the material dataset has one or two curves.
Overlap	✓	✓	×	✓	
Laser Overlap	N/A	✓	✓	✓	
Laser Edge Overlap	✓	✓	✓	✓	
Generic	✓	×	×	✓	

The Seamweld analysis engine has the following properties.

5.8.9 CombinationMethod

AbsMaxPrincipal, **CriticalPlane** or **WeldNormal**. See above for a definition of what these mean in the context of a seam weld analysis.

5.8.10 MeanStressCorrection

FKM or **None**. The FKM mean stress correction is described in more detail in the section on the "Standard SN Analysis Engine" on page 84.

5.8.11 MultiaxialAssessment

Multiaxial assessment is carried out in the same way as for the Standard S-N analysis engine. The following options are available:

None

No multiaxial assessment is carried out.

SimpleBiaxiality

A simple biaxiality assessment is carried out. The method is described in more detail in the section on the "Standard SN Analysis Engine" on page 84.

Standard

A more sophisticated biaxiality assessment is carried out, as described in more detail in the section on the "Standard SN Analysis Engine" on page 84.

Note that since the grid point force method provides only stresses normal to the weld toe, there is little point in carrying out a biaxiality analysis in this case.

Note that Auto mode is not available for seam weld calculations.

5.8.12 CertaintyOfSurvival

If this is set to 50% (the default), the mean curve will be used. Otherwise, the life will be modified, taking into account the Standard Error of log(life) defined in the material dataset, in the same way as for the Standard S-N analysis engine.

5.8.13 ScaleFactor

This scale factor will be applied to all stresses before the fatigue calculation.

5.8.14 ThicknessCorrection

(**True/False**). See above.

5.8.15 BackCalcMode

Back calculation mode works in the same way as for the standard S-N analysis engine.

5.8.16 EventProcessing

This property controls the handling of duty cycles across events. In principle, and specifically as regards rainflow counting, duty cycles are handled in the same way as for the Standard S-N analysis engine.

Independent

Each unique event in the duty is calculated separately. Each event uses the average bending ratio calculated from that event only.

CombinedFull

The whole duty cycle is logically concatenated and processed as if it were one long event, including all repeats of all events in the correct sequence. The bending ratio values calculated (and used for interpolation) represent the whole duty cycle.

CombinedFast

See the section on the "Standard SN Analysis Engine" on page 84. Note that the average bending ratio used for the fatigue calculation (and the other bending ratio statistics) are determined over the whole of the duty cycle, taking into account the number of repeats of each event.

5.8.17 OutputEventResults

True

Results are output for each event that contributes to the duty cycle, in addition to the summary results for the whole duty cycle.

False

Only summary results for the whole duty cycle are written out.

5.8.18 CheckStaticFailure

This works in the same way as for the Standard S-N engine.

Please note the following:

This property should be set to False for seamweld calculations. If not, then in addition to a static failure check being made which is not validated, the low cycle end of the S-N curve will be modified using the UTS. This modification of the curve is not required for seamweld fatigue and will result in unrealistically large damage values in the low cycle regime.

5.8.19 Damage Floor

This is the calculated damage value below which the damage is set to zero. The default value is zero, i.e., damage is always reported as calculated.

5.8.20 MaxDamage

This is the maximum damage that can be reported for each cycle. If the calculated damage for a single cycle exceeds this value, it is set back to this value, unless this value is set to 0, in which case it is ignored.

5.8.21 StaticFailureDamage

Caution! Do not use this option. CheckStaticFailure should be set to False.

5.8.22 Calculation of Weld Hot Spot Stress from Solid Elements

In welded structures the structural stress at the weld toe, the hot-spot stress, can be estimated by the extrapolation of the surface stress at points near the weld. This method is called surface stress extrapolation (SSE). Various extrapolation methods have been proposed and a number of them are implemented here.

Where the stress distribution depends on the plate thickness (type 'a') the most common method is linear extrapolation from stresses on the surface at distances of 0.4t and 1.0t from the weld toe, where t is the plate thickness. This method is recommended for a fine FE mesh for a course mesh distances of 0.5t and 1.5t are used.

Where there is pronounced non-linear structural stress a quadratic extrapolation is possible using distances of 0.4t, 0.9t and 1.4t.

In the case of weld toes on plate edges (type 'b') the stress distribution approaching the weld does not depend on the plate thickness. Extrapolation methods therefore depend on absolute distances from the weld toe rather than proportions of the plate thickness. Two options are available for absolute distances; for a fine mesh a quadratic extrapolation is used with distances of 4, 8 and 12mm, for a course mesh a linear extrapolation is used with distances of 5 and 15mm.

In order to provide full control of the stress extrapolation the user can set *OffsetMethod* = *Custom*. In custom mode the user can define a comer separated list (*OffsetValues*) of either ratios or distances (mm) and these will be used to calculate the hotspot stress. The number of values required in the list is defined by the property *ExtrapolationPoints* (Two/Three) and the type is defined by *OffsetType* (Ratio/Distance).

In DesignLife the location of the extrapolation points is determined from the FE model using information from a weld definition file as shown in Fig. 5-163 Sample Weld Definition file. This file is selected via the *WeldDefinitionFilename* property on the FE import analysis group. This file contains a starting location, a vector to define the direction of the SCL, a vector to define the primary stress direction perpendicular to the SCL, and an optional *max weld depth* distance. The detail of this file format is described in File Formats.

The weld definition file is also used in the analysis of Solid Seam Welds in the Seam Weld analysis engine and is described in detail in 5.8.7 Calculation of Stresses from Solid Elements. Section 5.8.7 also defines how the plate thickness is determined at the weld location.

Fig. 5-163 Sample Weld Definition file

```
<WeldDefinitions>
<WeldLine name="Weld 1 - Root Top Sheet">
<WeldDef Location="-9.8, 0.78,  5.0" NormalVector="0.0, -1.0, 0.0" ToeVector="1.0, 0.0, 0.0"
id="11" MaxWeldDepth="1.4"/>
<WeldDef Location="-9.8, 0.78, 15.0" NormalVector="0.0, -1.0, 0.0" ToeVector="1.0, 0.0, 0.0"
id="12"/>
<WeldDef Location="-9.8, 0.78, 25.0" NormalVector="0.0, -1.0, 0.0" ToeVector="1.0, 0.0, 0.0"
id="13"/>
</WeldLine>

</WeldDefinitions >
```

Practical Considerations

Care must be taken when using this technique to calculate weld life. The surface stress results used in the extrapolation are obtained at remote from the weld and depending on the structure there may not be any FE results available.

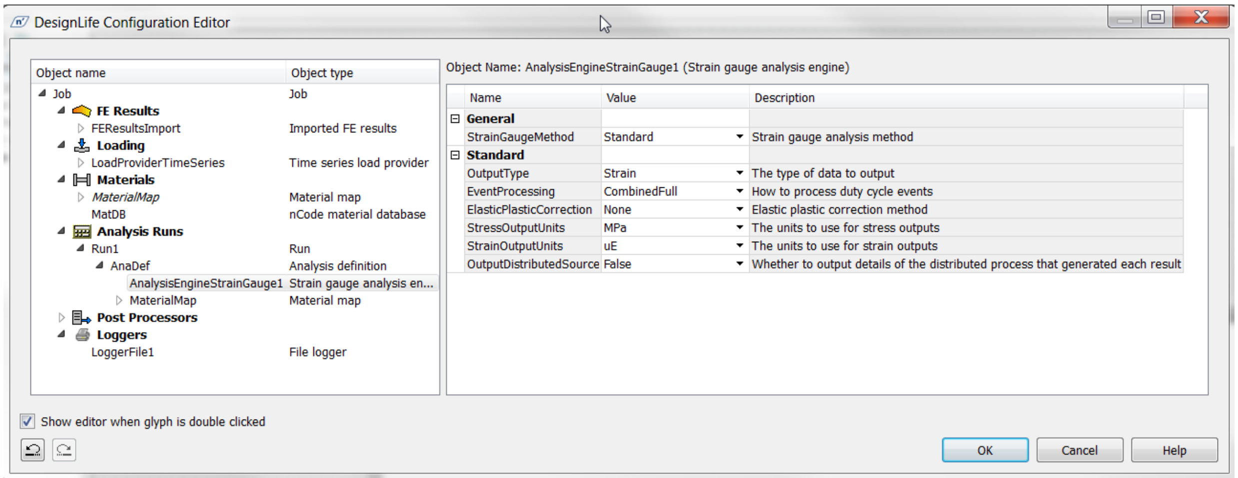
The software will search back along the vector *Vt* perpendicular to the weld toe and the up and down the vector *Vn* that defines the normal direction.

If a solid element can not be found on the search path for any of the extrapolation points then an error is given for the first analysis point that is found.

Currently the WeldHotSpot solution location can only be used in the SN Analysis engine.

5.9 Strain Gauge Analysis Engine

Fig. 5-164 Strain gauge analysis engine properties



The Strain Gauge Analysis engine has two purposes, the two strain gauge methods are:

- *Standard*: This allows the user to create strain (or stress) time histories from Virtual Strain Gauges. The strain histories produced are the direct strains resolved into the orientation of the gauge legs. They may therefore be suitable for correlation with actual strain gauges applied to the physical part being simulated.
- *StrainGaugePositioning*: This calculates optimal positions for strain gauges on the FE model in order for these to be used in a subsequent load reconstruction process. The strain gauge selection procedure works by selecting gauges that optimize the ability to invert the strain gauge response matrix reliably. The gauge selection procedure also has the capability of generating metrics that assist in selecting the appropriate number of gauges to use.

5.9.1 Standard - Prerequisites

When the *Strain Gauge Method* = *Standard* the Strain Gauge Analysis engine can work only on an *Analysis Group* for which the *SolutionLocation* = *StrainGauge*. See the section on the FE Import Analysis Group properties for details. Also, on the Analysis Group properties, EntityDataType must be set to one of the following:

- **Stress** - The software will be expecting elastic stresses.
- **Linear Strain** - The software will be expecting elastic strains.
- **StressAndStrain** - The software will be expecting elastic-plastic stresses and strains.

Clearly, some gauges will have to be available. These can be defined either graphically in the FE Input glyph (see the Studio User Guide) or from an external file.

Virtual gauges can be used in conjunction with any type of load provider, except a Vibration Load Provider or a duty cycle load provider containing Vibration Load Providers.

The virtual gauge will not actually produce any output unless a time series output object is attached to it ("Time series output to pipe" or "Time series output to file").

5.9.2 StrainGaugePositioning - Prerequisites

When the strain gauge method = StrainGaugePositioning the strain gauge analysis engine can work only on an Analysis Group for which the SolutionLocation = StrainGaugePositioning. Also, on the Analysis Group properties, Entity- DataType must be set to one of the following:

- **Stress** - the software will be expecting elastic stresses.
- **Linear Strain** - the software will be expecting elastic strains.

The analysis will use the average nodal results for all of the nodes to be analysed along with any optional virtual gauges that are defined. The gauges can be defined either graphically in the FE Input glyph (see the Studio User Guide) or from an external file.

Each time step should represent a single load input to the system. This is also true when using a Time Series input; in this case each point should represent a single load application. For most cases the Time Step input could be used, however, Time Series would be required if the stress caused by each individual load had to be calculated from a number of FE load cases.

For example if there is a load case representing the bolt up stress in an assembly, then the stress required for each load application could be achieved by a combination of the load cases and the appropriate time series data.

5.9.3 Properties and Functions of the Strain Gauge Analysis Engine

The Strain Gauge Analysis Engine has the following properties:

StrainGaugeMethod = **Standard** or **StrainGaugePositioning**. This selects the basic operation of the analysis engine.

- *Standard* will produce stress (or strain) time histories from the defined virtual stain gauges.
- *StrainGaugePositioning* will calculate optimum strain gauge positions for the model and the applied loads.

Properties for the *Standard* method

Output type = Stress, Strain or StressAndStrain. In each case, the strain or stress output is the direct stress and/or strain resolved into the orientation of the gauge leg. In the case of a gauge placed at a node, the output is based on the averaged FE results. If the node in question is attached to both shell and solid elements, FE results will be averaged from the attached shell elements only. In any case, FE results are transformed to a common coordinate system if necessary before averaging takes place, and subsequently transformed to a gauge coordinate system. This gauge coordinate system has the orientation of gauge leg 1 as the local x-axis and the outward surface normal as the local z-axis.

EventProcessing = Independent or CombinedFull. This property controls the handling of multiple events when a duty cycle load provider is used. If Independent mode is used, a separate time history will be created corresponding to each event in the duty cycle. If CombinedFull mode is used, the result will be long time histories representing the duty cycle in its entirety, including every repeat of every event in the correct sequence.

ElasticPlasticCorrection = None. Elastic Plastic correction has not been implemented yet in the Strain Gauge Analysis engine. When it is necessary to calculate stresses from strains or vice versa, elastic behavior will be assumed and the transformation made using Hooke's law (see page 162 for details).

StressOutputUnits and **StrainOutputUnits** are self-explanatory.

Various combinations of input and output data are possible for the engine. In each case, the output is always the direct stress or strain resolved in the orientation of each gauge leg.

FE EntityDataType	Output	Comments
Stress	Stress	Elastic stress used directly
	Strain	Strain calculated from stress (Hooke's law)
LinearStrain	Stress	Stress calculated from stress (Hooke's law)
	Strain	Strain used directly
StressAndStrain	Stress	Stress used directly
	Strain	Strain used directly

Properties for the *StrainGaugePositioning* method

MaximumGauges: This is the maximum number of strain gauges to output from the analysis. The minimum number is the number of time steps (or time series points), and this defines the minimum value for this property. There is no default value.

IncludeInputGauges = True/False. If this = *True* then any virtual strain gauges defined in the analysis will be used, and they will always be present in the output. The analysis will then determine the best additional gauges to add to these. The default value is True.

MinimumSignal: Default=20 (microstrain).

MaximumSignal: Default=3000 (microstrain).

The combination of *MinimumSignal* and *MaximumSignal* is used to eliminate gauges whose signals are too low for every load condition. The purpose of this step is to reduce the size of the optimisation problem by removing areas of low signal that would not be useful as strain gauge positions.

For each load the strain values for each entity are scaled by the ratio *MaximumSignal/Maximum Strain*, this will result in the maximum strain value for each load being equal to *MaximumSignal*. Any entities where the strain is less than *MinimumSignal* for all loads are then removed from the analysis.

OutputFilename: Default = candidate_set. This is the filename for the output strain gauge definition files (.ASG) that are created for each analysed number of gauges. The files created will be, for example: *candidate_set_7.asg*, where the number is the number of gauges in that file.

Overwrite: Default = True. This defines if any .ASG files that already exist will be overwritten.

5.9.4 Strain Gauge Positioning Background

The measurement of loads on a structure is a vital part of the design process; this is often done by the insertion of a load cell into the system. If this is not possible it is often necessary to utilise a part of the structure itself as the load transducer and this can done by strain gauging the component in appropriate positions. The precision to which the loads can be estimated is determined by the number and placement of the strain gauges.

The purpose of the strain gauge positioning method is to determine the optimum positions for the strain gauges on the component in order that the measured strains will give the best estimate of the input loads. The methodologies used are as defined in References 1-3.

The overall process is:

- Take the stress (or linear strain) results from the FE (linear/elastic) model of the structure with one load case per applied load. The average node on element result will be used in the analysis.
- Determine which entitles (nodes) to use in the analysis
  - Take all nodes defined for the analysis
  - Remove nodes where the feature angle is greater than AllowableFeatureAngle (defined on the Analysis Group)
  - Remove nodes where the radius of curvature is less than MinimumRadiusOfCurvature (defined on the Analysis Group)
  - Use any defined virtual strain gauges if IncludeInputGauges is True.
  - Use the MinimumSignal/MaximumSignal properties to remove entities with low strains.

- Pass the matrix of strain values for each entity and load case to the optimisation routine for each required number of gauges.

Note The optimisation routine used is described in References 4. It is an open source routine and has been included as a separate executable (sgopt.exe).

- Write out an .ASG, strain gauge definition file for each number of gauges and also a set of optional metadata from the calculation (add a pipe or file post processor to the analysis). The parameter Variation Inflation Factor is a measure of the matrix co-linearity and is a check on the selected result matrices.

5.9.5 Output Metadata

There is a set of multi-column data output when the strain gauge positioning is run and these results show the metrics quantifying the quality of the chosen gauge locations for the corresponding number of gauges. Each row represents the results for a particular number of gauges which is output in column 1. Also displayed is the determinant of At.A, the variance of each load, and the variance inflation factor (VIF) of each load.

- Determinant of At.A (DETAtA); this is the determinant of the matrix returned from the dot product of the input matrix and its transpose (At.A). The determinate is the property we are maximizing, so it should be increasing with the number of gages. Looking at its rate of increase can give some idea on benefit of increasing the number of gages used.
- Variance of LoadX (VarOfLoadX); these are the variances of the load estimates and they should be decreasing with additional gages. Again a drop off in the rate of decrease can be used as a guide of how many gages to use.
- VIF of Load X (VIFOfLoadX); the VIFs (variation inflation factors) are a measure of how collinear the stresses from one loadcase are with stresses from other loadcases. An ideal value of VIF is 1 (meaning the stress state is completely independent). If one loadcase has a stress state that is exactly some linear combination of stress states from other load cases, then their VIF values would be infinity. Values exceeding 10 are indications of problems, and one should examine the load cases involved to ensure they are not actually representing the same load condition. An example of this is when we think we have a condition where a positive and a negative load have different load paths, but in reality they are very similar. The VIF values for these two load cases would be very high.

The variance and VIF columns should be graphed against the number of gauges to help determine if the selected number of gauges to use is sufficient to get a good estimate of the loads when they are used for load regeneration.

5.9.6 References

1. Wickham, M. J., Galliart, D. R., Nachtshelm, C. J. and Riley, D. R., 1994, "The Design and Application of a Multi-Axis Load Transducer.", SAE paper 940250.

2. Wickham, M. J., 1992, "Optimal Experimental Design for the Instrumentation of a Multi-Axis Load Transducer", Ph.D. Thesis, University of Minnesota, Minneapolis, MN.

3. Wickham, M. J., Riley, E. R. and Nachtsheim, C. J., 1992, "The Optimul Design of a Multi-Axis Load Transducer", Proceedings of the 1992 ASME International Computers in Engineering Conference and Exposition, G. A. Gabriele, ed., ASME, New York, Vol. 1, pp. 175-179.

4. Wheeler, B., 2011, "AlgDesign, Algorithmic Experimental Design", h. www.rdocumentation.org/packages/AlgDesign/versions/1.1-7.3/topics/optFederov

5. Hoffman, K., 1989, "An Introduction to Measurements using Strain Gages.", Hottinger Baldwin Messtechnik, GmbH, Darmstadt.

5.10 Adhesive Bond Analysis Engine

Fig. 5-165 Adhesive Bond analysis engine properties

Object Name: AdhesiveBondEngine (Adhesive bond analysis engine)

Name	Value	Description
General		
ScaleFactor	1	The scale factor to apply prior to damage calculation
Output		
OutputSafetyFactor	True	Whether to output the safety factor
OutputEventResults	False	Whether to output results per event or not for duty cycle processing

5.10.1 Introduction

The adhesive bond analysis method implemented in this engine is based on work carried out at Volvo Technology, and is sometimes known as the "LIMFOG" method (named after the original in-house software, based on work by Gunnar Bjorkmann). The method was developed specifically for automotive components—in particular car and truck body structures—joined using adhesive bonds. The method is suitable for application to bonded joints between two sheets of steel or aluminium, of thickness in the range approximately 0.5 to 3 mm.

The LIMFOG method proceeds using the following steps:

1. A global finite element analysis of the structure is carried out. A simple, relatively coarse representation of the joints is used, in which beam elements represent the adhesive joining the flanges.
2. Line forces and moments along the edges of the flanges are derived from the FE results, combined with applied loading histories.
3. The line forces and moments are applied to an analytical "sandwich" model of the bonded flange, and used to calculate the strain energy release rate G (actually the J-integral) based on an initial small crack in the adhesive.
4. At each point along the edge of the flange, the calculated maximum J-integral during the loading history is compared to the threshold required for crack growth G_{th} to determine whether or not the joint is likely to fail.
5. The method does not calculate fatigue life, but in addition to the maximum value of the J-integral for a given flaw size, a safety factor may be calculated.

LIMFOG has two forms: congruent mesh and mesh independent versions. The mesh independent version uses ACM-type connections, and allows the parts to be meshed independently. However, as with spot welds, the results are not so reliable and for this reason only the congruent mesh version has been implemented in nCodeDT.

The following sections describe the essential features of the method in more detail.

5.10.2 Modeling Guidelines, Grouping and Required FE Results

The user must adhere to the following modeling guidelines/requirements:

- Models will be created using linear shell elements.
- Flanges require congruent meshes, typically two elements across the flanges, which should be parallel.
- Two flanges being joined will normally be in different property sets.
- There is no requirement that normals on the flanges being joined point in any particular direction, but within each shell component being joined, normals should be aligned.
- Four noded quads are strongly preferred everywhere, especially at the corners, where it is important to use square elements close to 10 mm in size.
- Spacing of shells at the flanges should ideally be $\frac{h_1 + h_2}{2} + h_a$ where h1, h2, ha are the thicknesses of the two sheets and the adhesive layer.
- The adhesive bond should be modeled by beam elements around the periphery of the bond, not in the middle of the bond.
- The beam properties should correspond to a 5 mm circular cross section (for NASTRAN CBARs, A=19.6 mm^2, Ix=Iy=30.7 mm^4 and J=61.4 mm^4). Elastic modulus E = 3000 MPa and Poisson ratio 0.4 regardless of adhesive properties.
- Within a bond line, the beam elements should be aligned (i.e., all start and all end on the same shell property sets).
- Beam elements should be normal to the flanges, and with the correct length.
- All the elements (shells and beams) associated with a bond must be put into an element SET. More than one bond line can be in the same set, so long as the adhered material properties, adhesive properties and bond thickness are the same, together with other properties that will be defined in DesignLife on a material group-by-group basis, such as offset and initial crack depth.

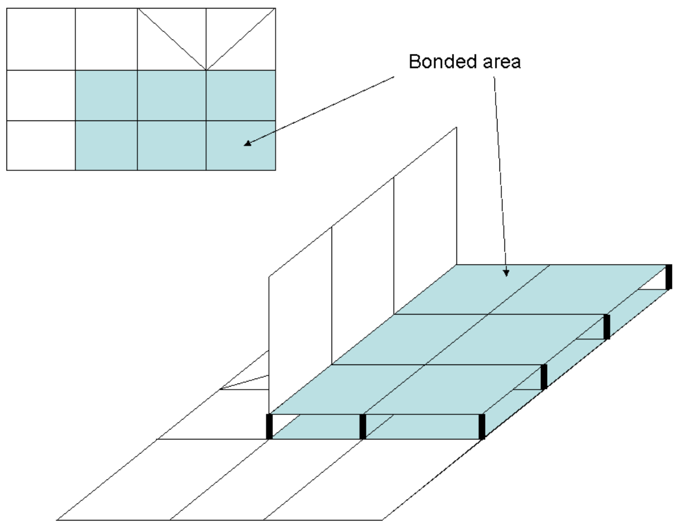
In the FE analysis, grid point forces must be calculated (GPFORCES in NASTRAN, NFORC in ABAQUS) and available for all the shell elements adjacent to the shell elements that form part of the bond set.

In addition, sheet thicknesses must be available in the FE results. These are normally present in NASTRAN op2 files, but for ABAQUS they must be explicitly requested as outputs in the input deck (STH).

An example model is illustrated in Figure 5-166. The bonded area is represented by the shaded elements. Note the bar elements around the periphery of the bond. These elements (shells and bars) must be included in an element set which will be used to define the Analysis Group.

Note Due to a change in default settings, later versions of MSC.NASTRAN may produce op2 files that do not contain the necessary element set information, even though the element sets are correctly defined in the NASTRAN input deck.
The problem can be avoided by setting the POSTEXT parameter (include the line PARAM, POSTEXT,YES in the input deck).
Alternatively, if the input deck is dropped onto the FE input glyph before the corresponding op2 file, the element set information will also be available.

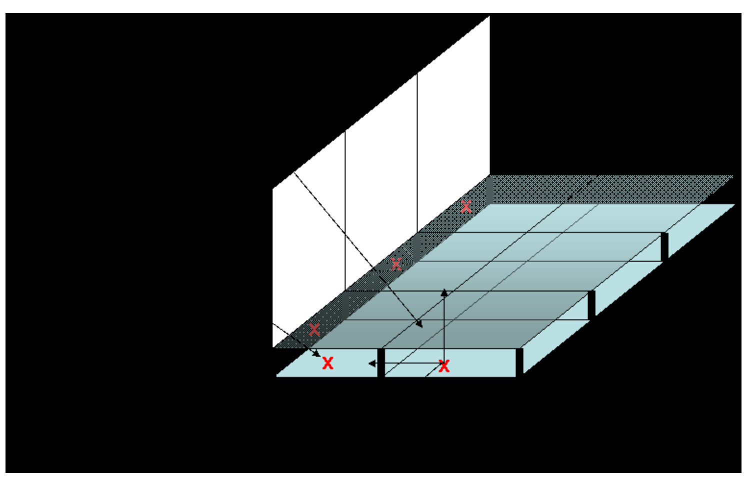
Fig. 5-166 An example model



5.10.3 Calculation Points and FE Results Translation

The calculation is made at points along the edge of the bonded area, corresponding to the mid points of the edges of the shell elements, as illustrated in Figure 5-167:

Fig. 5-167 Calculation points and local coordinate systems

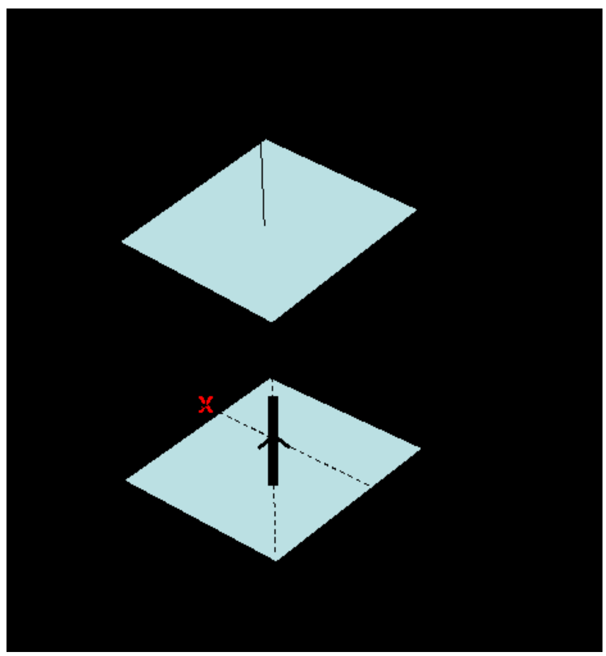


Each bond has a “top” and a “bottom” layer, defined by the orientation of the attached beam elements. Note the modeling guideline that all the beam elements should point in the same direction. Beam elements go from “bottom” to “top”.

The calculation points are highlighted with a red “x” in Figure 5-167 above. Note that unloaded edges (no attached shell elements) are not calculated.

The FE translation derives the line forces and moments (forces and moments per unit length) applied to the bond by the attached shell elements (i.e., the shell elements adjacent to the bond. These forces are translated to a local coordinate system denoted (ξ_1, ξ_2, ξ_3). The ξ_1 axis is an outward normal to the edge of the bond, the ξ_2 axis is normal to the shell elements and in the same sense as the beam orientations, and the ξ_3 axis lies along the edge. See Figure 5-168 below:

Fig. 5-168 Local coordinate system definition

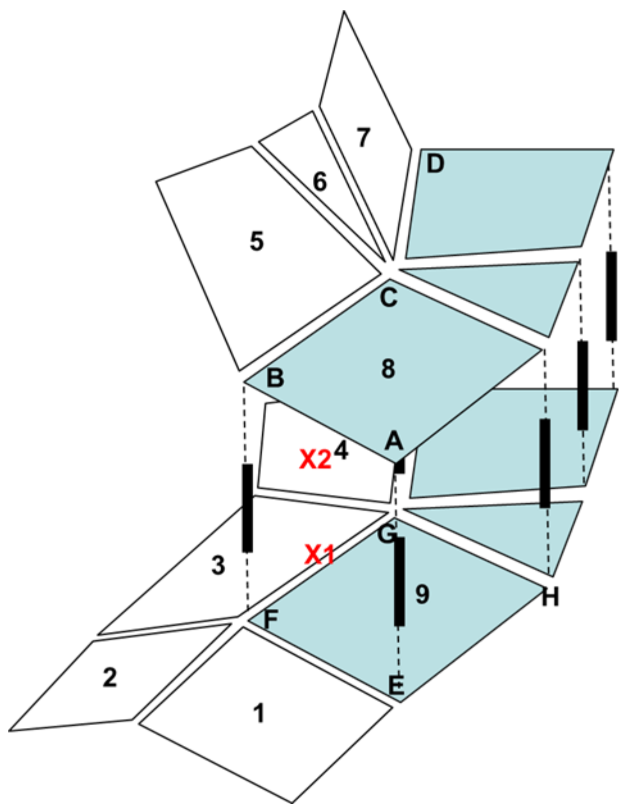


The line forces and moments are calculated as follows:

1. Calculate the forces applied to the edge of the bonds at each node by the adjacent shell elements.
2. Share these forces out in proportion to the lengths of the loaded edges.
3. Calculate the forces and moments per unit length.
4. Translate them to the local coordinate system.

The process is detailed below with reference to the expanded view of a joint in Figure 5-169.

Fig. 5-169 Bonded joint—expanded view



- At each calculation point we have two edges. For example, at point X1 we have the edge AB on the top element 8 and edge EF on the bottom element 9.
- Edge AB on element 8 has no attached elements outside the bond line, so the total forces and moments on this edge $F_{AB} = (F_{x,y,z}, M_{x,y,z})$ are zero, and so are the corresponding line forces and moments.
- The corresponding edge EF on the bottom does have an attached element outside the bond (element 1). For this edge, we need to determine the total forces and moments applied by elements outside the bond line on the two nodes E and F, and then share the forces out in proportion to the length of the loaded edges.
  - At node E the force comes from element 1 alone. There is no other edge than EF to share the force with because edge EH is unloaded. Call this F_E.
  - At node F the force is the total of the forces from elements 1, 2 and 3, and if this total force is F_{tot} then the share of this force F_F seen by the edge at node F is $F_F = F_{tot} * EF/(EF + FG)$
  - Now the total force on the edge is $F_{EF} = F_E + F_F$

- Divide this by the edge length EF and translate to the local coordinate system, and that gives us the required line forces on this edge.

For each calculation point, the following information is written to the intermediate (FEI) file.

```
Calculation point Index
Top shell element ID
Top shell edge node ID1
Top shell edge node ID2
Top shell thickness
Bottom shell element ID
Bottom shell edge node ID1
Bottom shell edge node ID2
Bottom shell thickness:
Line forces and moments in local coordinates:  This will be 12
components of line force and moment per load case or load step,
for top and bottom shells (2 sets of 6 components).
```

5.10.4 Material Properties

The Adhesive Bond analysis engine requires that materials are assigned for the top and bottom sheets and for the adhesive, using the material map interface. Only the adhesive material data needs to be of type "Adhesive Bond". The sheets require only the elastic modulus and Poisson ratio to be defined, and can therefore use any type of data for which these values are defined.

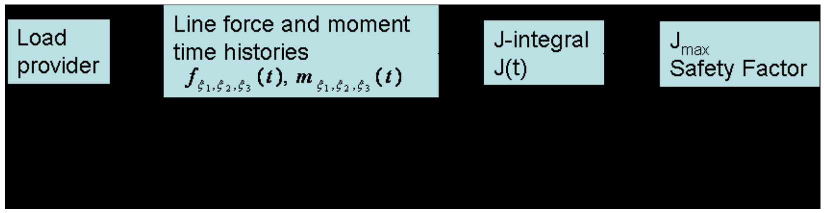
The Adhesive Bond data is as follows:

Parameter Name	Description
Material Type	Numeric code defining the type of material (not required)
E	Modulus of elasticity
me	Elastic Poisson ratio
Gth	Threshold strain energy release rate. If the strain energy release rate remains below this level, fatigue crack growth is not predicted to occur.
Comments	
References	

5.10.5 Calculation Method

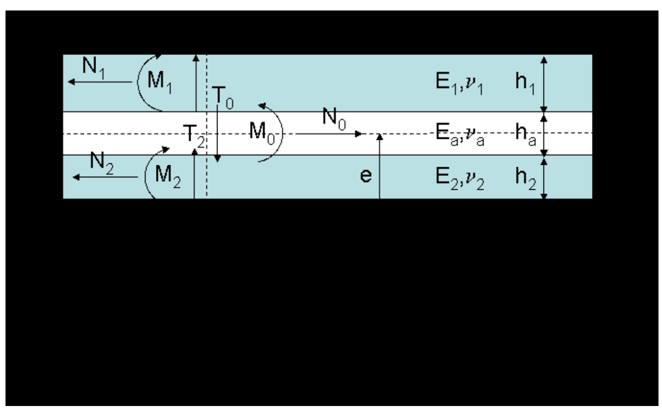
The calculation process is summarized in Figure 5-170 below:

Fig. 5-170 Adhesive bond calculation process



The basic premise of the method is that the strain energy release rate can be determined by transferring the line forces and moments along the edge of an adhesive bond to a "sandwich" model. For the purely elastic case, the strain energy release rate is equal to the J-integral, and it is the J-integral that is calculated by the software. It is evaluated along a path just before and just after the crack front. In Figure 5-171 below, a is the effective total crack depth relative to the beam elements in the shell model.

Fig. 5-171 Sandwich model schematic



The inputs to the analysis are:

$(f_{\xi_1,\xi_2,\xi_3}, m_{\xi_1,\xi_2,\xi_3})_1$: the line forces and moments on the top shell

$(f_{\xi_1,\xi_2,\xi_3}, m_{\xi_1,\xi_2,\xi_3})_2$: the line forces and moments on the bottom shell

h_1: the thickness of the top shell

h_2: the thickness of the bottom shell

h_a: the thickness of the adhesive layer

E_1: Modulus of sheet 1 (upper sheet)

ν_1: Poisson ratio of sheet 1

E_2: Modulus of sheet 2 (lower sheet)

ν_2: Poisson ratio of sheet 2

E_a: Modulus of adhesive

ν_a: Poisson ratio of adhesive

b: the offset from the beam to the edge of the adhesive

a_0: the initial flaw size

The total strain energy release rate:

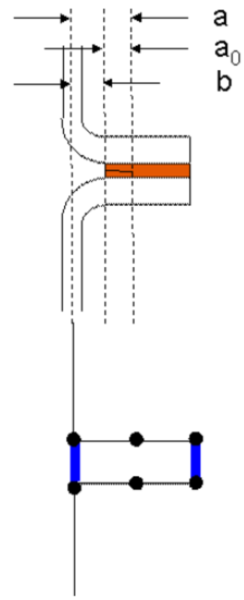
$G = J = J_1 + J_2 - J_0$ where the indices 1,2,0 represent the three sections of the J-integral path which have non-zero contributions to the J integral, corresponding to the top sheet, the bottom sheet just behind the crack front, and the sandwich just ahead of the crack front.

First the forces and moments are calculated at the crack front.

The crack front is offset from the beam elements by:

$a = b + a_0$ where b is the Bond Line Offset and a_0 is the initial crack size.

Fig. 5-172 Definition of bond offset b and initial crack size a_0



Top sheet (surface 1):

$$N_1 = f_{\xi_1}^{\ top}$$

$$T_1 = f_{\xi_2}^{\ top}$$

$$M_1 = m_{\xi_3}^{\ top} + a \cdot f_{\xi_2}^{\ top}$$

$$F_{z1} = f_{\xi_3}^{\ top}$$

Bottom sheet (surface 2):

$$N_2 = f_{\xi_1}^{\ bot}$$

$$T_2 = f_{\xi_2}^{\ bot}$$

$$M_2 = m_{\xi_3}^{\ bot} + a \cdot f_{\xi_2}^{\ bot}$$

$$F_{z2} = f_{\xi_3}^{\ bot}$$

Then we need to define some moduli, effective thicknesses and other intermediate parameters. The indices 1 and 2 denote the parts of the J-integral integration path corresponding to the top and bottom sheets, and the index 0 denotes the sandwich.

$$E_0 = E_1$$

The effective thickness of the sandwich...

$$h_0 = h_1 + \frac{E_a}{E_1} h_a + \frac{E_2}{E_1} h_2$$

Shear moduli: $\overline{G_i} = \dfrac{E_i}{2(1-\nu_i)}$ and $Q_i = \dfrac{E_i}{1-{\nu_i}^2}$ for i = 1,2

$G_0 = G_1$ and $Q_0 = Q_1$

$$e = \frac{\left(\frac{h_2}{2}\right)\left(\frac{Q_2}{Q_1}\right)h_2 + \left(h_2 + \frac{h_a}{2}\right)\left(\frac{Q_a}{Q_1}\right)h_a + \left(h_2 + h_a + \frac{h_1}{2}\right)h_1}{\left(\left(\frac{Q_2}{Q_1}\right)h_2 + \left(\frac{Q_a}{Q_1}\right)h_a + h_1\right)}$$

(This is the distance from the bottom to the centroid of the sandwich.)

Second moment of area...

$H_i = \dfrac{{h_i}^3}{12}$ for i = 1,2

$$H_0 = \left(\frac{\left(\frac{Q_2}{Q_1}\cdot h_2\right)^3}{12} + \left(\frac{Q_2}{Q_1}\cdot h_2\right)\left(e - \frac{h_2}{2}\right)^2\right) + \left(\frac{\left(\frac{Q_a}{Q_1}\cdot h_a\right)^3}{12} + \left(\frac{Q_a}{Q_1}\cdot h_a\right)\left(e - h_2 - \frac{h_a}{2}\right)^2\right) + \left(\frac{{h_1}^3}{12} + h_1 \cdot \left(e - h_2 - h_a - \frac{h_1}{2}\right)^2\right)$$

The distortion of the beam elements makes a significant contribution to the strain energy release, and hence the J-integral. The rotation of the beam cross sections is estimated from:

$\phi_i = K_i^M \cdot M_i + K_i^T \cdot T_i$ for i=1,2

$$\phi_0 = 0$$

where the slope constants K are functions of the sheet and adhesive thicknesses and moduli.

$K_i^M = K_i^M\left(h_i, E_i, h_a, E_a\right)$ and $K_i^T = K_i^T\left(h_i, E_i, h_a, E_a\right)$ for i=1,2

$$\phi_0 = 0$$

Then the forces and moments for surface 0 can be deduced by equilibrium.

$$N_0 = N_1 + N_2$$

$$T_0 = T_1 + T_2$$

$$F_{z0} = F_{z1} + F_{z2}$$

$$M_0 = M_1 + N_1\left(e - h_2 - h_a - \frac{h_1}{2}\right) + M_2 + N_2\left(e - \frac{h_2}{2}\right)$$

Now we can calculate the J-integral:

$$J_i = \frac{N_i^{\,2}}{2E_i h_i} + \frac{1.2 \cdot T_i^2}{2\overline{G}_i h_i} + \frac{F_{zi}^{\,2}}{2\overline{G}_i h_i} + \frac{M_i^{\,2}}{2Q_i H_i} + T_i \phi_i$$

...and finally:

$$J_{tot} = J_1 + J_2 - J_0$$

The Adhesive Bond analysis engine calculates the J-integral for each point during the loading history, and the maximum value $J_{\max}$ is written to the results. In addition, a time series of the J-integral may be generated by attaching a time-series output object to the analysis engine.

A SafetyFactor, based on the ratio of $J_{\max}$ and the threshold strain energy release rate G_{th} may also be calculated. Since J and G scale as the square of the applied load, the safety factor has been defined as follows:

$$SafetyFactor = \sqrt{\frac{G_{th}}{J_{\max}}}$$

For post processing, the results are written to the top and bottom shell elements associated with each calculation point. The same results are written to top and bottom shells.

Event handling for a duty cycle is simple. There are no options. The maximum value of $J_{\max}$ for a duty cycle is the maximum of the values from the individual events. Events are always calculated separately and the repeat count is ignored.

The minimum value of safety factor for a duty cycle will be the minimum of the values from the individual events.

The Adhesive Bond analysis engine has the following properties.

ScaleFactor

This defines a scale factor to be applied to the loading prior to the J-integral calculation. Default value = 1.0

OutputSafetyFactor

True/False. This property defines whether or not the safety factor will be calculated.

OutputEventResults

True/False. This property defines whether the results for individual events within a duty cycle will be reported in the results, in addition to the overall result for ALL events, which is always reported.

5.11 Creep Analysis Engine

Fig. 5-173 Creep analysis engine properties

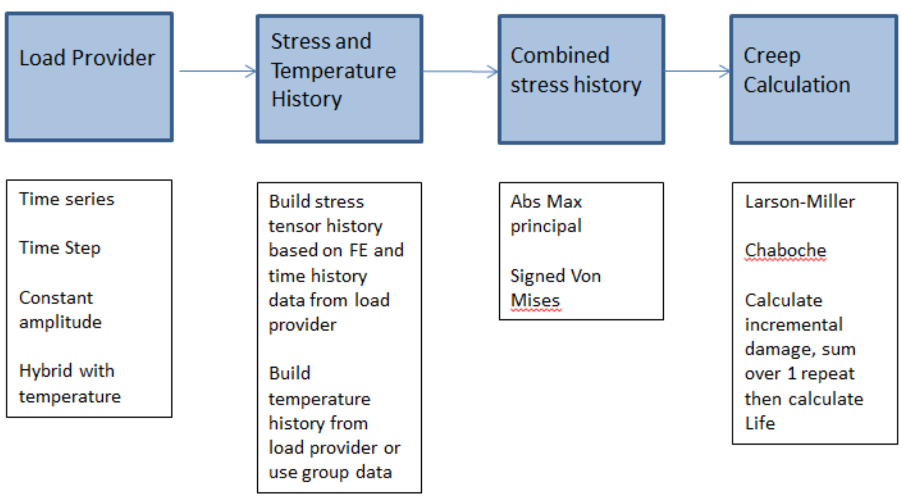
Object Name: CreepEngine (Creep analysis engine)

Name	Value	Description
General		
CreepMethod	LarsonMillerPolynomial	The method used to calculate creep
CombinationMethod	AbsMaxPrincipal	The stress combination method
ScaleFactor	1	The scale factor to apply to the combined data prior to performing damage calculation
TemperatureSelection	TimeSeries	How to use temperature data from the load provider
DutyCycle		
OutputEventResults	False	Whether to output per event results or not in duty cycle mode
Advanced		
StaticFailureDamage	1E10	Damage value to output in the case of max stress or max temperature exceeded

5.11.1 Summary

The basic operation of the creep analysis engine is as follows:

Fig. 5-174 Creep analysis engine workflow



A typical creep analysis has the following basic steps, which are executed for each analysis entity (Node, Element, Node-on-Element) and shell surface.

1. The stress tensor history, as a function of time or data points, is assembled from the information provided in the load provider. This applies to time-series, time step, constant amplitude and hybrid load providers. See the section on Loading for details. If the temperatures are coming from the FE results, then a hybrid load provider must be used, and must include a single temperature load provider. In addition, the FixedSampleRate option must be used to generate the correct time

information for the creep analysis. Duty-cycle load providers are addressed later.

2. From the stress tensor, extract a combined stress parameter (Abs Max Principal or Signed von Mises).
3. Calculate incremental creep damage using either the Larson-Miller or Chaboche method point by point through the time history. Sum the incremental damages to calculate a total damage for 1 repeat.
4. Calculate the life in repeats and life in hours from the repeat damage. Note that Chaboche uses a non-linear summation formulation at this stage.

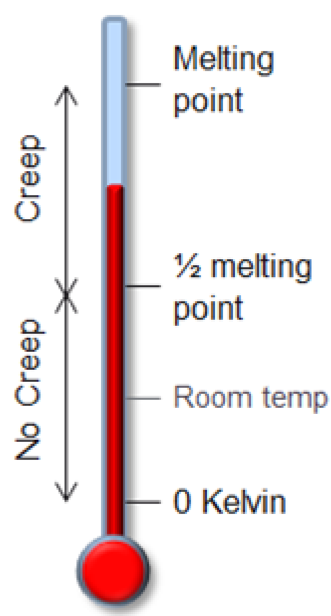
Because time information is required, load providers such as the Vibration Load Provider that generate cycles cannot be used.

There is no rainflow counting in creep analysis; damage can occur due to creep even if the stress is constant, provided that the temperature exceeds the creep threshold.

5.11.2 Creep—Background

Creep is a damage mechanism in materials that occurs when a material is subjected to a constant stress at temperatures close to the melting point. The level at which this occurs is material dependent and often begins at around half the melting point (in Kelvin).

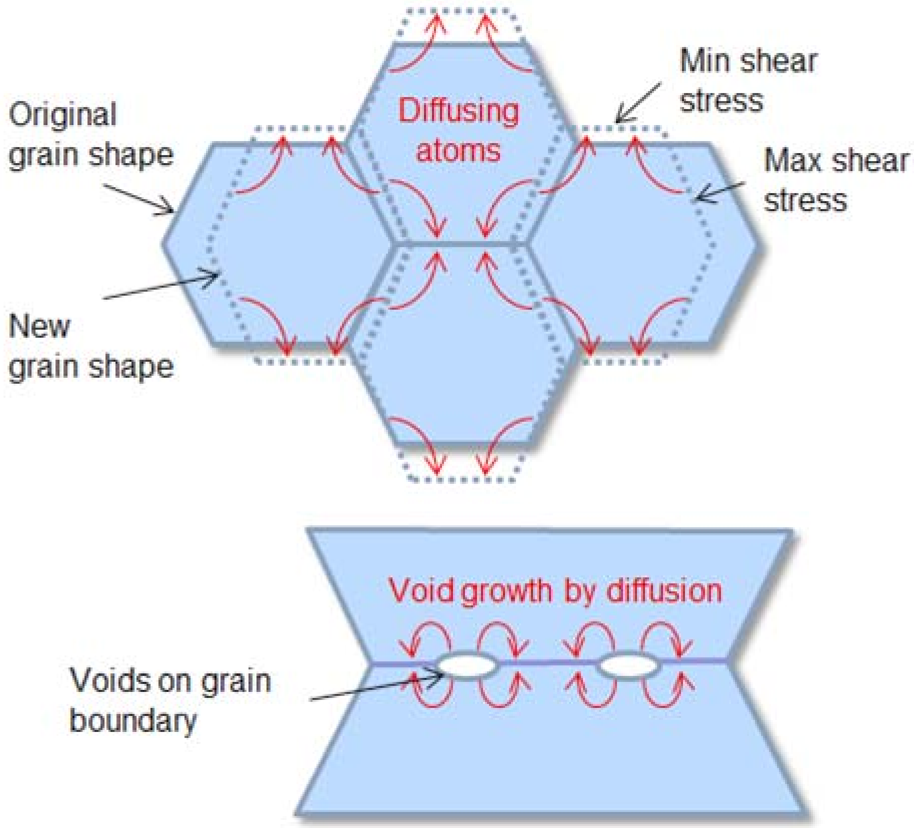
Fig. 5-175 The creep zone



Unlike fatigue, the stress does not need to be "cycled" for creep damage to occur.

As temperature increases, the atoms within the material are excited and diffuse from the stressed face to the unstressed face, causing microscopic gaps to occur that gradually, over time, grow larger.

Fig. 5-176 Creep mechanism - diffusion and growth



The rate of deformation within the structure is a function of the material properties, exposure time, exposure temperature and the applied structural load. Creep is usually of concern to engineers and metallurgists when evaluating components that operate under high stresses or high temperatures. Eventually creep can cause the material to fail, or "rupture".

Several models have been proposed that estimate the time to rupture based on a constant temperature and stress. Two of these methods are implemented in DesignLife: Larson-Miller and Chaboche. The starting point for these calculations is the materials data. To obtain this data, creep tests must be performed. A set of creep tests consists of taking a specimen of known geometry, hanging a weight from the specimen and heating the specimen to a known, constant temperature.

Eventually the specimen will rupture and fail. The time to rupture is recorded and so a table of values for stress, temperature and time to rupture is obtained. For the different methods, these test results are used in different ways to obtain the curves or equations that describe the materials behavior.

If either or both of the temperature and stress are varying with time, it is possible to calculate partial creep damage values for small increments of time and then sum them together to calculate damage against time. When the total damage reaches 1, rupture is estimated to have occurred.

In the Larson-Miller method, damage is summed linearly, whereas Chaboche includes a non-linear damage summation model.

Using Robinson's Rule, creep damage can be summed with fatigue damage to get a total damage. This assumes that the interaction between creep and fatigue modes is linear.

CreepMethod

The CreepMethod property defines the model to be used for creep calculations. The way the stress and temperature data is provided to the two models is the same.

For Larson-Miller, two materials data formulations are used, described as Larson-Miller Polynomial and Larson-Miller Curve. They perform the same function, but one describes the relationship between stress and the Larson-Miller parameter as a polynomial (up to 4th order) and the other uses a series of X-Y points to achieve the same goal.

The Chaboche method is separate and uses a different equation and method.

The properties that are required in the Materials database are as follows:

Larson-Miller Curve

The Larson-Miller Curve dataset is a piecewise curve defined by pairs of points for Stress and the Larson-Miller parameter. There must be the same number of points defined for each.

Property	Description
MaximumStress	The maximum value of stress allowed in the calculation
MinimumStress	The minimum value of stress allowed in the calculation
MaximumTemperature	The maximum value of temperature allowed in the calculation
MinimumTemperature	This is effectively the creep temperature limit. Below this temperature, no creep damage is accumulated.
C	Larson-Miller constant
StressValues	The values of stress in the stress-P(LM) pairs
PLMValues	The corresponding values of P (Larson-Miller)
Comments	
References	

Larson-Miller Polynomial

The Larson-Miller Polynomial dataset defines the Larson-Miller Parameter as a polynomial function of Stress.

Property	Description
MaximumStress	The maximum value of stress allowed in the calculation
MinimumStress	The minimum value of stress allowed in the calculation
MaximumTemperature	The maximum temperature allowed in the calculation for this material
MinimumTemperature	This is effectively the creep temperature limit. Below this temperature, no creep damage is accumulated.
C	Larson-Miller constant
b	The constant term of the polynomial
a1	The multiplier of the linear term in the polynomial
a2	The multiplier of the quadratic term in the polynomial
a3	The multiplier of the cubic term in the polynomial
a4	The multiplier of the 4th order term in the polynomial
Comments	
References	

Chaboche

The ChabocheCreep materials dataset is defined as a parent dataset, with each temperature curve defined as a child set.

The parent properties are:

Property	Description
MaterialType	Numeric code defining the type of material. The material type is not currently used for creep calculations and is used purely for reference.
MaximumStress	The maximum allowable stress independent of tempera-ture
MinimumStress	The stress limit independent of temperature—below this value the damage is always set to 0.
MaximumTemperature	The maximum temperature allowed in the calculation for this material

Property	Description
MinimumTemperature	This is effectively the creep temperature limit. Below this temperature, no creep damage is accumulated.
k	The non-linear damage summation parameter
h	Compressive healing factor
Comments	
References	

The child properties are:

Property	Description
Temperature	The temperature value for this curve
A	The co-efficient of the Chaboche Creep Stress-Life curve
r	The exponent of the Chaboche Creep stress-life curve

Parameters k, MaximumStress and MinimumStress are shown in this dataset but are inherited from the parent and cannot be modified.

5.11.3 The Larson-Miller Method

The Larson-Miller method collapses the results from the creep tests onto a single curve that is independent of temperature.

$$P(T, t_r) = T(C + \log(t_r))$$

Where P is the Larson-Miller parameter, T is the temperature in Kelvin, C is the Larson-Miller constant, and t_r is the time to rupture. C is a parameter defined during the curve fitting process that, for steels, is around 20. The following figure shows a general example of this function.

Fig. 5-177 Larson-Miller curve

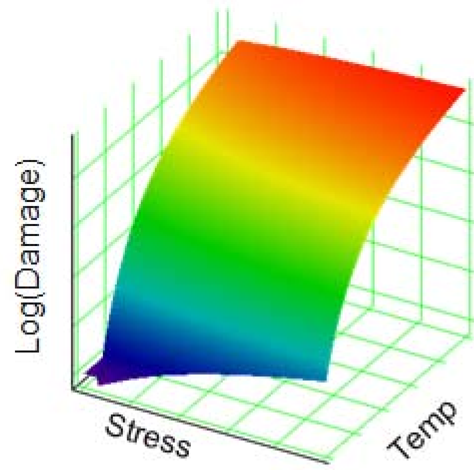
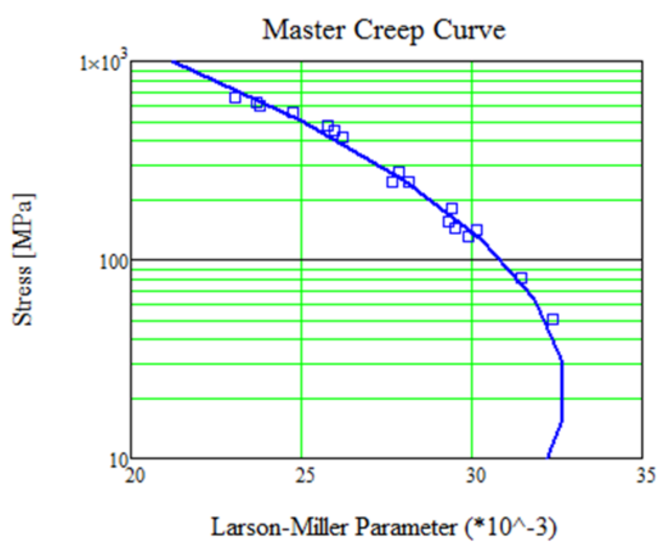


Fig. 5-178 Master creep curve

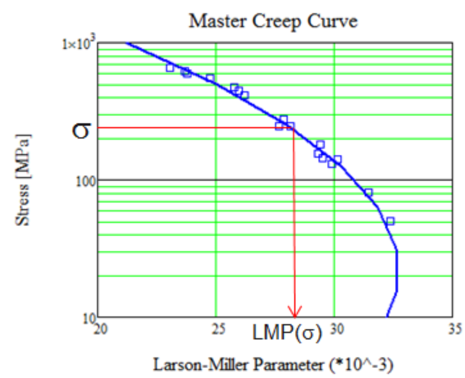
A master creep curve is generated from the test data for Stress v. Larson-Miller parameter. The curve looks like this:



In this case, a polynomial is fitted through the data—this is one of two methods allowed in the software to represent the master creep curve. The other is a piece-wise X-Y curve—these are described in "The Chaboche Method" on page 343. In both cases, for a given value of stress, the curve is used to obtain a value for the

Larson-Miller parameter. Polynomial materials definition is currently in terms of stress, in MPa.

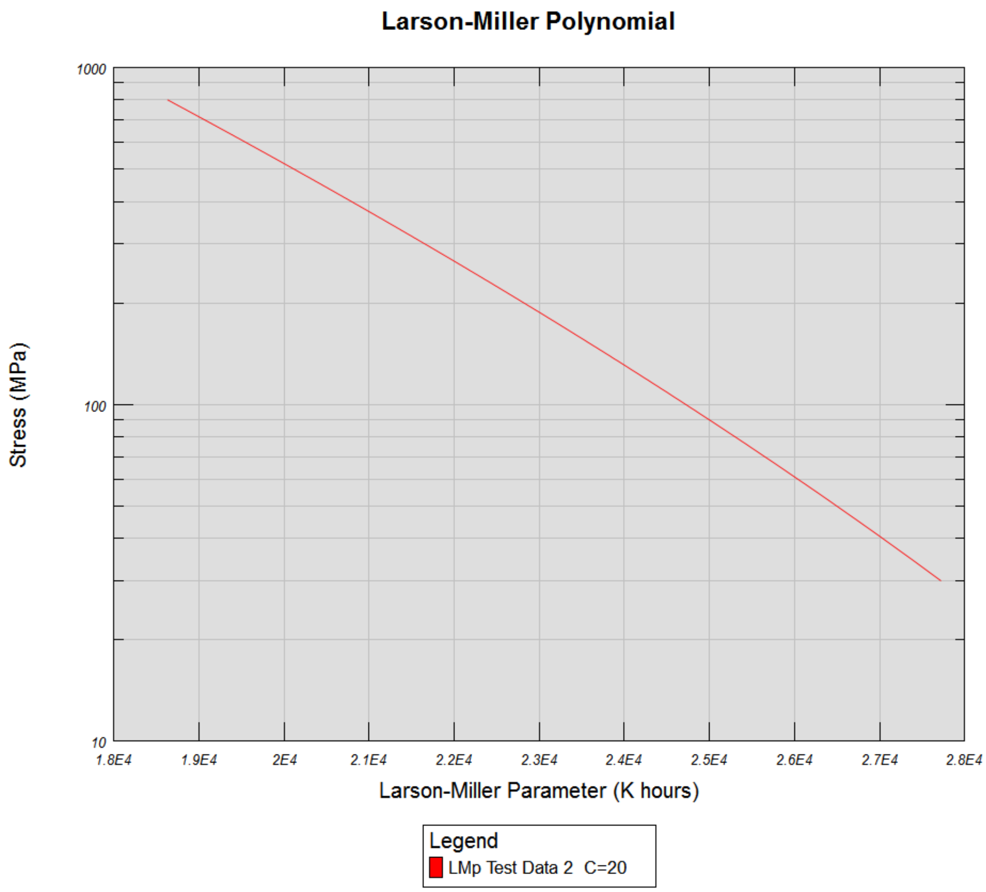
Fig. 5-179 Derive Larson-Miller Parameter (LMP)



If this is a polynomial, the Larson-Miller parameter can be calculated directly as a function of σ.

	LMp Test Data 2	Description
MaximumStress	800	Maximum Allowable Stress (MPa)
MinimumStress	30	Creep Stress Threshold (MPa)
MaximumTemperature	1050	Maximum Temperature (deg C)
MinimumTemperature	500	Creep Temperature Threshold (deg C)
C	20	Larson-Miller Constant
a4	0	Multiplier on the 4th order of the polynomial.
a3	0	Multiplier on the 3rd order of the polynomial
a2	-705	Multiplier on the 2nd order of the polynomial
a1	-3280	Multiplier on the linear term of the polynomial
b	3.41E4	Constant term of the polynomial
Comments		Comments
References		References

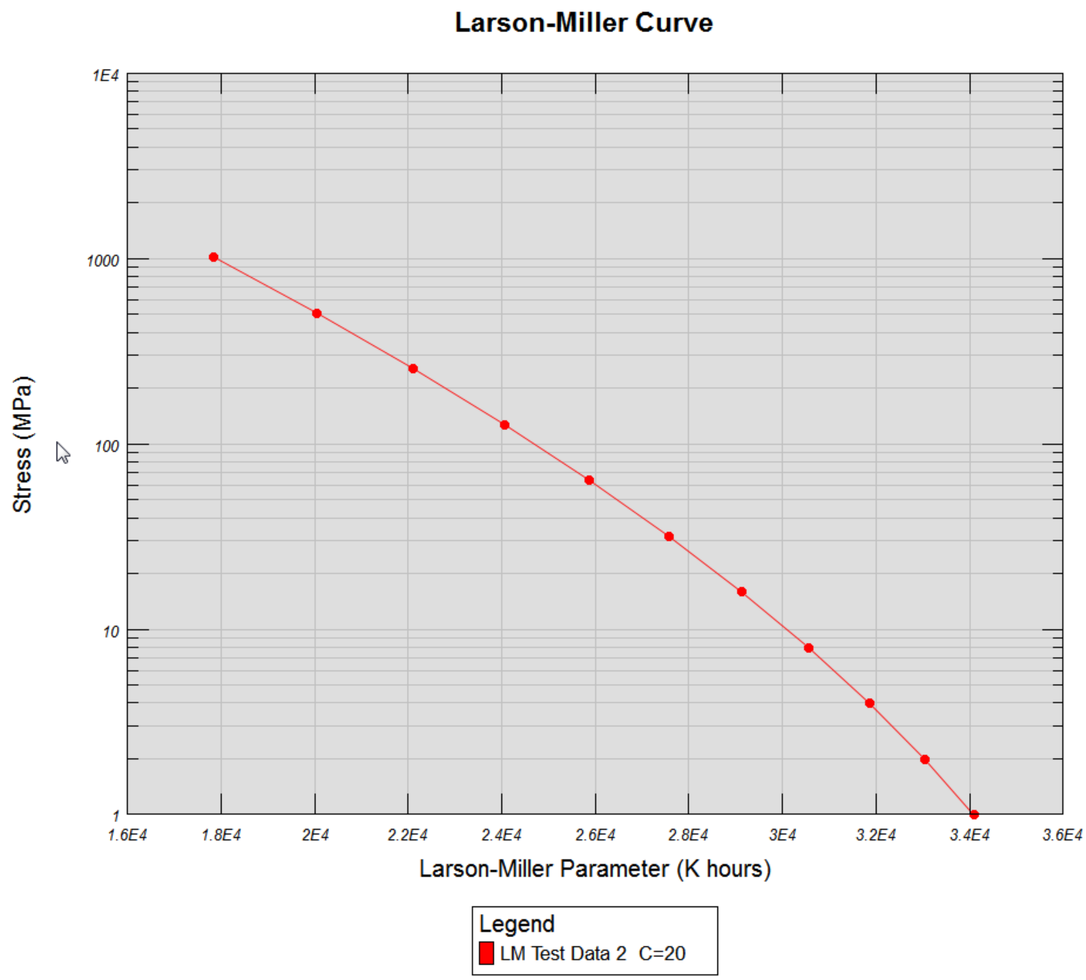
Fig. 5-180 LM Curve, constant = 20



For a piecewise curve, linear interpolation is used to get the Larson-Miller parameter from stress. Note that the data is plotted, and the interpolation performed with log Stress against linear Larson-Miller parameter.

	LM Test Data 2	Description
MaterialType		Material Type
MaximumStress	800	Maximum Allowable Stress (MPa)
MinimumStress	30	Creep Stress Threshold (MPa)
MaximumTemperature	1100	Maximum Temperature (deg C)
MinimumTemperature	500	Creep Temperature Threshold (deg C)
C	20	Larson-Miller Constant
StressValues	1,2,4,8,16,32,64,128,256,512,1024	Stress values (MPa)
PLMValues	3.408E4,3.303E4,3.185E4,3.055E4,2.911E4,2.755E4,2.586E4,2.404E4,2.21E4,2.003E4,1.783E4	Larson-Miller parameter values (K hours)
Comments		Comments
References		References

Fig. 5-181 Log stress v LMP



Note The Larson-Miller method assumes that compressive stresses contribute no creep damage.

From the Larson-Miller parameter and the temperature, the time to rupture is defined as

$$t_r = 10^{\left(\frac{PLM(\sigma)}{T} - C\right)}$$

So, in the simple case where we have a static temperature and a static stress, the lookup from the master creep curve to get the Larson-Miller parameter from stress and the temperature can be used to calculate the life (in hours) directly.

However, if either the stress or temperature, or both, varies with time, then the incremental damage is calculated per sample point. This is derived by calculating the time to rupture based on the stress S_n at time t_n and the corresponding value of T, T_n , at the nth time step. This value is called tr_n. The incremental damage is defined as

$d_n = \Delta t / tr_n$

where Δt is the time increment in hours between samples.

The total damage, D, is the sum of all the incremental damages for time steps 1 to N.

$$D = \sum_{n=1}^{n=N} d_n$$

The number of repeats of the sequence to failure (rupture) is:

$$R = \frac{1}{D}$$

The time to failure (rupture) in hours is obtained from the length of the input data, L (hours):

$$T_f = \frac{L}{D}$$

5.11.4 The Chaboche Method

Based on the work of Rabotnov (1958) and Kachanov (1969), Chaboche and Lemaitre (1978, 1981) proposed a creep damage evolution equation which is expressed as Equation 1:

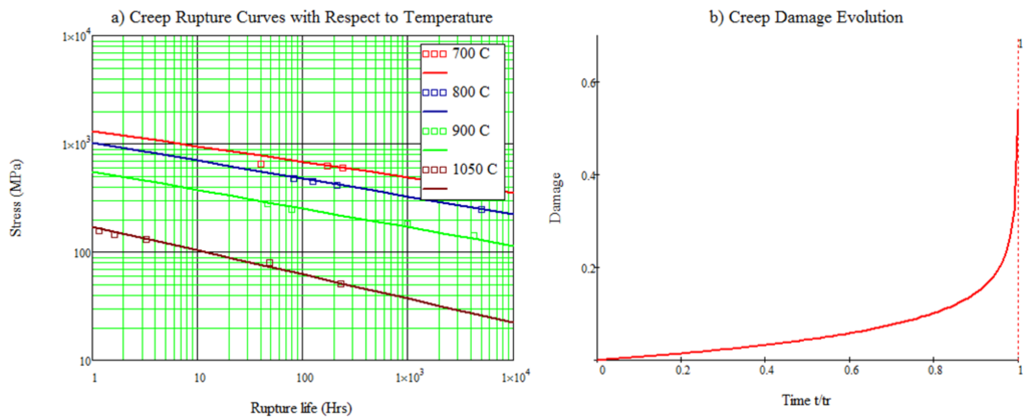
Equation 1

$$dD = \left(\frac{\sigma}{A}\right)^r (1 - D)^{-k} dt$$

where D is the damage variable ranging between 0, for the virgin condition, and 1 for the failed condition. A, r and k are temperature-dependent material parameters and are determined from creep rupture tests at several stress levels, as illustrated in “Creep Rupture Curves with Respect to Temperature” in Figure 5-182.

The creep rupture curve is idealized as a straight line in log-log space. Parameter A represents the intercept of the curve with the stress axis while the exponent r represents the slope. k is used to describe the non-linear damage evolution as observed from damage measurements. See “Creep Damage Evolution” in Figure 5-182. A value of k = 0 implies linear damage accumulation as per Robinson’s rule (1938). k has been found larger than r in general and is a function of increasing stress. However, constant k is preferred for creep-fatigue life calculations and the method used in DesignLife assumes that k is constant with respect to both stress and temperature. Some variations of the above model can be found in the work of Chaboche (1976, 1977). In this document, the focus is on life calculation for different stress and temperature cycles.

Fig. 5-182 Creep rupture and creep damage evolution curves



If both the stress and temperature are constant, a straightforward equation for the time to rupture is derived by integrating Equation 1. This comes out as:

$$t_r = \frac{1}{k+1}\left(\frac{\sigma}{A}\right)^{-r}$$

A and r are the values at the constant temperature; if this temperature is not one of the values for which tests have been performed, an interpolation algorithm is used to calculate the values for that temperature. A method that uses logarithmic interpolation for both A and r is used in DesignLife and this seems to give reasonable results when compared to test data. Note that because of the logarithmic nature of the interpolation it is done with temperatures in units of Kelvin. Extrapolation of A and r is allowed, as long as the temperature is within the maximum and minimum material temperatures.

When processing time histories we can calculate the incremental damage for a time series sample.

If the inter-sample time is Δt, and this is reasonably small, the incremental damage for stress σ is

$$\Delta D = \Delta t.\left(\frac{\sigma}{A(T_i)}\right)^{r(T_i)}.(1-D)^{-k}$$

where $A(T_i)$ and $r(T_i)$ are the material properties at the temperature over which incremental damage is being calculated, D is the total damage so far, and k is the non-linear summation factor.

The total damage D=D+ ΔD and this process is then repeated continuously, looping around the time series as many times as required, until D=1 at rupture.

The problem with this method is that it is very inefficient and an optimized algorithm is required for long durations. The solution relies on the non-linear damage evolution constant k being assumed invariant with temperature and that the time history is repeated a number of times before rupture occurs.

In this case we can remove the term involving k from the calculation of ΔD to reduce it to:

$$\Delta D = \Delta t.\left(\frac{\sigma}{A(T_i)}\right)^{r(T_i)}$$

The total damage D is calculated D=D+ ΔD and at the end of one repeat of the time series we have the damage for the first repeat.

To calculate the number of repeats to rupture:

$$N_f = \frac{1}{(k+1).D}$$

And the time to rupture, in hours, is:

$$t_r = N_f.Duration$$

...where Duration is the length of the signal in hours, assuming that the signal is repeating. This means the equation is:

$$Duration = \Delta t.N$$

...where N is the number of samples. This differs from the normal calculation of duration in DesignLife, which uses N-1. In long time histories, this difference is not really noticeable, but can be seen on short sequences.

The algorithm accounts for rare cases of compressive healing in materials. The healing coefficient 'h' is specified in the range $-1 \leq h \leq 1$ and applies to the incremental damage resulting from compressive stresses. The formula was derived by Chaboche and Lin (2006) and the total creep damage is obtained from Equation 2.

Equation 2

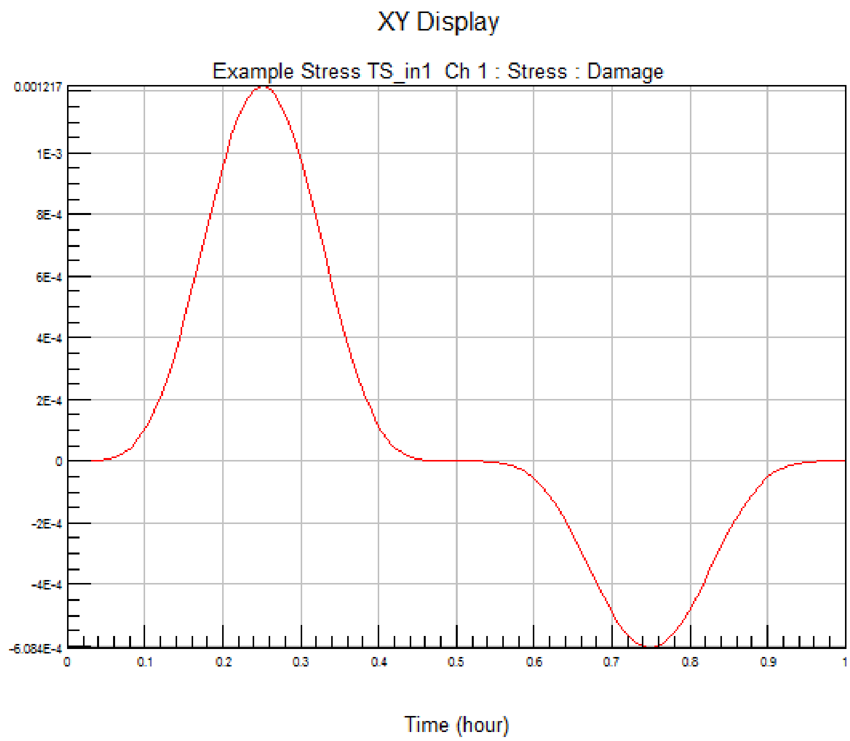
$$D_{total} = D(\sigma_{tension}) + h \cdot D(|\sigma_{compression}|)$$

where D_{total} = total creep damage, $D(\sigma_{tension})$ is damage associated with tensile stress and $D(|\sigma_{compression}|)$ is damage associated with compressive stress.

A value of h = 0 implies no creep damage arising from compressive stresses. This is the most commonly used assumption for creep-fatigue analysis. A value of h > 0 implies positive damage arising from compressive stresses. A value of h = 1 implies equal damage from compressive and tensile stresses. A value of h < 0 implies a proportion of compressive healing. The algorithm prevents the damage sum going negative.

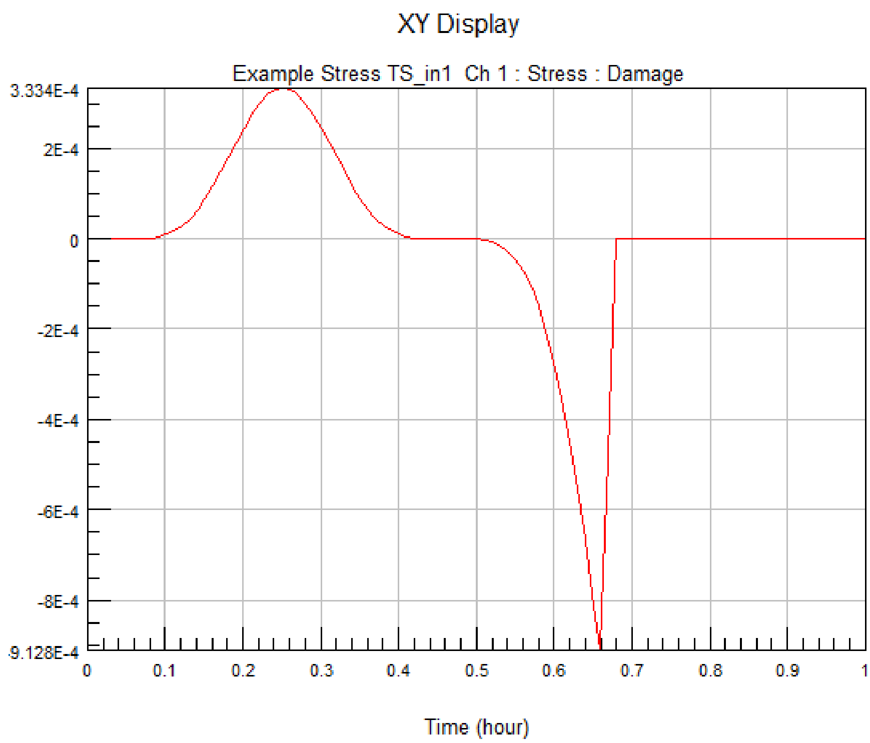
An example of how this affects the incremental damage is shown below, where the input stress history is a sine wave at a constant temperature. In this case h=-0.5 so the healing effect is half of the damaging effect.

Fig. 5-183 Input stress = sine wave at constant temperature



In the next example, the data has a negative mean, so the compressive stresses have a larger magnitude than the tensile stresses. This shows the point at which healing takes the total damage back to zero. All subsequent compressive stresses have an incremental damage of zero.

Fig. 5-184 Data with negative mean



Note that to use healing, the parameter h must be set in the materials data, and the minimum stress must be set to a negative value to allow the compressive stresses to be used in the algorithm.

5.11.5 CombinationMethod

For all load providers apart from the Vibration Load Provider, the analysis engine creates a stress tensor history, $\sigma_{ij}(t)$. In order to make a creep calculation, we need to reduce this stress tensor to a single scalar value to use in the creep materials curves. This process is called stress combination. The available options for combined stress damage parameters are:

- MaxPrincipal
- VonMises
- Shear
- AbsMaxPrincipal
- SignedVonMises

Note! Careful consideration should be taken when selecting the stress combination for creep. In cyclic fatigue damage calculations, the signed options (Absolute Maximum Principal, Signed Von Mises, Signed Shear) tend to be more conservative as the cycles go to the full range of the data. However, in creep calculations, cycles are not used, so unsigned options (Maximum Principal, Von Mises, Shear) can be more conservative. Bear in mind that if using Chaboche that compressive healing can only be taken into account fully if signed options are selected.

In general, a stress tensor has nine components, but due to symmetry, this reduces to six.

$$\sigma_{ij} \equiv \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{zx} \\ \sigma_{xy} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{yz} & \sigma_{zz} \end{bmatrix}$$

At a free surface, we can assume that there is no direct or shear stress applied to the surface. This means that if we choose an appropriate Cartesian coordinate system—one where the z-axis is a surface normal—all stress components containing a “z” disappear and we are left with a stress tensor with only three non-zero components.

$$\sigma_{ij} \equiv \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & 0 \\ \sigma_{xy} & \sigma_{yy} & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

However, creep occurs throughout the model so that, if only creep is considered, data can be processed from internal nodes and elements, which are normally discarded for fatigue calculations.

Most of the combination methods require the determination of principal stresses. The principal stresses are the eigenvalues of the stress tensor matrix.

In the case of 2-D stresses, the calculation of the two in-plane principal stresses reduces to the solution of a quadratic:

$$\sigma_{principal} = \frac{\sigma_{xx} + \sigma_{yy}}{2} \pm \sqrt{\left(\frac{\sigma_{xx} - \sigma_{yy}}{2}\right)^2 + \sigma_{xy}{}^2}$$

The third principal in this case is the surface normal stress, which is always zero.

Whether the stress tensor is flagged as 2-D or 3-D, the result is three principal stresses $\sigma_{1,2,3}$ which are ordered so that $\sigma_1 \geq \sigma_2 \geq \sigma_3$. These are useful for calculating the other combined stress options as follows.

MaxPrincipal

MaxPrincipal is the maximum principal stress. It may be smaller in magnitude than the minimum principal stress. In fatigue, this option can be less conservative and should be used with caution; however in creep it can be more conservative. If you are using Chaboche and wish to take healing into account in compression, it will be necessary to use one of the signed options, in this case AbsMaxPrincipal, to ensure that compressive stresses appear.

VonMises

VonMises is the von Mises stress. It is always positive (not signed by the absolute maximum principal stress) and will therefore not take into account any compressive healing if that option is in use for the Chaboche method.

Shear

The Shear stress is the Maximum Shear stress (Tresca Criterion). It is not signed and will therefore not take into account any compressive healing if that option is in use for the Chaboche method.

AbsMaxPrincipal

The Absolute Maximum Principal stress is defined as the principal stress with the largest magnitude:

$$\sigma_{AMP} = \sigma_3 \ if \ \left|\sigma_3\right| > \left|\sigma_1\right| \ otherwise \ \sigma_{AMP} = \sigma_1$$

SignedVonMises

The Signed von Mises stress is the von Mises stress, but forced to take the sign of the Absolute Maximum Principal stress, i.e.,

$$\sigma_{SVM} = \frac{\sigma_{AMP}}{\left|\sigma_{AMP}\right|} \cdot \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}$$

SignedShear

The Signed Shear stress is the Maximum Shear stress (Tresca Criterion), forced to take the sign of the Absolute Maximum Principal stress. A factor of 2 is applied to ensure that the parameter has the same value as the Absolute Maximum Principal stress under uniaxial loading conditions.

5.11.6 Stress Limits

For both methods, the solution is bounded for both low values and high values of stress. The **minimum allowable stress** effectively sets a creep limit below which no creep damage is accrued. This is particularly important when using a polynomial based Larson-Miller materials dataset as the equations can become unstable in areas where the data is not fitted to the curve.

The **maximum allowable stress** is the largest stress for which the calculation is performed. This could be defined as the UTS, or at least the UTS for a fixed temperature. However, because the Larson-Miller parameter is defined for a range of temperatures, the creator of the dataset should define the “maximum allowable stress”. If the calculation exceeds this value, “static failure” is reported and the damage is set to 1.

5.11.7 Temperature Limits

For both methods, the solution is bounded for both low values and high values of temperature. The **minimum allowable temperature** is the temperature below which no creep, or insignificant creep, occurs. This is typically around 50% of the melting point in Kelvin, but again may be noted based on materials tests.

The maximum allowable temperature is set to prevent temperatures close to or above the melting point being used in the analysis. If it is exceeded, damage is set to 1.

5.11.8 Duty Cycles

For creep calculations, the damage from each event is multiplied by the number of those events and the total damage from each event added together to get the total duty cycle damage.

There are no options—”Independent” and “CombinedFull” modes will give the same answer, so for performance purposes, only independent mode is required.

OutputEventResults

If this property is set to True, the accumulated damage (and other calculated results) associated with each individual event from a duty cycle will be output in addition to the total damage. Otherwise, only the total damage for all events is output.

StaticFailureDamage

This parameter sets a numerical value that may be set for the damage to a particular node or element in the event of the stress exceeding the maximum stress or the maximum temperature being exceeded (or both).

TemperatureSelection

This property controls the temperature value, or values, that are used in the calculation when a hybrid load provider is configured that provides multiple temperature-time values.

The options are:

- AllAvailable
- OverallMax
- OverallMin

- OverallMedian

In *AllAvailable* mode, the temperature time history is used point-by-point alongside the stress time history. In the other modes, the maximum, minimum or median value from the temperature data is used for each point in time as a constant temperature. This has the same effect as using the equivalent options in the hybrid load provider.

Note that for performance reasons, the Median is estimated as the average of the maximum and minimum values.

ScaleFactor

The scale factor is applied to the stress history on every node prior to the creep calculation. Note that scale factors can also be applied at a group level by editing the Material Map.

5.12 Short Fibre Composite Analysis Engine

Fig. 5-185 Short Fibre Composite analysis engine properties

Object Name: AnalysisEngineShortFibreComposite1 (Short Fibre Composite analysis engine)

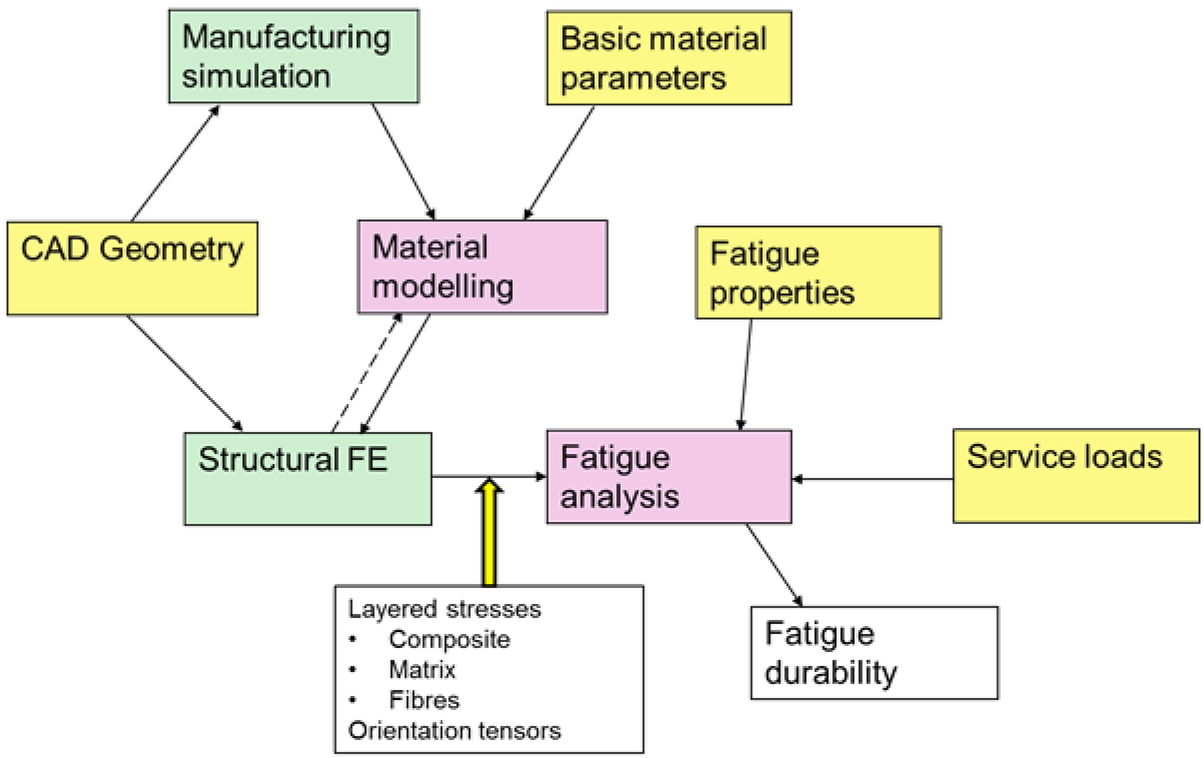
Name	Value	Description
⊟ **General**		
CombinationMethod	AbsMaxPrincipal	The method used to combine component stresses/strains
SNMethod	Basquin	The method used to define the SN curves used by the fatigue calculation
MultiAxialAssessment	None	Whether to perform assessment of the multi-axial stress state
CertaintyOfSurvival	50	Required confidence level on damage results
ScaleFactor	1	The scale factor to apply prior to damage calculation
OutputMaxMin	False	Whether to output max and min stresses
OutputMaterialNames	False	Whether to output material names to the results
OutputDistributedSource	False	Whether to output details of the distributed process that generated each result
OutputVibrationStats	False	Whether to output Vibration PSD parameters such as ExpectedZeroUpcrossings, Expected
⊟ **MeanStressCorrection**		
MeanStressCorrection	None	The method used to correct the damage calculation for mean stress
⊟ **DutyCycle**		
EventProcessing	Independent	How to process separate events in duty cycles
OutputEventResults	False	Whether to output results per event or not for duty cycle processing
⊟ **Advanced**		
CheckStaticFailure	False	The action to take on static failure
DamageFloor	0	The calculated damage value below which the damage is set to zero
MaxDamage	1E30	The maximum damage value
StaticFailureDamage	1E10	The damage value to insert on static failure

5.12.1 Introduction

Any analysis of components made from composite materials must take into account the fact that composites are, in contrast to the assumptions we normally make for metallic components, inhomogeneous and anisotropic. This means that the properties of a composite may in general be different for every element in the model and, for layered composite shell elements, also vary through the thickness of each element. In addition, properties including static and fatigue strength are strongly dependent on the direction of loading relative to the fibre orientations. For a given composite system (combination of matrix and reinforcing fibres), the properties of the composite are principally influenced by the fibre orientations, which in turn are determined by the manufacturing process.

Although the following discussion focuses on injection-moulded short fibre reinforced thermoplastics, the nature of the FE and fatigue analysis methods is such that they could also be applied to laminates where a simple ply-by-ply approach is considered adequate.

Fig. 5-186 General fatigue analysis process for composite materials



A general picture of the analysis process required for composite materials is illustrated in Figure 5-186. A manufacturing process simulation is required in order to determine the fibre orientations in the finished product. The type of simulation depends on the composite system. For example, a draping simulation might be required for a carbon-fibre layup. For a short fibre injection-moulded composite, the simulation will be of the moulding process. As far as the subsequent analysis is concerned, the key information deriving from this simulation is the fibre orientations, which depend on the flow and solidification of material in the mould. For a moulding simulation this information is summarized in the form of an orientation tensor, which summarizes the distribution of fibre orientations. The tensor looks like this:

$$a = \begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{bmatrix}$$

Given this information, together with other information about the composite, including the properties of the matrix and fibres, volume fraction of fibres, etc., the properties of the composite as a function of the orientation tensor may be predicted, and after mapping the anisotropic material stiffness matrices to the structural FE mesh, the stress analysis may be carried out. Typically, the properties will be assumed orthotropic on a ply-by-ply or layer by layer basis. Since for injection moulded parts the properties will vary through the thickness, they are normally treated by the FE as a number of layers, in the same way as a laminate. Alternatively, a software tool such as Digimat may be used in a strongly coupled analysis with the FE to replace the FE solver's normal material models.

The information required for the subsequent fatigue analysis is as follows:

- The stress tensor at the element centroid: Structures may be represented by solid elements (e.g. with a layer of solid elements per ply) and/or composite shells. For composite shells the stress tensor is required at each layer and section point (some FE codes allow more than one integration point or section point per layer).
- In addition to reading normal FE stress datasets, for ABAQUS and ANSYS only in this release, stress analysis results that have been calculated using Digimat to provide a user material model may be read. These stresses will be stored as State Variables in the FE results files. In this case, the averaged matrix and fibre stresses will be available in addition to the normal composite (macro) stresses. See the section on "FE Results" for details.
- The material orientation tensor at each calculation point. The material orientation tensor must be mapped to the structural mesh and in the same coordinate system as the stress analysis results. If the FE analysis has been executed with Digimat providing the material model, the orientation tensor will normally be available in the FE results file, stored as state variables. Otherwise it must be provided by the user, in the form of an ASCII file. This will be described later. The only exception to this requirement arises if solid elements are to be analyzed using SN curves or a Haigh Diagram generated by Digimat. In this case the orientation tensors are available in the Digimat analysis files and are not directly required by the analysis engine

The orientation tensor has to obey certain rules:

1. It is real.
2. It is symmetric ($a_{xy} = a_{yx}$).
3. The diagonal terms cannot be negative.
4. The tensor is normalized so that the sum of the diagonal terms $a_{11} + a_{22} + a_{33} = 1$.

The diagonal terms of the matrix describe the "share" of fibres pointing in the directions of the coordinate axes. A few examples follow.

3-D random distribution of fibres:

$$a = \begin{bmatrix} 0.333 & 0 & 0 \\ 0 & 0.333 & 0 \\ 0 & 0 & 0.333 \end{bmatrix}$$

2-D random distribution (xy plane):

$$a = \begin{bmatrix} 0.5 & 0 & 0 \\ 0 & 0.5 & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

Completely aligned in the x direction:

$$a = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

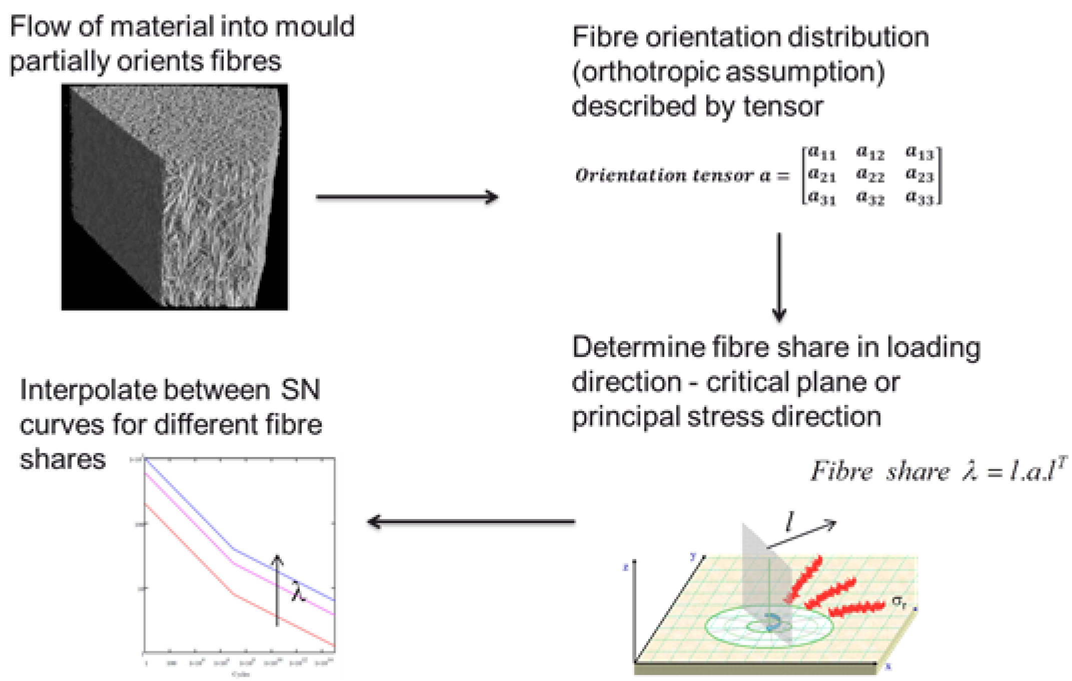
Mainly aligned in the y direction:

$$a = \begin{bmatrix} 0.05 & 0 & 0 \\ 0 & 0.9 & 0 \\ 0 & 0 & 0.05 \end{bmatrix}$$

The orientation tensor is used in conjunction with the loading direction or stress state to determine a suitable SN curve for use in the fatigue calculation.

Two methods are supported for determination of SN curves. The first method interpolates SN (Basquin) curves based on the fibre share in the loading direction. This method is summarized in Figure 5-187 and described in more detail below.

Fig. 5-187 Determination of fatigue properties depending on direction and orientation tensor



The second method uses a third-party software product called Digimat to predict the required SN curves. In order to use this method, the FE model solution must have been carried out linked with Digimat and the fatigue properties must also be defined in the Digimat project files. The theoretical background to this method will be briefly summarized later.

5.12.2 Engine Operation Summary

The basic operation of the Short Fibre Composite analysis engine can be summarized in Figure 5-188 and Figure 5-189, depending on whether SN curves are determined by interpolation or using Digimat.

Fig. 5-188 Short Fibre Composite analysis engine process—SN curves by interpolation

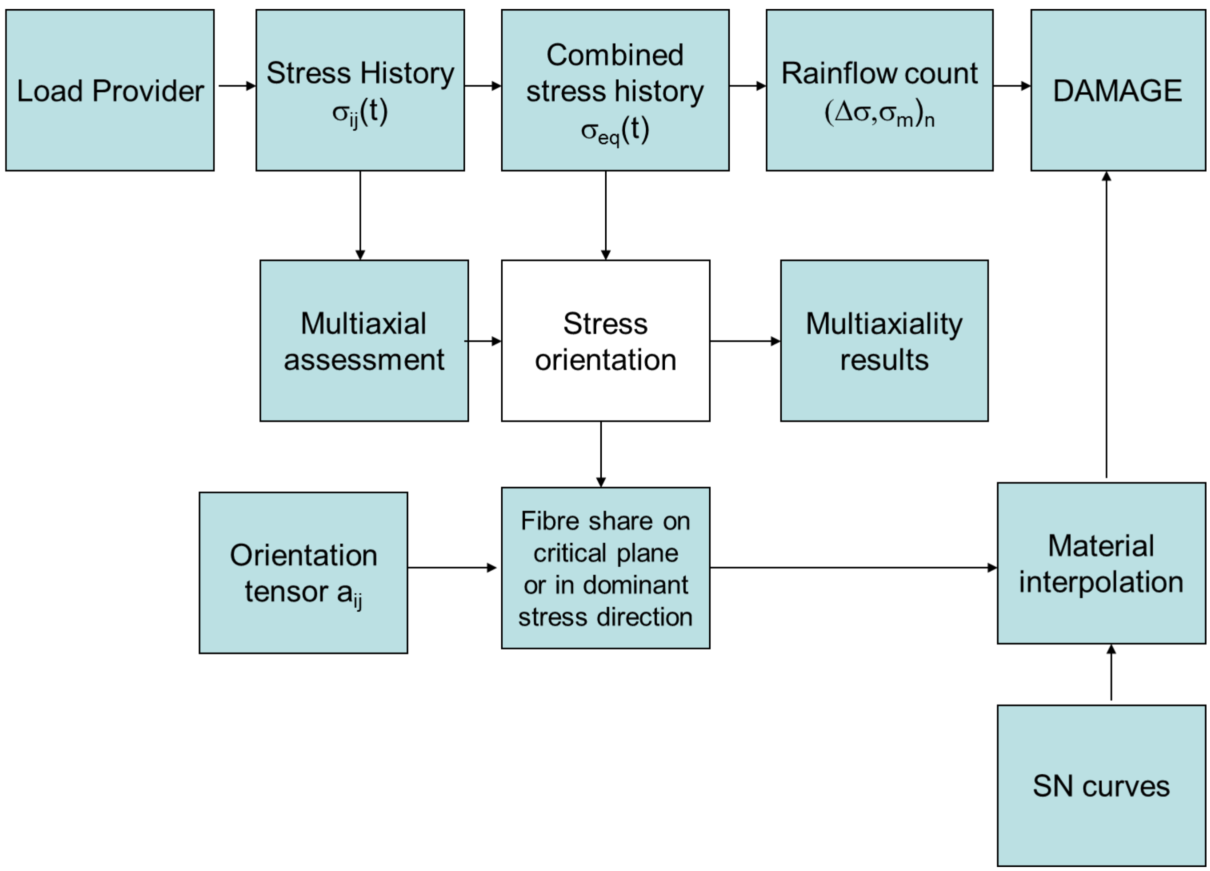
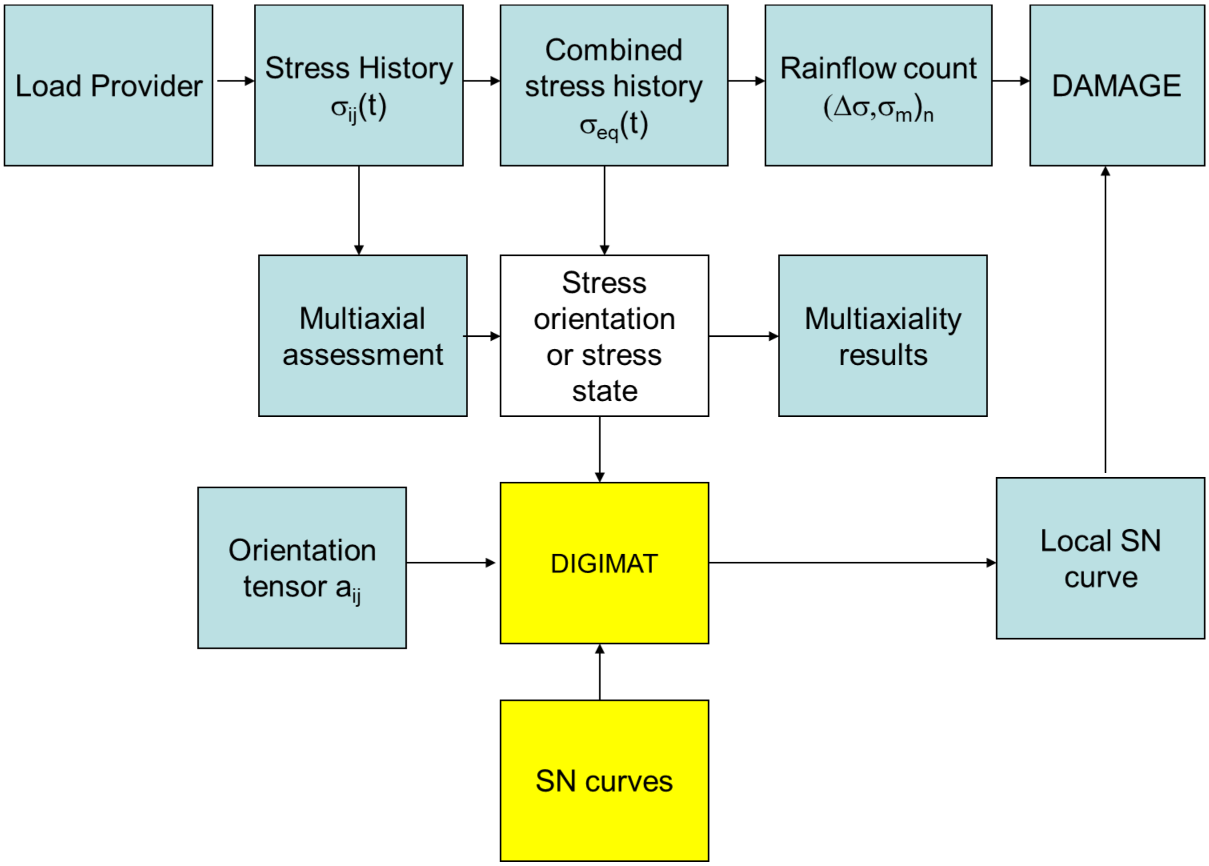


Fig. 5-189 Short Fibre Composite analysis engine process—SN curves from Digimat



Essentially, the Short Fibre Composite Analysis Engine is based on the Standard SN Analysis engine, but has been modified to allow analysis of anisotropic materials.

In DesignLife the engine has certain limitations, namely:

- It can handle only element centroid results
- Solid elements should be non-layered, with results in global (3D) co-ordinates
- Shell elements should be layered with results in local (2D) co-ordinates, i.e. the x-y plane of the results co-ordinate system lies in the plane of the element
- Stress combinations are limited to Abs Max Principal and Critical Plane stress

Note So that multi-layered composite combined solid and shell element models can be translated correctly from an *ANSYS* analysis, a change has been made in the way that solid property IDs are identified, based on *section ID assignments*.

In ANSYS, solid elements can have section ID assignments even though they do not need them or they do not exist, and this can be used as a way of grouping elements for a later analysis.

However if any of the assigned section IDs for solids have shell section attributes defined in the model, the property IDs for the solids are now set to zero so as not to get confused with any possible shell elements with the same assigned section IDs.

In this situation, the solids can still be grouped by assigning new section IDs that do not have shell sections defined, or alternatively by using dummy repeated element type definitions (translated as Element Sets) or dummy material datasets (translated as different material groups).

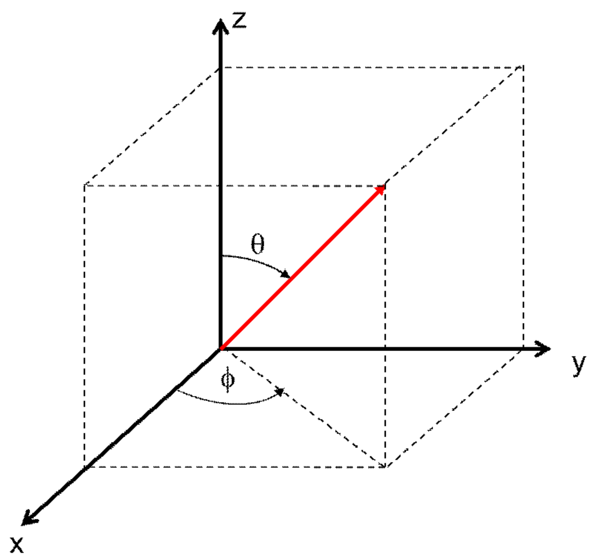
Unless otherwise defined here, details and definitions may be found in the section on the “Standard SN Analysis Engine”.

The basic analysis process is similar to that of the Standard SN analysis engine, and is executed for each analysis entity (Element + Layer ID + Section point ID) and critical plane where appropriate.

1. The stress tensor history, as a function of time or data points, is assembled from the information provided in the load provider. This applies to time-series, constant amplitude and hybrid load providers. See the section on Loading for details. Duty cycles are handled according to the same principles as for the Standard SN Analysis engine.
2. From the stress tensor, extract a combined stress parameter, either Abs Max Principal or Critical Plane. Note that if the Critical Plane option is being used with solid elements (3D stresses) the distribution of critical planes to be analysed is also three dimensional. In order to achieve a resolution of 10 degrees, around 200 planes must be analysed, each plane being defined by the orientation of its normal, using two angles (theta and phi). This will significantly increase run-time.
3. If Abs Max Principal stress is used, a multiaxial assessment is automatically carried out in order to determine the dominant stress direction and the stress state. For shell elements, the multiaxial assessment is carried out as described in the section on the SN analysis engine. However, this method is suitable for 2D (plane stress) states only. For solid elements the method must be extended to 3D. Mathematically the method is similar to the standard 2D method, but each direction must be defined by two angles (theta and phi) and the resulting stress state is characterised by two biaxiality ratios rather than one.
4. The combined stress history is rainflow counted to obtain a list of cycles, each with a known range and mean value.

5. The dominant stress direction or critical plane orientation is compared to the material orientation tensor in order to determine the fibre share in the loading direction.
6. Either...
   - If Basquin curves are being used, the fibre share is used to determine the local SN curve in the current loading (stress) direction by interpolation.
   - Or, if SN curves are being determined using the link to Digimat, Digimat will predict a local SN curve based on the relationship between the stress direction or stress state and the orientation tensor.
7. Damage is calculated and accumulated based on the local SN curve.

Fig. 5-190 For 3D (solid element) cases



For 3D (solid element) cases, the critical plane or dominant stress orientation is defined using two angles, theta and phi. Figure 5-190 illustrates what these mean with reference to the results co-ordinate system (xyz).

5.12.3 Fibre Share Calculation

The fibre share on the critical plane or in the dominant stress direction is determined by resolution according to the following equation:

Fibre share $\lambda = lal^T$

where l is a vector in the direction of the normal to the critical plane or the dominant stress direction.

5.12.4 Material Properties and Local SN Curve Determination

The Short Fibre Composite analysis engine makes fatigue calculations using an SN approach, with the material curves for each location being uniquely determined based on the stress state and the fibre orientations. There are three methods for determining the curves. The first method uses SN (Basquin) curves from the MXD database, with the local SN curve being determined by interpolation. The second method calls out to a third party software product (Digimat) to determine the appropriate SN curve for each location and stress state or direction. The third method also calls out to Digimat, but in this case to determine a Haigh diagram for each location so that the mean stress effect may be considered.

Calculation from Basquin Curves

If the first (Basquin curve) approach is being used, the Short Fibre Composite analysis engine makes fatigue calculations using an SN approach, with the SN curve for each location being uniquely determined based on the stress state and the fibre orientations.

The Short Fibre Composite analysis engine requires a specific material data type, “Short Fibre Composite Basquin Curves”. This data type contains a set of one or more SN curves, which may be used to determine the SN curve to be used for each calculation point and orientation, by interpolation and extrapolation.

Each material dataset consists of a parent (containing generic properties) and a number of child SN curves. This type of data is found only in the MXD database (not supported in the nSoft MDB database). The generic data consists of the following parameters:

Parameter Name	Description
MaterialType	Numeric code defining the type of material.
RR	R-ratio (ratio of minimum to maximum load) of the tests used to define the S-N curve
VolFraction	The volume fraction of fibres in the composite
Nfc	Numerical fatigue cutoff life. Beyond this life, damage will be assumed to be zero.
SE	Standard error of $\log_{10}(N)$. This is used to calculate the life adjusted to a certain probability of survival.
Comments	
References	

There must be at least one child dataset (nCode Short Fibre Composite Basquin data set) which consists of the following parameters:

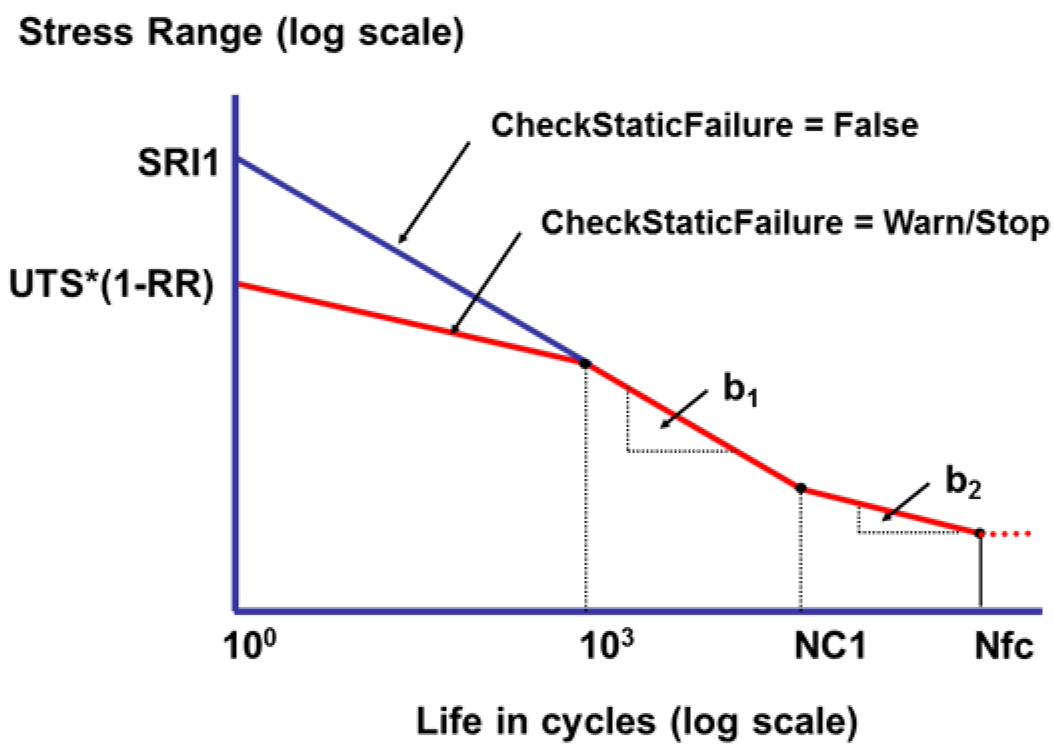
Parameter Name	Description
f	Fibre share in the loading direction for the SN curve (corresponding to λ in this theory document)
Phi	Orientation of loading direction compared to dominant fibre direction
UTS	Ultimate tensile strength. This is required for the mean stress correction (if used) and may also be used to check if the UTS is exceeded and to modify the upper part of the S-N curve.
UCS	Ultimate compressive strength. This may be used to check if the UCS is exceeded.
SRI1	Stress range intercept
b1	First fatigue strength exponent
Nc1	Transition life
b2	Second fatigue strength exponent
T	Temperature at which tests were conducted
RH	Relative humidity at which tests were conducted

Shaded items are mandatory. The other items may provide useful information, but are not used by the analysis.

Typically, S-N curves will be determined for different values of the fibre share f, by testing specimens cut from an injection-moulded sheet, and oriented at different angles to the flow direction.

The individual S-N curves are basically in the same format as Standard SN curves, that is, the resulting S-N curve is in three segments, as illustrated in Figure 5-191:

Fig. 5-191 Short Fibre Composite Basquin curve



Note that, for composites, there is typically no obvious transition to a lower slope at longer lives, so normally b1=b2 and Nc1 is arbitrary.

The S-N curve is otherwise interpreted in the same way as for the Standard SN analysis engine, including the modification of the SN curve in the region below 1000 cycles, when the property CheckStaticFailure is set to Warn or Stop.

If there is only one S-N curve defined in the dataset, this is used by the engine directly. Otherwise, if there is more than one S-N curve defined for different values of the fibre share f, a new SN curve will be determined for each location and orientation, based on the share of fibres in the loading direction at that point. The parameters of the new SN curve are determined by interpolation/extrapolation as follows.

Suppose that we have a pair of S-N curves defined by the following parameters:

Parameter Name
$f_{1,2}$

Parameter Name
$UTS_{1,2}$
$SRI1_{1,2}$
$b1_{1,2}$
$Nc1_{1,2}$
$b2_{1,2}$

and we need to calculate the S-N curve corresponding to the fibre share λ .The method used is similar to that used to interpolate S-N curves in the Seam Weld analysis engine.

First we calculate an interpolation factor:

$$I = \frac{\lambda - f_1}{f_2 - f_1}$$

Then the UTS and Intercept of the new S-N curve are calculated by linear interpolation/extrapolation.

$$UTS_{calc} = UTS_1 + (UTS_2 - UTS_1) \cdot I$$

$$SRI1_{calc} = SRI1_1 + (SRI1_2 - SRI1_1) \cdot I$$

This defines the stress level at 1 cycle.

Next, the transition life $Nc1_{calc}$ is determined by interpolation/extrapolation.

$$Nc1_{calc} = Nc1_1 \cdot \left(\frac{Nc1_1}{Nc1_2} \right)^I$$

The stress levels at the transition life s_{11} and s_{12} are calculated for the two original curves, and then the stress point at $Nc1_{calc}$ on the new curve is calculated from:

$$s1_{calc} = s1_1 + (s1_2 - s1_1) \cdot I$$

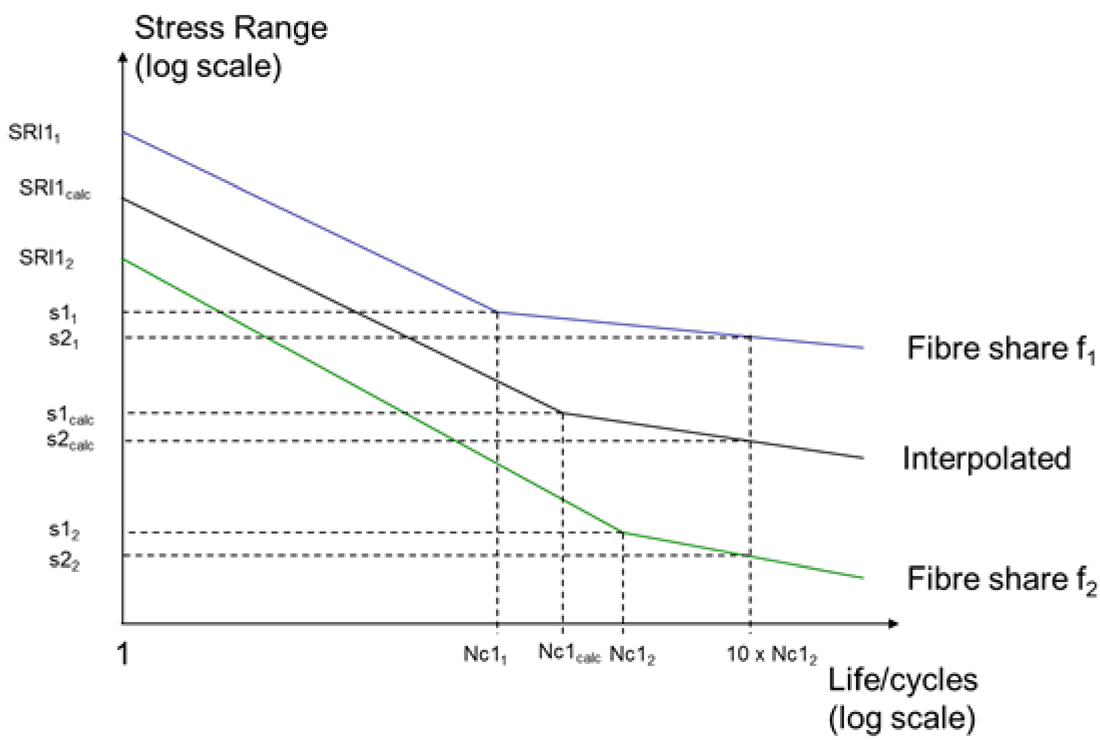
These two points define the first section of the S-N curve up to $Nc1_{calc}$ cycles. The lower section of the curve is defined by finding a third point as follows.

A life value is defined, being 10 x the greater of the Nc1 values for curves 1 and 2. At this life we can calculate corresponding stress values $s2_1$ and $s2_2$. We can then interpolate/extrapolate these values to determine $s2_{calc.}$

$$s2_{calc} = s2_1 + (s2_2 - s2_1) \cdot I$$

The interpolation/extrapolation process is illustrated in Figure 5-192:

Fig. 5-192 S-N curve interpolation/extrapolation



Note that it is possible to define curve sets which, when extrapolated to fibre share 0 or 1 (the extreme possible values) would produce an invalid curve, for example, with negative intercept, or positive slope. Such datasets must be considered invalid, and in these circumstances the analysis will stop with an error message.

After the calculation of the S-N curve, the rest of the calculation is carried out in the same way as for the Standard SN analysis engine.

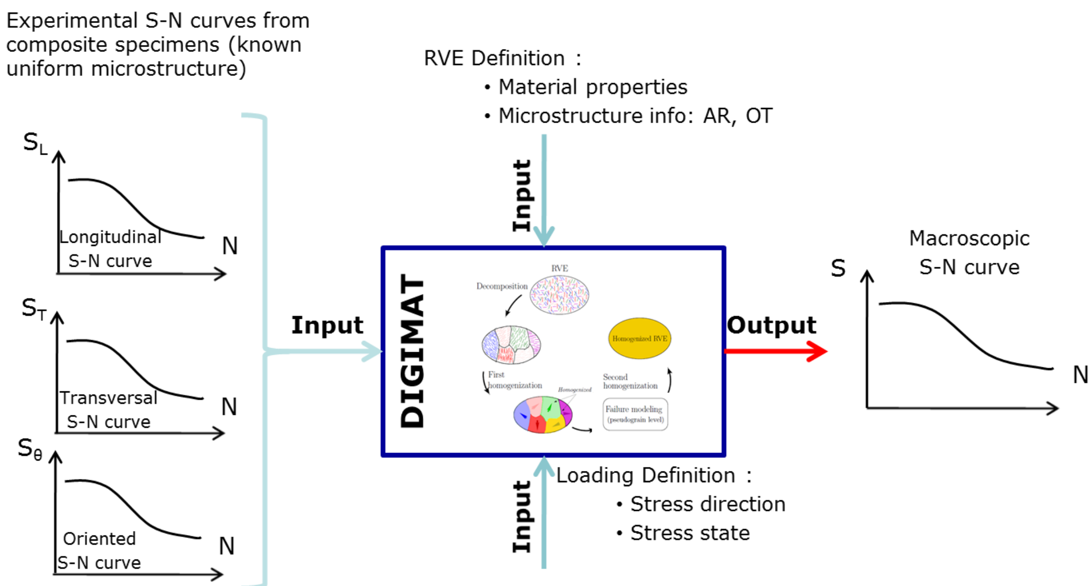
SN Curves from Digimat

Alternatively, the SN curves required for the fatigue calculation may be obtained by calling out to a third party software product, Digimat. Digimat is a software system that provides a number of tools for modelling the behavior and properties of composite materials, including a high cycle fatigue model suitable for short fibre composite fatigue applications where the behavior can be assumed mainly elastic. The fatigue model is based on the well-known Tsai-Hill composite failure criterion, extended to fatigue damage and applied at the meso-scopic (pseudo-grain) level.

In order to use this method, the FE model should have first been solved using Digimat, and the fatigue properties must also have been defined in Digimat.

For further details of the method, the user should consult the Digimat documentation. A short summary is provided here, and illustrated in Figure 5-193:

Fig. 5-193 SN curve estimation from Digimat



The following properties of the material must be defined in the Digimat system:

- Experimental SN curves
    - These should be determined using fatigue test specimens with as uniform as possible a microstructure. For example, they might be cut from the center of injection-moulded flat plates.
    - Three SN curves are required, determined using specimens cut parallel to, transverse to and at a specified orientation to the flow direction.
    - No particular curve formulation is required. The SN curves are defined by paired X-Y points (same life values in each curve).
    - The SN curves should be determined at the same R-ratio.
- The basic properties (elastic modulus and Poisson ratio) of the matrix and fibres and static strengths
- The composition of the test specimens (volume fraction of fibres, fibre aspect ratio)
- The orientation tensor (averaged across the section) in the test specimens

Digimat processes this data based on the principles of mean field homogenization (MFH). MFH involves describing a representative volume element (RVE) of a composite as a collection of pseudo-grains with different orientations, each of which has unidirectional fibres. The fatigue strength of each pseudo-grain at each life value is described by the Tsai-Hill criterion.

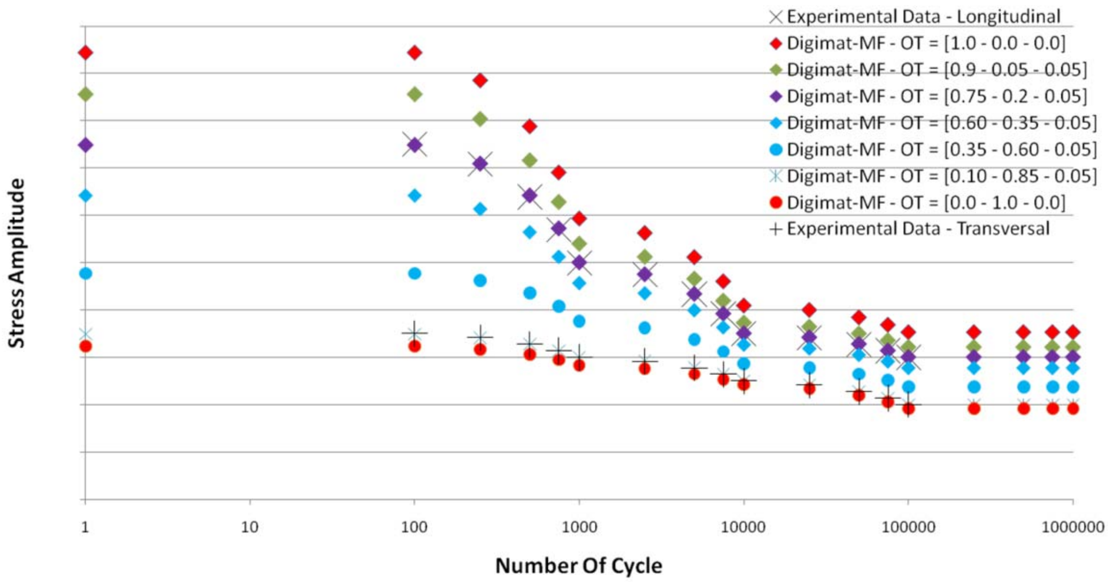
$$\left(\frac{\sigma_L}{S_L(N_c)}\right)^2 + \left(\frac{\sigma_T}{S_T(N_c)}\right)^2 + \left(\frac{\sigma_{LT}}{S_{LT}(N_c)}\right)^2 - \frac{\sigma_L \sigma_T}{S_L(N_c)^2} = 1$$

with the required fatigue strengths at each life value (SL(Nc) etc.) being determined by reverse engineering based on the experimental SN curves.

Once this has been done, the fatigue strength of any RVE at any life value may be predicted based on the stress direction or stress state and the microstructure of that RVE as described by the orientation tensor.

The Short Fibre Composite analysis engine can make use of this capability by calling out to Digimat to provide suitable SN curve for each fatigue calculation, based on the orientation tensor at that point and on the stress direction or stress state.

Fig. 5-194 SN curves may be estimated by Digimat based on experimental SN curves and local stress state and microstructure.



In this way, each calculation point is provided by Digimat with a potentially unique SN curve in the form of paired X-Y points (stress-life) as illustrated in Figure 5-194.

The fatigue calculation then proceeds as for a normal SN calculation. The SN curve is used as a lookup table with logarithmic interpolation between points. Points beyond the limits of each curve are determined by extrapolation.

Haigh Curves from Digimat

As an alternative to calling out to Digimat to receive a unique SN curve at each location, a set of Haigh curves can be obtained. The basic principles are the same as for Digimat SN curves, but the material information is returned in the form of a set of constant life curves. The format and processing of the curves within the Short Fibre Composite analysis engine is identical to that described in section "Standard SN Analysis Engine" on page 84.

5.12.5 Analysis Engine Properties

The analysis engine has the following properties.

CombinationMethod

The analysis engine uses the information from the load provider to create a stress tensor history, $\sigma_{ij}(t)$. In order to make a fatigue calculation, we need to reduce this stress tensor to a scalar value so that we can cycle count it and compare the resulting cycles to the local S-N curve. This process is called stress combination.

The available options for combined stress parameters for this engine are Abs Max Principal and Critical Plane. When used in conjunction with Basquin Curves (see SN Method below), these damage parameters have the same meanings as in the Standard SN Analysis engine. Note that the engine assumes that the stresses are 2-D stresses, and will therefore ignore any interlaminar shear stresses.

When the SN method is set to Digimat, the behavior is a little more complex. The Critical plane option assumes that the stress state on each critical plane is uniaxial, so the SN curve calculated by Digimat will correspond to a uniaxial stress normal to the critical plane. For the Abs Max Principal option, the analysis engine also carries out a multiaxial assessment and uses the results of that assessment to define a normalised averaged stress state for Digimat. The SN curve provided by Digimat corresponds to this stress state (approximately proportional loading assumption).

SN Method

The SN Method property defines the format of the SN data to be used for the fatigue calculation. The options are:

Basquin - The local SN curve will be determined based on a set of SN (Basquin) curves from the nCode MXD database. Note that the corresponding Material Type property on the Material Map should be set to “Short Fibre Composite Basquin curves”.

Digimat - The local SN curve will be determined by calling out to Digimat. Note that the corresponding Material Type property on the Material Map should be set to “Digimat SN”. The link to the required material properties is defined using a Digimat material database object. This object stores the path to the Digimat project (.dmp) files, which in turn link to the required material data files.

DigimatHaigh - A set of Haigh Curves will be determined by calling out to Digimat. Note that the corresponding Material Type property on the Material Map should be set to “Digimat SN”. The link to the required properties is defined using a Digimat material database object in the same way as described above. The Digimat material definition should include the information required to quantify the mean stress sensitivity of the material.

MeanStressCorrection

The main feature of a stress cycle that affects fatigue damage is its range. Fatigue damage is also influenced by the mean stress of each cycle. Mean stress correction methods allow the effect of mean stress to be modelled and taken into account in the life prediction. In this engine, there are two options: GoodmanTensionOnly and None. These work in the same way as for the Standard SN analysis engine.

MultiaxialAssessment

The multiaxial assessment property has two possible settings - None and Standard. The way the Standard assessment method works is as described in the section on the Standard SN analysis engine, except that to support solid elements, the method has been extended to 3D stress states. Mathematically, this method is similar, but the dominant stress direction now requires two angles to define it (as illustrated in Figure 5-190), and there are also two biaxiality ratios. For the 2D case (shell elements) the angle theta will always be 90 degrees, and one of the biaxiality ratios will be zero.

If None is selected, no multiaxial assessment results will be produced. However, if CombinationMethod = AbsMaxPrincipal, a multiaxial assessment is required and will be automatically carried out, whether or not it is requested, in order to determine the dominant stress direction (plane angle) to be used for determining the fibre share, and when using Digimat SN curves, the biaxiality ratio is also used.

CertaintyOfSurvival

The certainty of survival allows statistical variations in material behavior to be taken into account. It works in the same manner as for the Standard SN analysis engine. The default setting is 50% (mean life will be calculated).

ScaleFactor

This scale factor will be applied to the stresses prior to the fatigue calculation. The default value is 1.0

OutputMaxMin

When this property is set to True, the maximum and minimum values of the combined stress (Abs Max Principal or Critical Plane), as passed into the rainflow counter, will be reported in the results.

OutputMaterialNames

When this property is set to True, the material names will be reported in the results.

EventProcessing

A duty cycle is a sequence of events to be processed through the analysis engine. The creation and various different types of duty cycle are described in the section on Loading. Duty cycles may be processed in three different ways, which are basically the same as described in the section on the Standard SN analysis engine.

- **Independent** mode. Each unique event in the duty cycle is processed separately, and the damage from each is multiplied by the number of times it appears in the complete sequence. The total damage is the sum of the damage from all the different events. This is fast, but may miss significant cycles that cross events. Note that when CombinationMethod = AbsMaxPrincipal, because the dominant stress direction may be different for each event, so may the S-N curve.
- **CombinedFull** mode. All the events in the duty cycle are concatenated, including all repeats of each event, and the resulting stress history is processed as if it were one long event. This is accurate, but may take a long time to process.

- **CombinedFast** mode. An intermediate method that cycle counts events individually, but also captures cycles that cross events. A good compromise between the other options usually.

OutputEventResults

If this property is set to True, the accumulated damage (and other calculated results) associated with each individual event from the duty cycle will be output in addition to the total damage. If either of the "Combined" options is being used, some cycles may cross events. The damage associated with these cycles is split equally between the two events with which it is associated.

CheckStaticFailure

Static failure checking compares the maximum value of the CombinedStress, as cycle counted, with the material UTS. It also compares the Equivalent stress amplitude (after any mean stress correction) with the UTS. There is currently no criterion applied for compressive failure. There are three options:

- **Warn**—The calculation continues, but a warning message is issued. The damage for this entity is also assigned a numeric value equal to the value of the StaticFailure Damage property, typically 2.0.
- **Stop**—The analysis job stops with a message.
- **False**—No static failure check is made. Damage is calculated up to the numerical limit defined by the MaxDamage property. Note that this may use an extrapolation of the S-N curve and may predict damage values per cycle greater than 1.0.

Note that the static failure check is simply a comparison of the maximum combined stress at each calculation point with the UTS at that point and orientation. It is an indicator of high stress, and a warning of unrealistic life predictions, but should not be interpreted as an indication that the component is likely to experience static failure. Note also that setting StaticFailure = Warn or Stop will also result in modification of the S-N curve below 1000 cycles, in the same way as for the Standard S-N engine. If you wish to avoid this modification, it is recommended to set this property to False.

DamageFloor

This parameter sets a lower limit on calculated damage. If a cycle has less predicted damage, the damage value is set to zero. This parameter is somewhat redundant because similar functionality may be accessed using the fatigue cutoff Nfc in the material definition.

MaxDamage

This parameter sets a numerical upper limit on the damage that can be predicted for each cycle.

StaticFailureDamage

This parameter sets a numerical value that may be set for damage to a particular calculation point in the event of the stress exceeding UTS. See "CheckStaticFailure" above.

5.12.6 Note on Results

In addition to the usual fatigue analysis results, the Short Fibre Composite analysis engine outputs the Fibre share used at each calculation point. It also reports a direction in terms of two angles theta and phi for each calculation point, which for *CombinationMethod = CriticalPlane* will be the critical plane orientation, and for *CombinationMethod =AbsMaxPrincipal* will be the dominant stress direction.

Concerning multiaxial assessment, requesting a multiaxial assessment will normally result in the following extra columns in the results:

- Two biaxiality ratios (Mean biaxiality ratio 1 and 2).
- Non proportionality
- Dominant stress direction (phi and theta).

However, if *CombinationMethod = AbsMaxPrincipal*, the dominant stress direction and the Plane Angle will be the same, so only Plane Angle is reported. Note that for the 2D case, the angle theta is always 90 degrees, and "Mean biaxiality ratio 2" is always 0.

5.13 Animation Analysis Engine

Fig. 5-195 Animation Analysis Engine properties

Object Name: AnalysisEngineAnimation1 (Animation analysis engine)

Name	Value	Description
General		
Filename	animationfile.txt	Name of the animation file to create
Overwrite	True	Whether to overwrite an existing animation file
Decimation	1	The decimation factor for reducing the number of frames output
MaxFramesToOutput	100	Set a limit for the number of frames to output

5.13.1 Introduction

The function of the Animation Analysis Engine is to produce an animation file by processing an FE model. The animation file contains displacement information for use in animations, and is typically smaller that the FE model, and faster to animate.

The animation file that is generated can be binary with an .anm extension, or ASCII with any other extension.

5.13.2 Analysis Engine Properties

The analysis engine has the following properties.

Filename

Either ASCII or binary animation files can be created. If the file specified has an extension of .anm, (which is the default if no extension is specified), then a binary file will be created. Any other extension, e.g. .txt, will be written as an ASCII file.

Note that an ASCII file will take longer to create, and will display more slowly. ASCII can be viewed in text editors, but binary is faster in operation

Overwrite

If set to True and the animation file already exists, the existing file will be overwritten. If set to False and the animation file already exists, the existing file is not overwritten and an error message is displayed. The job will not run.

If the animation file does not exist, then this property has no effect.

Decimation

This property reduces the number of frames of data written into the animation file.

If the value is set to an integer, NN, greater than 1, then only 1 frame in NN is written to the animation file. For example, if the value is set to 10, then frame numbers 1,11,21,31 etc. will be written to the animation file.

Note that this simple method of reducing the quantity of data written can lose peaks in the data. Therefore, it should typically only be used when the input data has a higher than necessary sample rate, in which case the effect of losing peaks would be less.

MaxFramesToOutput

The animation engine can generate very large output files if the loading history being applied is long. To prevent the creation of files that are too big this property limits the number of frames that will be output.

Beware that increasing this for long loading histories may cause the analysis to take a long time, and later display of the animation files may be slow.

5.14 Safety Factor Analysis Engine

Fig. 5-196 Safety Factor Analysis Engine Properties

Object Name: SafetyFactorEngine (Safety factor analysis engine)

Name	Value	Description
General		
SNFactorOfSafety	Standard	The method used to calculate the stress based safety factor
CombinationMethod	AbsMaxPrincipal	The method used to combine component stresses/strains
MeanStressCorrection	None	The method used to correct the damage calculation for mean stress
InterpolationLimit	UseMaxCurve	Multicurve material interpolation limit
CertaintyOfSurvival	50	Required confidence level on damage results
ScaleFactor	1	The scale factor to apply prior to damage calculation
OutputMaxMin	False	Whether to output max and min stresses
OutputMaterialNames	False	Whether to output material names to the results
OutputDangerFactor	False	Whether to output danger factor
TargetLife	1E6	Target life for safety factor calculation
MaxSafetyFactor	100	Limits the calculated safety factor to a maximum
FactorOfSafetyType	ConstantMean	Method for calculating Haigh based safety factor.
MaxPrincipalPlaneStress	MostPositive	Use most positive or largest absolute value
DutyCycle		
EventProcessing	Independent	How to process separate events in duty cycles
OutputEventResults	False	Whether to output results per event or not for duty cycle processing
Advanced		
CheckStaticFailure	Warn	The action to take on static failure
StressGradients		
StressGradients	Auto	Controls whether stress gradient corrections are done
StressGradientsUser		The name of the file that contains the stress gradient parameters

5.14.1 Summary

The summary describes the basic operation of the Safety Factor Analysis Engine.

There are several combination methods used to calculate stress based safety factors. This implementation employs a method used in both the rail, and the road vehicle industries where the stress is resolved to the plane of the maximum absolute principal stress (*MaxPrincipalPlane*), along with standard stress combination methods such as *AbsMaxPrincipal* and *CriticalPlane*.

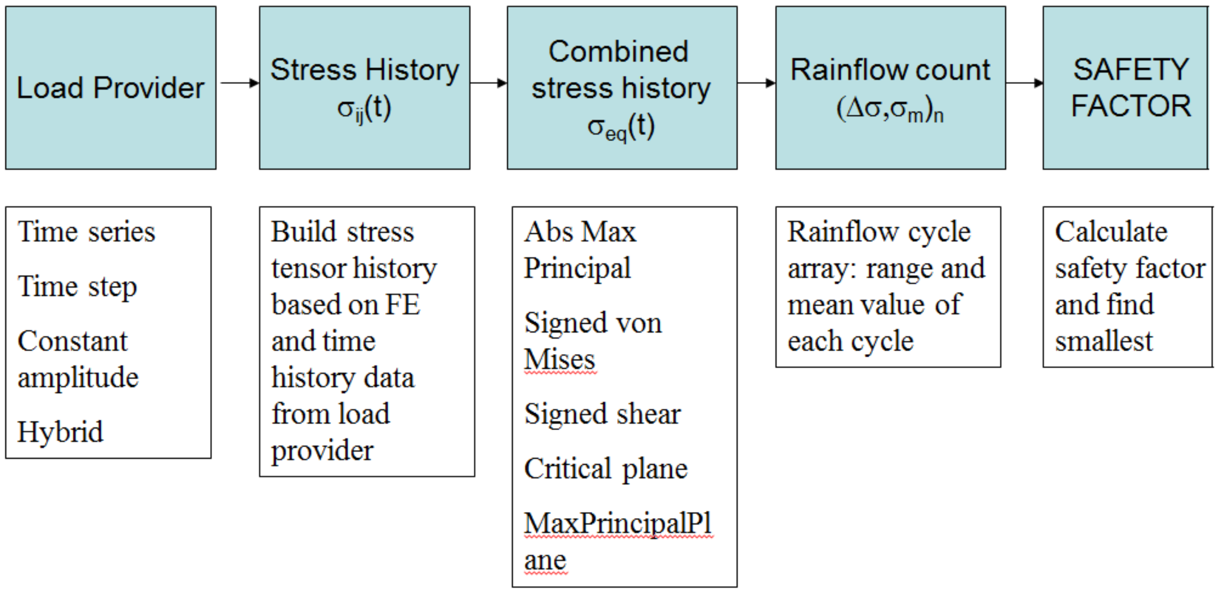
- Inputs will be linear stress or strain (it is an SN based technique)
- Material input will be standard SN or Haigh multi-curve

The user selects a target life, and then the stress based safety factor is calculated based on:

- The stress to achieve the target life
- Using the Haigh curve for that life

Note *InterpolationLimit* is not used for Haigh curves. When a material has two or more Haigh curves, material values are always extrapolated. For more details see "Haigh" on page 103 in *5.3.4 SNMethod*.

Fig. 5-197 Safety Factor Analysis Engine workflow



5.14.2 Properties of Safety Factor Analysis Engine

The properties of the Safety Factor analysis engine (see Fig. 5-196 Safety Factor Analysis Engine Properties) are:

Safety Factor Method

Several different SN based safety factor methods are supported. Each one uses a different method for mapping the relationship between stress amplitude and the corresponding safety factor. Each method requires use of the appropriate type of material data. The options available and the corresponding material data set type keywords are as follows:

- Standard (material data set keyword: *nCodeSNMatData*): This method uses a single parameterised stress-life curve.
  The curve is in multiple sections, typically changing below 1000 cycles and above a user-specified transition life. Mean stress corrections can be applied using modification methods such as Goodman.
- MultiMeanCurve (material data set keyword: *nCodeSNMeanStressCurveContainer*): When this option is selected, the Safety Factor analysis engine expects the S-N data in the form of a family of S-N curves representing the fatigue strength of the material at different mean stress levels.
  Details of this type of data can be found in the *Standard SN Analysis Engine*. The Safety Factor engine requires the multi mean curve data set to contain a 0 mean stress curve which in then uses to calculate the reference stress.
- MultiRRatiCurve (material data set keyword: *nCodeSNRRatioCurveContainer*): When this option is selected, the Safety

Factor analysis engine expects the S-N data in the form of a family of S-N curves representing the fatigue strength of the material at different R ratio levels.
Details of this type of data can be found in the *Standard SN Analysis Engine*. The Safety Factor engine requires the multi R Ratio curve data set to contain a R Ratio = -1 curve, which it then uses to calculate the reference stress.

- Haigh (material data set keyword: *nCodeSNHaighCurveContainer*): This method uses multiple stress amplitude versus mean stress curves, each one corresponding to a particular life.
  A single curve is calculated for the required target life and the safety factor is derived by comparing the stress amplitude and mean to this curve. If a material with a single Haigh curve is selected then this is used for the calculation regardless of the target life.

Combination Method

The stresses and strains at a node or element are stored in *component* form. For uniaxial analysis, it is necessary to combine these into an equivalent *scalar* value.

The available methods are:

- AbsMaxPrincipal
- SignedVonMises
- SignedShear
- CriticalPlane
- MaxPrincipalPlane

The *CriticalPlane* option resolves the data to a series of angles, and performs a fatigue calculation on each of these planes.

The *MaxPrincipalPlane* option finds the plane of the largest *Abs Maximum Principal Stress* and then resolves the stress to this single plane. This selection can be made based on either the most positive or the largest absolute value of the *Abs. Max. Principal* stress and this is controlled by the property *MaxPrincipalplaneStress*.

When the *MaxPrincipalPlane* combination method is selected for *3D stress results* three planes will be selected based on the maximum *Max Principal stress* the maximum *Mid Principal stress* and the minimum *Min Principal stress*. The stress is then resolved to all three planes, and the safety factor calculated for each.

InterpolationLimit

This specifies how to get material values from a multicurve material from positions above the maximum curve. The options are:

UseMaxCurve: Find a position on the maximum curve

Extrapolate: Extrapolate to a position above the maximum curve using the top two curves.

Notes: *InterpolationLimit* is not used for Haigh curves. When a material has two or more Haigh curves, material values are always extrapolated. For more details see "Haigh" on page 103 in *5.3.4 SNMethod*.

InterpolationLimit does not apply to interpolation between curves at different temperatures for a temperature corrected calculation. See the separate property *TemperatureInterpolationLimit*

Mean Stress Correction

The main feature of a strain cycle that affects fatigue damage is its range. Fatigue damage is also influenced by the mean stress of each cycle. Mean stress correction methods allow the effect of mean stress to be modeled and taken into account in the life prediction. For further information see the "Standard SN Analysis Engine" on page 84. The mean stress methods supported are:

- None
- Goodman
- Gerber
- FKM
- GoodmanTensionOnly
- GerberTensionOnly
- Interpolate (Haigh method only)

Table 5-14 Allowable S-NFactorOFSafetyMethod/MeanStressCorrection Combinations

		SNFactorOfSafety Method			
		Standard	**MultiMean curve**	**MultiRRatio Curve**	**Haigh**
Mean Stress Correction	None	✓	✓*	✓*	*X*
	Goodman	✓	✓*	✓*	*X*
	Gerber	✓	✓*	✓*	*X*
	Interpolate	*X*	*X*	*X*	✓
	FKM	✓	✓*	✓*	*X*
	Goodman TensionOny	✓	✓*	✓*	*X*
	Gerber TensionOnly	✓	✓*	✓*	*X*

✓ = Allowed
X = Not allowed
✓* = Allowed but a curve for RR=-1 or zero mean stress must be present.

CertaintyOfSurvival

The certainty of survival allows statistical variations in material behaviour to be taken into account. It works in the same way as for Standard S-N calculations (see the "Certainty of Survival" in the Standard SN Analysis Engine). The default value (50%) will result in the mean life being predicted.

ScaleFactor

This scale factor is applied to the stresses before the fatigue calculation. The default value is 1.0.

OutputMaxMin

When this property is set to *True*, the maximum and minimum values of the combined stress, as passed into the rainflow counter, will be reported in the results.

OutputMaterialNames

When this property is set to *True*, the material names will be reported in the results.

OutputDangerFactor

This property determines whether to output danger factor results. The DangerFactor is calculated from the SafetyFactor:

DangerFactor = (1 / SafetyFactor) - 1

When the SafetyFactor is equal to 1 the DangerFactor will be 0 and as the stress gets larger the DangerFactor will get larger. Therefore negative DangerFactors are "safer" and positive DangerFactors and more "dangerous". The maximum value is set to 1000.

TargetLife

This is the target life that is used to calculate the allowed stress value that is used in the safety factor calculation. When *Haigh* curves are used the TargetLife is used to interpolate a single curve for the required life. If a single Haigh curve is selected and the life does not match the TargetLife, then this will error.

MaxSafetyFactor

For convenience when viewing and plotting results, this property allows an upper limit to be set on the Safety Factor.

FactorOFSafetyType

For Haigh curves the safety factor can be calculated based on a constant R Ratio or a constant mean stress. The calculation of the safety factor is based on the distance the mean stress/stress amplitude is from the Haigh constant life line.

The options are:

Table 5-15 Safety factor options

ConstantMean	The distance is calculated by starting from the zero stress amplitude axis and going vertically through the point to intersect the line keeping the same mean stress.
ConstantRRatio	A line is drawn from the origin through the stress point and extended until it intersects the Haigh curve.
ConstantMinimum	The distance is calculated by starting from the zero stress amplitude point and going vertically through the point to intersect the line keeping the same minimum stress on a plot of maximum vs. minimum cycle stress.
ConstantMaximum	The distance is calculated by starting from the zero stress amplitude point and going vertically through the point to intersect the line keeping the same maximum stress on a plot of minimum vs. maximum cycle stress.

MaxPrincipalPlaneStress

This property controls how the CombinationMethod-MaxPrincipalPlane operates. The method finds the plane of the largest Abs. Maximum Principal stress and this option determines if the angle is selected based on the most positive or the largest Absolute value of Absolute Maximum Principal Stress.

EventProcessing

This property controls the way that the Safety Factor engine handles duty cycles. The options are:

Table 5-16 Event processing options

Independent	Dang Van safety factor calculations are made for each event separately. Note that because of the nature of the calculations, it is only necessary to consider each unique event in the duty cycle once. The repeat count is ignored for Dang Van calculations. The overall result is the worst safety factor taken from the individual events.
CombinedFull	Dang Van safety factor calculations are made for the complete duty cycle. Note that because of the nature of the Dang Van calculation, neither the number of repeats nor the exact sequence of events should have any significant effect on the final result (in practice they may have a minor effect on the shakedown part of the calculation).

OutputEventResults

This can be set to either *True* or *False*. It controls the output of results when processing duty cycles. It determines whether or not individual event results are output as well as the overall result. When using the *CombinedFull* option, this should be set to False.

CheckStaticFailure

Static failure checking compares the maximum value of the *CombinedStress* (as cycle counted) with the material's UTS. It also compares the Equivalent Stress Amplitude (after any mean stress correction) with the UTS. There is currently no criterion applied for compressive failure. There are three options:

Table 5-17 Check static failure options

Warn	The calculation continues, but a warning message is issued. The safety factor for this entity is also assigned a numeric value of 0.
Stop	The analysis job stops with a message.
False	No static failure check is made. Safety factor is calculated by the standard method.

StressGradients

This property has three options::

Table 5-18 Stress gradient options

Auto	A correction for stress gradient is applied if stress gradients are present (i.e., if they were requested at the translation stage in the Analysis Group properties).
User	A user-defined stress gradient correction is applied (if stress gradients are available) based on a lookup table provided in a file. A valid file must be specified using the StressGradientsUser property.
Off	No stress gradient correction is applied.

The stress gradient correction works in exactly the same way as for the Standard S-N Analysis engine, with the correction factor being applied to the stress before stress combination, cycle extraction, safety factor calculation, etc.

StressGradientsUser

If StressGradients is set to User, the name of a valid file must be provided here (see Standard S-N Analysis Engine for more information).

StressGradientMethod

See 5.3.26 StressGradientMethod in “Standard SN Analysis Engine”.

OutputStressGradientFactors

See 5.3.27 OutputStressGradientFactors in “Standard SN Analysis Engine”.

5.14.3 Details of Safety Factor Analysis Methods

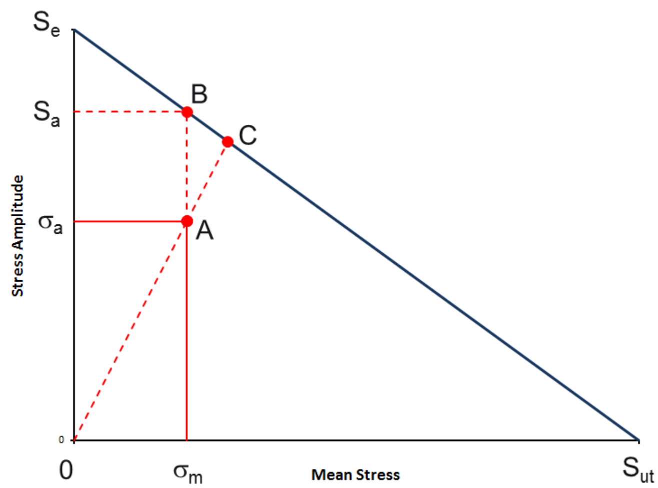
The stress-based safety factor method calculates a safety factor relative to a reference stress which will typically be the endurance limit of the material, taking into account the mean stress. It calculates a safety factor for each stress cycle in the loading history, and returns the lowest safety factor.

The endurance limit used for this calculation is calculated off of the materials stress life curve, after all the user specified corrections have been applied. These corrections include surface treatment factor, user surface factor, surface roughness type, and certainty of survival. A user specified target life is used to evaluate the corrected SN curve, and determine the stress amplitude associated with the target life to use as the endurance limit.

The definition of safety factor using the Goodman correction is illustrated below.

The stress cycle is considered, with amplitude σ_a and mean σ_m. The safety factor can be calculated either as constant mean or as constant R-ratio. A factor less than 1 is considered unsafe, and a factor greater than 1 acceptable.

Fig. 5-198 Safety factor using Goodman correction



Constant Mean Stress factor of safety

The mean stress due to things such as manufacturing operations or assembly exists prior to loading. This residual mean is significantly higher than the mean stresses due to the applied loads, and dominates the solution.

In this case, the stress amplitude is scaled, keeping the mean stress constant. The resulting factor of safety is calculated as Sa/σ_a.

Constant R-ratio factor of safety

The structure is unstressed prior to loading. The mean stress is entirely due to the applied loading, and hence the change in stress amplitude is proportional to the change in mean stress.

The factor of safety is calculated as $|\overrightarrow{0C}|/|\overrightarrow{0A}|$, which is the ratio of the length of the vector from the origin to point C by the length of the vector from the origin to point A.

If the applied loading consists of a single cycle, the scale factor from a SN backcalculation is the same as the constant R-ratio stress based factor of safety.

Constant Minimum Stress factor of safety

In this case, the stress amplitude is scaled, keeping the cycle minimum stress constant. The resulting factor of safety is calculated in an analogous way to the Constant mean stress method but based on a plot of cycle minimum stress vs. cycle maximum stress.

Constant Maximum Stress factor of safety

In this case, the stress amplitude is scaled, keeping the cycle maximum stress constant. The resulting factor of safety is calculated in an analogous way to the Constant mean stress method but based on a plot of cycle minimum stress vs. cycle maximum stress.

Goodman

See Fig. 5-198 Safety factor using Goodman correction for the plot.

SN Curve R-ratio=-1, Constant mean stress

$$S_a = \sigma_a / \left(1 - \frac{\sigma_m}{S_{ut}}\right)$$

$$SF = \frac{S_e}{S_a}$$

SN Curve R-ratio=-1, Constant R-ratio

$$SF = 1 / \left(\frac{\sigma_a}{S_e} + \frac{\sigma_m}{S_{ut}}\right)$$

It is possible in the compressive region to have the R-ratio line not intersect the Goodman constant life curve. In this case the factor of safety is set to the user specified maximum for the safety factor. Using the equation above, this condition exhibits itself be returning a negative factor of safety.

if $SF < 0$

$$SF = SFmax$$

SN Curve R-ratio<>-1, Constant mean stress

$$Sa = \sigma_a S_{ut} / (S_{ut} - \sigma_m + \sigma_a (1 + RR)/(1 - RR))$$

$$SF = \frac{S_{e\_RR}}{S_a}$$

SN Curve R-ratio<>-1, Constant R-ratio

$$S_{m\_RR} = S_{e\_RR} (1 + RR) / (1 - RR)$$

$$S_e = S_{e\_RR} / \left(1 - \frac{S_{m\_RR}}{S_{ut}}\right)$$

$$SF = 1 / \left(\frac{\sigma_a}{S_e} + \frac{\sigma_m}{S_{ut}}\right)$$

Constant Minimum and Maximum Stress factor of safety

Calculations are analogous to the Constant mean stress method based on a plot of cycle minimum stress vs. cycle maximum stress.

Goodman tension only

For Goodman tension only, any cycle with a mean stress less than zero is calculated as if it has a zero mean stress.

SN Curve R-ratio=-1, Constant mean stress

If $\sigma_m > 0$

$$S_a = \sigma_a / \left(1 - \frac{\sigma_m}{S_{ut}}\right)$$

$$SF = \frac{S_e}{S_a}$$

else

$$SF = \frac{S_e}{\sigma_a}$$

SN Curve R-ratio=-1, Constant R-ratio

If $\sigma_m > 0$

$$SF = 1 / \left(\frac{\sigma_a}{S_e} + \frac{\sigma_m}{S_{ut}}\right)$$

else

$$SF = \frac{S_e}{\sigma_a}$$

SN Curve R-ratio<>-1, Constant mean stress

If $\sigma_m > 0$

$$Sa = \sigma_a S_{ut} / (S_{ut} - \sigma_m + \sigma_a (1 + RR)/(1 - RR))$$

$$SF = \frac{S_{e\_rr}}{S_a}$$

else

$$S_{m\_RR} = S_{e\_RR} (1 + RR)/(1 - RR)$$

$$S_e = S_{e\_RR}\Big/\left(1 - \frac{S_{m\_RR}}{S_{ut}}\right)$$

$$SF = \frac{S_e}{\sigma_a}$$

SN Curve R-ratio<>-1, Constant R-ratio

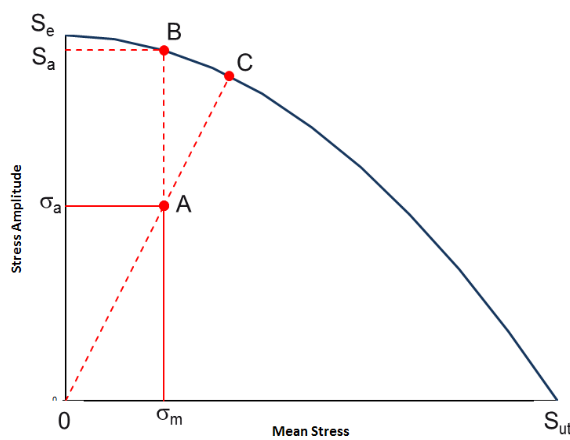
$$S_{m\_RR} \;=\; S_{e\_RR}\,(1 + RR)/(1 - RR)$$

$$S_e = S_{e\_RR}\Big/\left(1 - \frac{S_{m\_RR}}{S_{ut}}\right)$$

$$SF = \frac{S_e}{\sigma_a}$$

Gerber

Fig. 5-199 Safety factor using Gerber stress amplitude vs mean stress curve



SN Curve R-ratio=-1, Constant mean stress

if $\sigma_m > 0.0$

$$S_a \;=\; \sigma_a\Big/\left(1 - \left(\frac{\sigma_m}{S_{ut}}\right)^2\right)$$

else

$$S_a = \sigma_a / \left(1 + \left(\frac{\sigma_m}{S_{ut}}\right)^2\right)$$

$$SF = \frac{S_e}{S_a}$$

SN Curve R-ratio=-1, Constant R-ratio

It is possible in the compressive region to have the R-ratio line not intersect the Gerber constant life curve. In this case the factor of safety is set to the user specified maximum for the safety factor. Using the equation , this condition exhibits itself be returning a negative value for the term inside the square root.

If $\sigma_m = 0.0$

$$SF = \frac{S_e}{\sigma_a}$$

else if $\sigma_m > 0.0$

$$SF = \frac{1}{2}\left(\frac{S_{ut}}{\sigma_m}\right)^2 \frac{\sigma_a}{S_e}\left[-1 + \sqrt{1 + \left(\frac{2\sigma_m S_e}{S_{ut}\sigma_a}\right)^2}\right]$$

else

$$val = 1 - \left(\frac{2\sigma_m S_e}{S_{ut}\sigma_a}\right)^2$$

If $val < 0.0$

$$SF = SFmax$$

else

$$SF = -\frac{1}{2}\left(\frac{S_{ut}}{\sigma_m}\right)^2 \frac{\sigma_a}{S_e}\left[-1 + \sqrt{val}\right]$$

SN Curve R-ratio<>-1, Constant mean stress

If $\sigma_m > 0.0$

$$S_{eq} = \sigma_a / \left(1 - \left(\frac{\sigma_m}{S_{ut}}\right)^2\right)$$

else

$$S_{eq} = \sigma_a/\left(1 + \left(\frac{\sigma_m}{S_{ut}}\right)^2\right)$$

If RR > -1

$$val = 1 + \frac{4S_{eq}^2(1+RR)^2}{\left(S_{ut}^2(1-RR)^2\right)}$$

If *val* < 0.0

$$SF = SFmax$$

else

$$S_a = \left[-1 + \sqrt{val}\right]\frac{(1-RR)^2 S_{ut}^2}{2S_{eq}(1+RR)^2}$$

$$SF = \frac{S_e}{S_a}$$

else

$$val = 1 - \frac{4S_{eq}^2(1+RR)^2}{\left(S_{ut}^2(1-RR)^2\right)}$$

If *val* < 0.0

$$SF = SFmax$$

else

$$S_a = \left[-1 + \sqrt{val}\right]\frac{S_{ut}^2(1-RR)^2}{-2S_{eq}(1+RR)^2}$$

$$SF = \frac{S_e}{S_a}$$

SN Curve R-ratio<>-1, Constant R-ratio

$$S_{m\_RR} = S_{e\_RR}(1+RR)/(1-RR)$$

If $\sigma_m > 0$

$$S_e = S_{e\_RR}/\left(1 - \left(\frac{S_{m\_RR}}{S_{ut}}\right)^2\right)$$

else

$$S_e = S_{e\_RR} \Big/ \left(1 + \left(\frac{S_{m\_RR}}{S_{ut}}\right)^2\right)$$

$$SF = -\frac{1}{2}\left(\frac{S_{ut}}{\sigma_m}\right)^2 \frac{\sigma_a}{S_e}\left[-1 + \sqrt{1 + 4(\sigma_m S_e / S_{ut}\,\sigma_a)^2}\,\right]$$

Gerber tension only

SN Curve R-ratio=-1, Constant mean stress

If $\sigma_m > 0.0$

$$S_a = \frac{\sigma_a}{\left(1 - \left(\frac{\sigma_m}{S_{ut}}\right)^2\right)}$$

$$SF = \frac{S_e}{S_a}$$

else

$$SF = \frac{S_e}{\sigma_a}$$

SN Curve R-ratio=-1, Constant R-ratio

If $\sigma_m > 0.0$

$$SF = \frac{1}{2}\left(\frac{S_{ut}}{\sigma_m}\right)^2 \frac{\sigma_a}{S_e}\left[-1 + \sqrt{1 + \left(\frac{2\sigma_m S_e}{S_{ut}\sigma_a}\right)^2}\,\right]$$

else

$$SF = \frac{S_e}{\sigma_a}$$

SN Curve R-ratio<>-1, Constant mean stress

If $\sigma_m > 0.0$

$$S_{eq} = \sigma_a \Big/ \left(1 - \left(\frac{\sigma_m}{S_{ut}}\right)^2\right)$$

else

$$S_{eq} = \sigma_a$$

If RR > -1

$$val = 1 + \frac{4S_{eq}^2(1+RR)^2}{(S_{ut}^2(1-RR)^2)}$$

If *val* < 0.0

$$SF = SFmax$$

else

$$S_a = [-1 + \sqrt{val}]\frac{S_{ut}^2(1-RR)^2}{2S_{eq}(1+RR)^2}$$

$$SF = \frac{S_e}{S_a}$$

else

$$val = 1 - \frac{4S_{eq}^2(1+RR)^2}{(S_{ut}^2(1-RR)^2)}$$

If *val* < 0.0

$$SF = SFmax$$

else

$$S_a = [-1 + \sqrt{val}]\frac{S_{ut}^2(1-RR)^2}{-2S_{eq}(1+RR)^2}$$

$$SF = \frac{S_e}{S_a}$$

SN Curve R-ratio<>-1, Constant R-ratio

$$S_{m\_RR} = S_{e\_RR}(1+RR)/(1-RR)$$

If $\sigma_m > 0$

$$S_e = S_{e\_RR}/\left(1 - \left(\frac{S_{m\_RR}}{S_{ut}}\right)^2\right)$$

$$SF = -\frac{1}{2}\left(\frac{S_{ut}}{\sigma_m}\right)^2 \frac{\sigma_a}{S_e}\left[-1+\sqrt{1 + 4(\sigma_m S_e / S_{ut}\,\sigma_a)^2}\,\right]$$

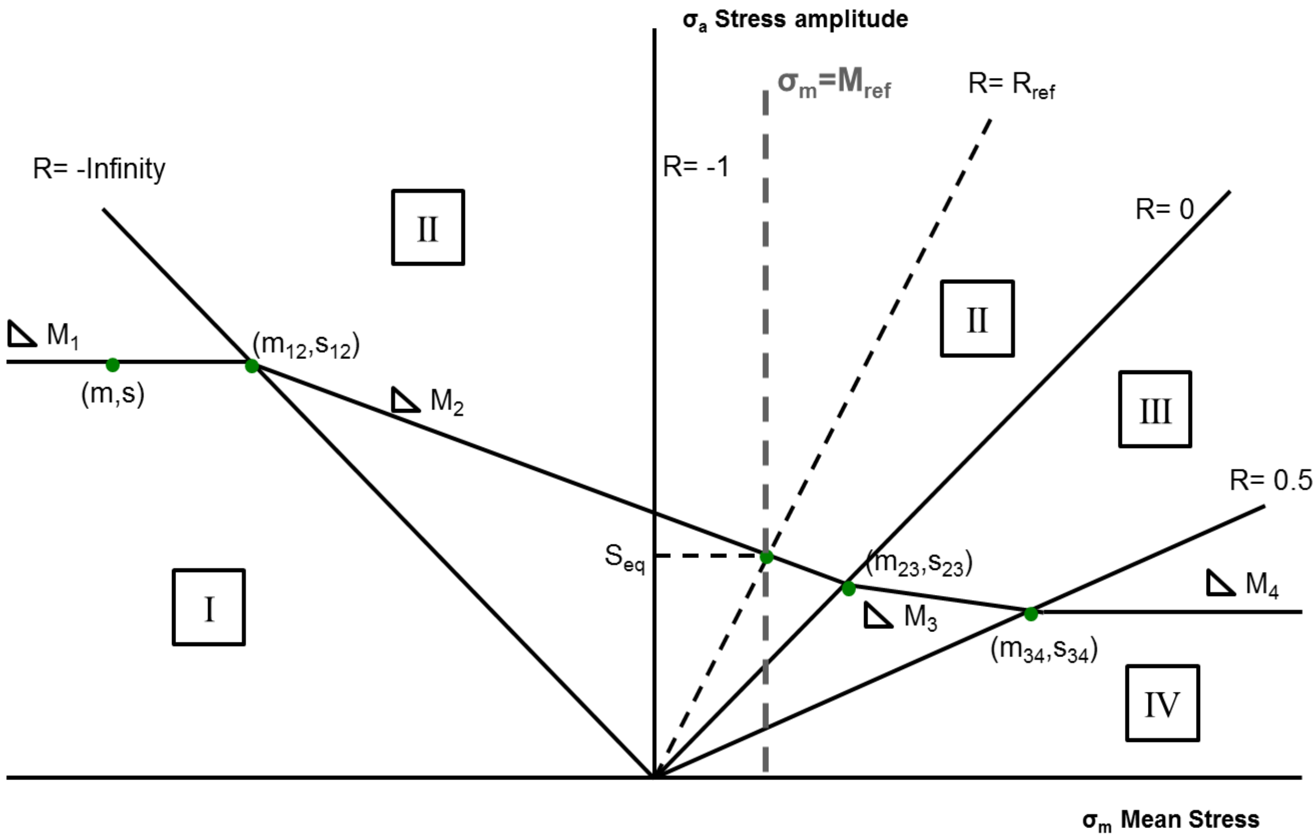
else

$$S_e = S_{e\_RR} \Big/ \left(1 + \left(\frac{S_{m\_RR}}{S_{ut}}\right)^2\right)$$

$$SF = \frac{S_e}{\sigma_a}$$

FKM

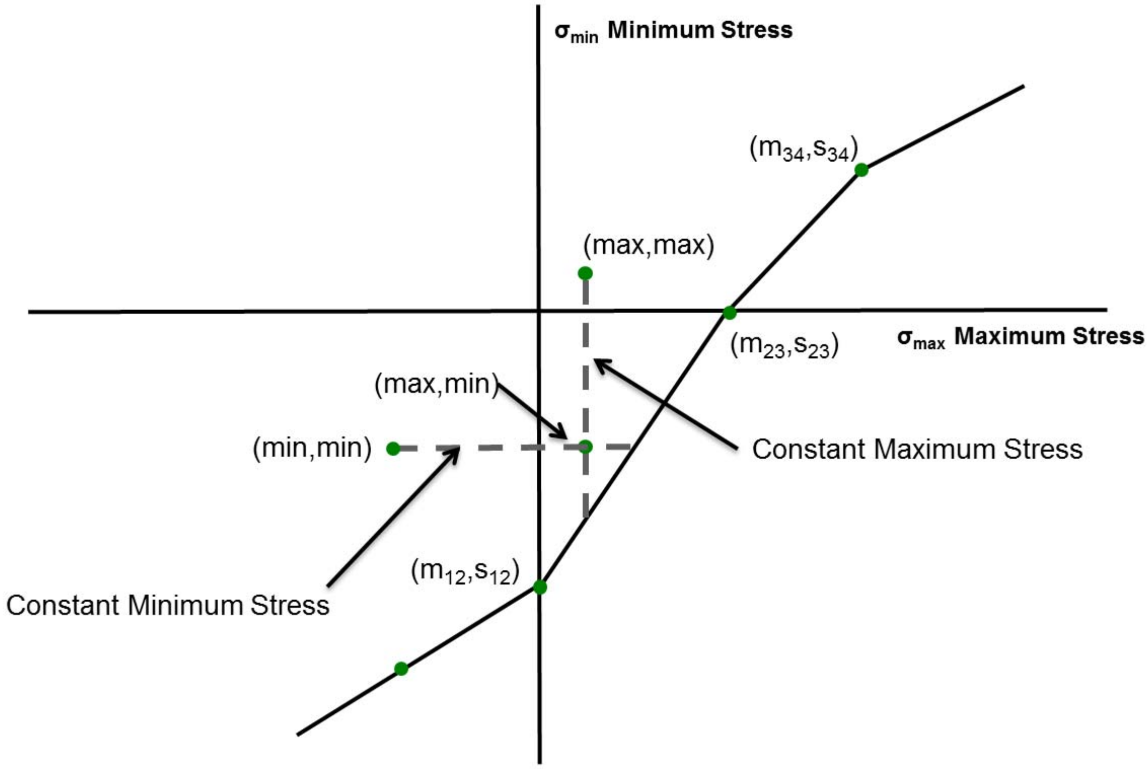
Fig. 5-200 FKM Constant Life diagram



A *Constant Life* diagram defined by the FKM Guideline "*Analytical Strength Assessment of Components in Mechanical Engineering*", Tr. E. Haibach. 2003, is used to convert the cycle's stress amplitude into an equivalent stress amplitude.

In order to calculate an equivalent stress for the Constant Maximum and Constant Minimum options, the Constant Life diagram should be plotted in terms of the Maximum and Minimum stress.

Fig. 5-201 Constant Life Diagram in terms of Maximum and Minimum Stress



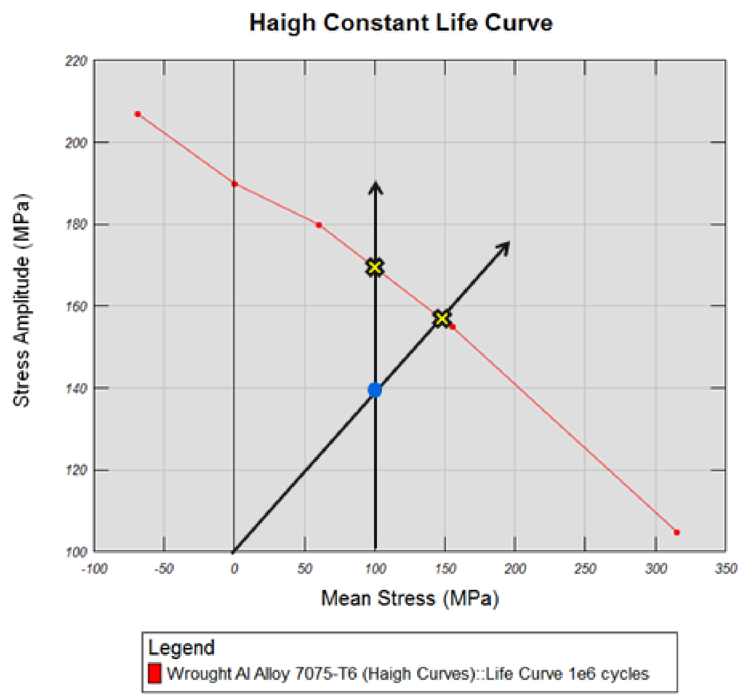
Four options are available for the calculation:

- *Constant R-ratio*: At a particular material R-ratio, R_{ref} the constant life diagram is used to calculate an equivalent stress amplitude. This equivalent amplitude is used with the endurance limit S_e at the R-ratio, R_{ref} to calculate the constant R-ratio safety factor.
- *Constant Mean*: At a particular material mean stress, M_{ref} the constant life diagram is used to calculate an equivalent stress amplitude. This equivalent amplitude is used with the endurance limit S_e at the mean stress, M_{ref} to calculate the constant mean stress safety factor.
- *Constant Maximum:* At a particular material maximum stress, max, the constant life diagram (Maximum vs. Minimum stress) is used to calculate an equivalent stress amplitude. This is taken from the (max,max) point to the (max,min). This equivalent amplitude is used with the endurance limit S_e at the maximum stress, MAX_{ref} to calculate the constant maximum stress safety factor.
- *Constant Minimum:* At a particular material minimum stress, min, the constant life diagram (Maximum vs. Minimum stress) is used to calculate

an equivalent stress amplitude. This is taken from the (min, min) point to the (max,min). This equivalent amplitude is used with the endurance limit S_e and the minimum stress MIN_{ref} to calculate the minimum stress safety factor.

Haigh Constant Life Curve

Fig. 5-202 Haigh Constant Life curve plotted



A Haigh constant life curve defines combinations of mean stress and stress amplitude that have the same fatigue life. The Haigh curve consists of a set of point pairs of mean stress vs stress amplitude, with piecewise linear segments connecting the points. The factor of safety calculations uses a Haigh curve selected by the user specified target life. If a curve does not exist for that life value, an new curve will be calculated by interpolation of the input curves.

Constant mean stress

A straight line is determined that passes through the points defined by the stress cycle's mean stress (σ_m) and stress amplitude (σ_a), and the stress cycle's mean stress and a zero stress amplitude. This straight line is used to calculate an intersection with each line segment defining the Haigh curve. When an intersection point is found its location is used as the mean corrected endurance limit (Se).

The factor of safety is calculated as:

$$SF = \frac{S_e}{\sigma_a}$$

Constant R-ratio

A straight line is determined that passes through the points defined by the origin, and the stress cycle's mean stress (σ_m) and stress amplitude (σ_a). This straight line is used to calculate an intersection with each line segment defining the Haigh curve. When an intersection point is found, that location is used to calculate the target mean stress (Sm) and stress amplitude (Sa). The factor of safety is calculated as

$$SF = \frac{\sqrt{S_a^2 + S_m^2}}{\sqrt{\sigma_a^2 + \sigma_m^2}}$$

Constant Minimum and Maximum Stress factor of safety

Calculations are analogous to the Constant mean stress method based on a plot of cycle minimum stress vs. cycle maximum stress.

5.14.4 Outputs of a Safety Factor Analysis

The outputs of a safety factor analysis are as follows:

Safety factor

This is defined in the sections above.

Danger Factor

DangerFactor = (1 / SafetyFactor) – 1

When the safety factor is equal to 1 the danger factor will be zero and as the applied stress gets larger so will the danger factor. Therefore negative danger factors are “safer” and positive danger factors are more “dangerous”.

The maximum possible value is set to 1000.

Maximum/Minimum Stress

The maximum and minimum values of the combined stress.

Plane Angle Phi and Plane Angle Theta

When the *MaximumPrincipalPlane* combination option is selected the stress is resolved either to a single plane (2D stress) of three planes (3D stress). These are the angles the selected planes make with the X and Z axes. For the 2D stress case only Phi is calculated.

5.15 Strain Energy Analysis Engine

5.15.1 Introduction

Fig. 5-203 Strain Energy analysis engine properties

Object Name: AnalysisEngineStrainEnergy1 (Strain Energy analysis engine)

Name	Value	Description
General		
WNMethod	PowerLaw	The method used to calculate damage from the strain energy of a cycle
CombinationMethod	ProportionalisedStress	The method used to combine component stresses/strains
MeanStressCorrection	None	The method used to correct the damage calculation for mean stress
MultiAxialAssessment	None	Whether to perform assessment of the multi-axial stress state
CertaintyOfSurvival	50	Required confidence level on damage results
ScaleFactor	1	The scale factor to apply prior to damage calculation
OutputMaxMin	False	Whether to output the strain energy, W parameter, the max and min stresses and strains
OutputMaterialNames	False	Whether to output material names to the results
OutputDistributedSource	False	Whether to output details of the distributed process that generated each result
DutyCycle		
EventProcessing	Independent	How to process separate events in duty cycles
OutputEventResults	False	Whether to output results per event or not for duty cycle processing
Advanced		
CheckStaticFailure	Warn	The action to take on static failure
DamageFloor	0	The calculated damage value below which the damage is set to zero
MaxDamage	1E30	The maximum damage value
StaticFailureDamage	1E10	The damage value to insert on static failure

5.15.2 Engine Operation Summary

The basic operation of the strain energy analysis engine can be summarized in Figures 5-204, 5-205, and 5-206, depending on the options used. The engine requires that both stresses and strains are recovered from the finite element analysis results file.

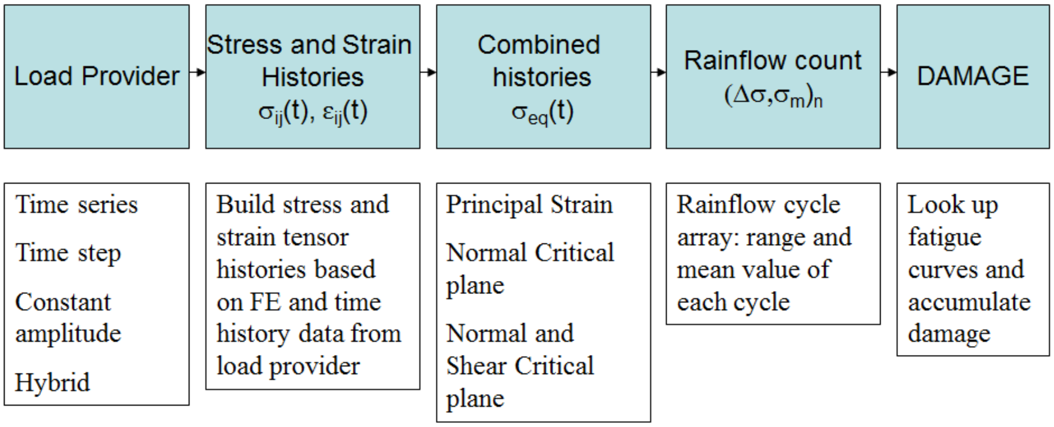
A typical strain energy analysis has the following basic steps, which are executed for each analysis entity (Node, Element, Node-on-Element), shell surface and critical plane where appropriate.

1. The stress and strain tensor histories, as a function of time or data points, are assembled from the information provided in the load provider. This applies to time-series, time step, constant amplitude and hybrid load providers.

   See the section on load providers (“Loading” on page 29) for details. *Vibration Load Providers* are not currently supported for WN calculations.
2. From the stress or strain tensor, extract a combined parameter for cycle counting. Possible cycle counting parameters include a Proportionalised Stress (defined later), Principal Strain, Critical Plane Normal Strain and Critical Plane Shear Strain.
3. Rainflow count the combined stress or strain parameter stress history to obtain a list of rainflow cycles, each with a known range and mean value, and for each cycle, from the stress and strain histories calculate a parameter based on strain energy density.
4. Calculate and accumulate damage from the W-N curve definitions.

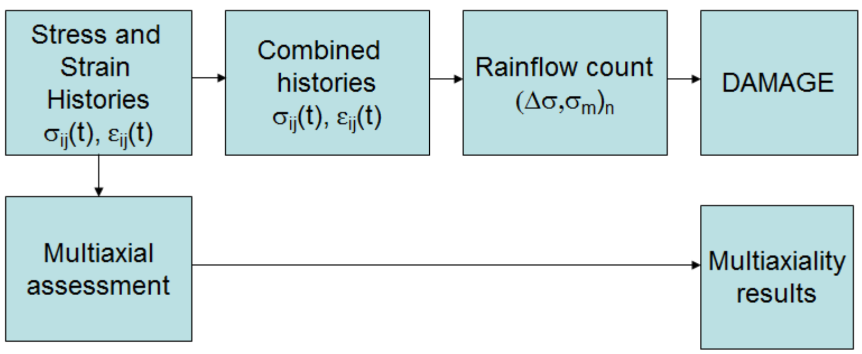
When no multiaxial assessment is to be performed, and the Proportionalised Stress combination method is not used, then the process operates as shown in Figure 5-204.

Fig. 5-204 Basis Strain Energy engine steps



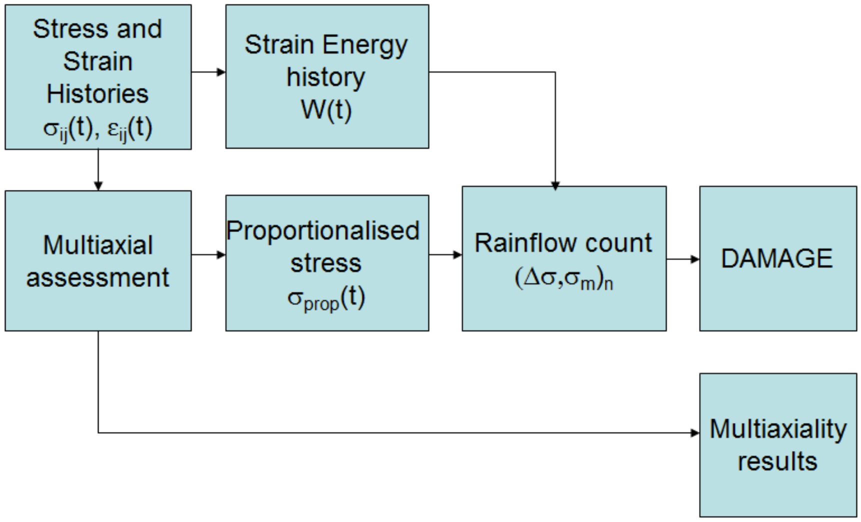
When the Multiaxial assessment is requested for these methods, the Multiaxial assessment is based on the stress tensor, but doesn't affect the predicted life, as illustrated in Figure 5-205.

Fig. 5-205 Strain Energy process with Multiaxial assessment



Now when the *Proportionalised Stress* method is used, a Multiaxial assessment is required and the process is modified as shown in Figure 5-206.

Fig. 5-206 Strain Energy process with Proportionalised Stress combination method



The stress history provides the Multiaxial assessment data to calculate a normalised averaged stress tensor, which is used to determine a stress history along this direction, which is then used in the cycle count.

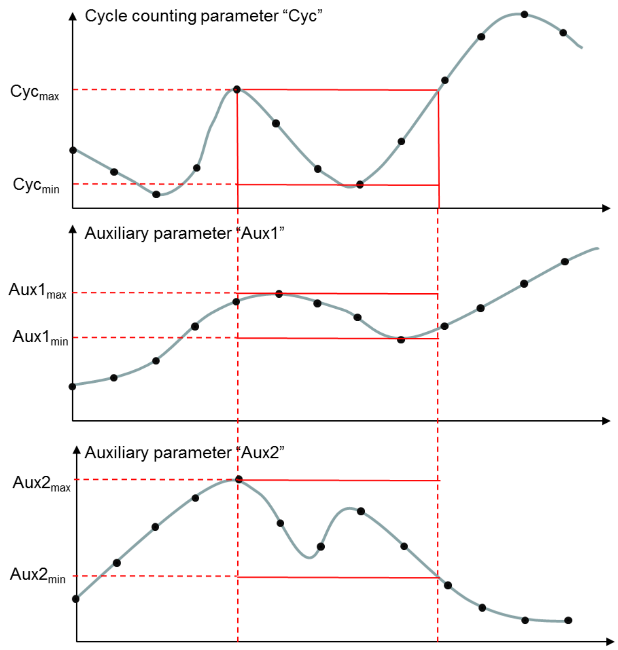
5.15.3 Rainflow Cycle Counting in Strain Energy Analysis

Rainflow cycle counting is based on the same principles as for the S-N method (see "Note on Rainflow Cycle Counting" on page 88), with a number of additions.

For each cycle, we identify the cycle open and close points, in addition to the maximum and minimum values, based on the primary cycle counting parameter. As the cycle close point will often be between points of the time history, a linear interpolation between adjacent points is normally required to determine the close point. These cycle open and close points allow the tracking parameters other than the primary cycle counting parameter during each cycle, and we can capture the maximum and minimum values of a number of auxiliary parameters, as is illustrated in Figure 5-207.

As for the main parameter, the value at the end point of each cycle is also determined by a linear interpolation, as it is quite likely that the auxiliary maximum or minimum will occur at this point.

Fig. 5-207 Enhanced rainflow cycle counting method



5.15.4 Properties

The analysis engine has the following properties:

WNMethod

The WNMethod property defines the method for calculating damage. Currently, there is only one method available:

PowerLaw

For this method, the change in strain energy density ΔW, is related to the damage by the power law $\Delta W = \kappa(N_f)^\alpha$

...for strain energy density intercept, K and fatigue strength exponent, α.

CombinationMethod

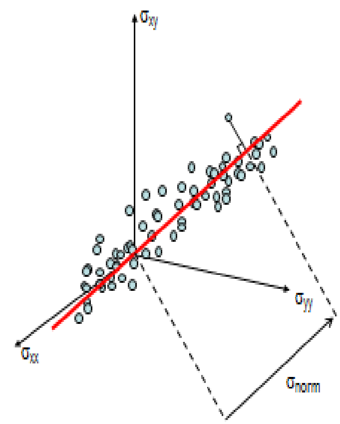
There are four options:

- ProportionalisedStress
- PrincipalStrain
- CriticalPlaneNormal
- CriticalPlaneNormalAndShear

ProportionalisedStress

This combination method first performs a multiaxial assessment on the data to determine a normalised average stress tensor $\sigma_{ij}^{normalised}$. In practice, loadings are frequently nearly proportional, so lie close a line through the origin in stress-space, as illustrated for the 2D case in Figure 5-208, where $\sigma_{ij}^{normalised}$ is represented by the red line.

Fig. 5-208 Strain Energy process with ProportionalisedStress method



The proportionalised stress, σ_{prop} at any instant is defined by the projection of the current stress onto the vector defined by the normalised averaged stress tensor and is determined from the dot product:

$$\sigma_{prop}(t) = \sigma_{ij}(t) \bullet \sigma_{ij}^{normalised}$$

The normalisation is carried out in such a way that for a proportional loading, the maximum value of the proportionalised stress is equal to the maximum value of the principal stress during the loading history.

This proportionalised stress is used as the primary rainflow cycle countingparameter.

Additionally, the time history of the total strain energy is calculated, and this becomes an auxiliary parameter in the rainflow cycle count.

$$W_{total}(t) = \frac{\varepsilon_{ij}(t) \cdot \sigma_{ij}(t)}{2}$$

In this case, the maximum value W_{max} is tracked for each cycle of σ_{prop}. The load ratio for each cycle is determined through:

$$R = \frac{\sigma_{prop,\min}}{\sigma_{prop,\max}}$$

Then the strain energy W-parameter is determined to be:

If $|R| > 1$ $$W = \frac{W_{\max}(1-R)^2}{4R^2}$$

Else $$W = \frac{W_{\max}(1-R)^2}{4}$$

At this point, a mean stress correction can be introduced if required, prior to the damage calculation by the W-N method. The *ProportionalisedStress* method is available for both 2-D and 3-D stress states.

PrincipalStrain

This combination method first determines the absolute maximum principal strain in the same manner as for the "Standard SN Analysis Engine" on page 84. The stress parallel to the largest principal strain direction is tracked as an auxiliary parameter. The total strain energy, as defined in the ProportionalisedStress method is also tracked.

Cycle counting is performed on the absolute maximum strain, whilst the maximum and minimum values of the stress are recorded.

These are used to determine the strain energy W-parameter as defined by

$$W = \frac{\Delta\varepsilon_{AMP}\Delta\sigma_{\|\varepsilon_{AMP}}}{8}$$ where $$\Delta\varepsilon_{AMP} = \varepsilon_{MAX} - \varepsilon_{MIN}$$ $$\Delta\sigma_{\|\varepsilon_{AMP}} = \sigma_{\|\varepsilon_{AMPMAX}} - \sigma_{\|\varepsilon_{AMPMIN}}$$

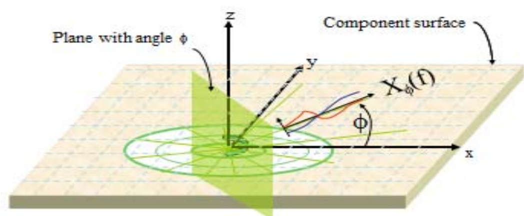
At this point, a mean stress correction can be introduced, prior to the damage calculation by the W-N method.

Note, this approach may not provide a particularly good measure of the damage when the system experiences pure shear/torsion loading.

CriticalPlaneNormal

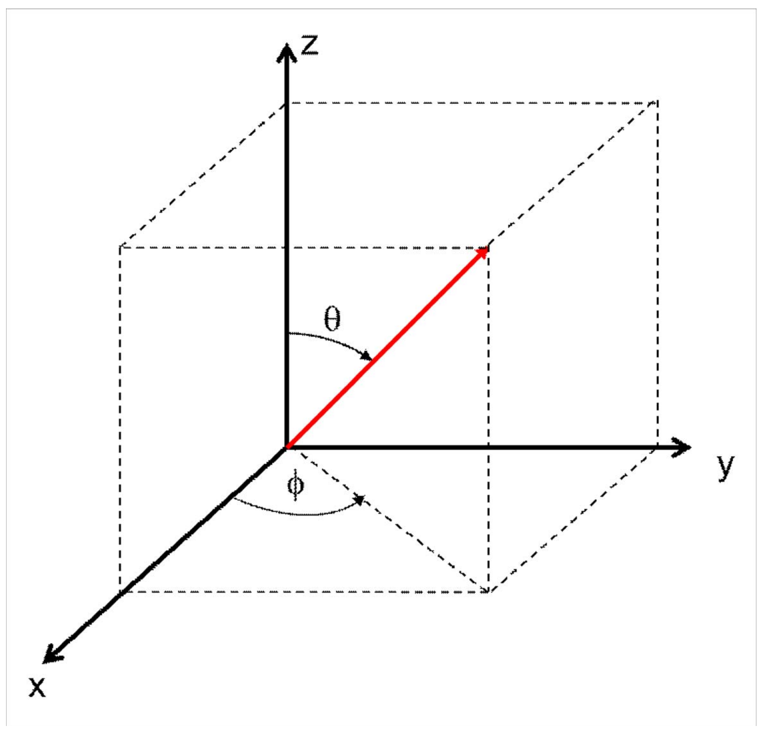
The *Critical Plane Normal* option is available for both 2D and 3D data. The critical plane is the plane with the most predicted fatigue damage. For the 2-D case, the planes on which the normal stress is determined have normals that lie in the plane of the physical surface, i.e., in the x-y plane of the 2-D stress results coordinate system. The orientation of each plane is defined by the angle ϕ made with the local x-axis, as indicated in Figure 5-209.

Fig. 5-209 Definition of the critical plane angle (2D Case)



In the 3D case 2 angles are required to define the orientation of the normal to the critical plane, as illustrated in Figure 5-210.

Fig. 5-210 Plane orientation angles illustrated relative to results co-ordinate system



For both 2-D and 3-D cases, a set of angles is chosen that gives approximately 10 degrees of separation between planes (18 planes for 2-D and 204 for 3-D).

The cycle counting parameter is the normal strain on the critical plane, and the normal stress is used as an auxiliary parameter. These values are calculated and rainflow counted on multiple planes.

Normal strain $\varepsilon_n = l \cdot \varepsilon \cdot l^T$ and normal stress $\sigma_n = l \cdot \sigma \cdot l^T$, (where l is the normal to the plane).

The total strain energy, as defined in the *Proportionalised Stress* method is also tracked. The strain energy density is determined from:

$$W = \frac{\Delta\sigma_n \cdot \Delta\varepsilon_n}{8} \quad \text{where} \quad \begin{aligned} \Delta\varepsilon_N &= \varepsilon_{N_{MAX}} - \varepsilon_{N_{MIN}} \\ \Delta\sigma_N &= \sigma_{N_{MAX}} - \sigma_{N_{MIN}} \end{aligned}$$

At this point, a mean stress correction effect can be re-introduced, before the damage calculation.

CriticalPlaneNormalAndShear

This critical plane option is only available for 2D data. It is based on the concept that damage on the plane is the result of a combination of normal and shear loading.

In this method, the shear stress, shear strain, normal stress and normal strain are tracked, where the normal stress and strain are defined as previously described, so that:

$$\sigma_n = \sigma_{xx}\cos^2\phi + 2\tau_{xy}\cos\phi\sin\phi + \sigma_{yy}\sin^2\phi$$

$$\varepsilon_n = \varepsilon_{xx}\cos^2\phi + 2\varepsilon_{xy}\cos\phi\sin\phi + \varepsilon_{yy}\sin^2\phi$$

While the shear stress is defined as:

$$\tau_n = \sin\phi\cos\phi\left(\sigma_{yy} - \sigma_{xx}\right) + \tau_{xy}\left(\cos^2\phi - \sin^2\phi\right),$$

...and the shear strain is given as:

$$\gamma_n = 2\left(\sin\phi\cos\phi\left(\varepsilon_{yy} - \varepsilon_{xx}\right) + \varepsilon_{xy}\left(\cos^2\phi - \sin^2\phi\right)\right)$$

...where the angle ϕ is defined as indicated in Figure 5-209.

For each cycle, the strain energy parameter, W is calculated from:

$$W = \frac{\Delta\sigma_n \cdot \Delta\varepsilon_n}{8} + \frac{\Delta\tau_n \cdot \Delta\gamma_n}{8}$$

...where:

$$\begin{aligned} \Delta\varepsilon_N &= \varepsilon_{N_{MAX}} - \varepsilon_{N_{MIN}} \\ \Delta\sigma_N &= \sigma_{N_{MAX}} - \sigma_{N_{MIN}} \end{aligned} \quad \text{and} \quad \begin{aligned} \Delta\tau_N &= \tau_{N_{MAX}} - \tau_{N_{MIN}} \\ \Delta\gamma_N &= \gamma_{N_{MAX}} - \gamma_{N_{MIN}} \end{aligned}$$

At this point, one can again introduce a mean stress correction effect, prior to the damage calculation. As for the *CriticalPlaneNormal* method, this approach makes calculations on many planes (18), with the predicted life being based on the most damaged plane.

MeanStressCorrection

For the *ProportionalisedStress*, *Principal Strain* and *CriticalPlaneNormal* combination methods, a *mean stress correction* is available that is analogous to a *Smith-Watson-Topper* correction, through the modification of the W-parameter, by the equation:

$$W_{eff} = fW_{par} \quad \text{where} \quad f = \frac{2}{1-R} \quad \text{or zero if} \quad R \geq 1,$$

Where $R = \frac{\sigma_{prop,\min}}{\sigma_{prop,\max}}$, for the Proportionalised stress method, while $\sigma_{\|\varepsilon}, \sigma_N$ are used for the PrincipalStrain and CriticalPlaneNormal methods, respectively.

For the CriticalPlaneNormalAndShear combination method, it is generally accepted that mean shear is not significant so the mean stress correction term is given by

$$W_{eff} = f\,\frac{\Delta\sigma_n \cdot \Delta\varepsilon_n}{8} + \frac{\Delta\tau_n \cdot \Delta\gamma_n}{8}$$

where $f = \frac{2}{1-R}$ or zero if $R \geq 1$, as for the other cases, and R is determined based on the normal stress.

MultiaxialAssessment

Multiaxial assessment provides information about how the stress state varies throughout the loading history.

There are two Multiaxial assessment options:

- None
- Standard

For the basic principles of Multiaxial assessment, please refer to the appropriate part of the section on the "Standard SN Analysis Engine" on page 84.
The basic principles of the Standard Method are the same as the S-N engine.

CertaintyOfSurvival

The *certainty of survival* allows statistical variations in material behaviour to be taken into account. It works in the same manner as for the "Standard SN Analysis Engine" on page 84. The default setting is 50% (mean life will be calculated). The calculation is based on the standard error of log(N) which must be set as a material parameter.

ScaleFactor

This scale factor will be applied to the stresses and strains before the fatigue calculation. The default value is 1.0.

OutputMaxMin

When this property is set to *True*, the maximum and minimum values of the combined stress (and where available combined strain), as passed into the rainflow counter, will be reported in the results. In addition the maximum value of the strain energy, and W-parameter are returned.

OutputMaterialNames

When this property is set to True, the material names will be reported in the results.

EventProcessing

A duty cycle is a sequence of events to be processed through the analysis engine. The creation and various different types of duty cycle are described in the section on Loading. Duty cycles may be processed in two different ways, which are basically the same as described in the section on the "Standard SN Analysis Engine" on page 84.

- **Independent** mode. Each unique event in the duty cycle is processed separately, and the damage from each is multiplied by the number of times it appears in the complete sequence. The total damage is the sum of the damage from all the different events. This is fast, but may miss significant cycles that cross events.
- **CombinedFull** mode. All the events in the duty cycle are concatenated, including all repeats of each event, and the resulting stress history is processed as if it were one long event. This is accurate, but may take a long time to process.

OutputEventResults

If this property is set to *True*, the accumulated damage (and other calculated results) associated with each individual event from the duty cycle will be output in addition to the total damage. If the *Combined* option is being used, some cycles may cross events. The damage associated with these cycles is split equally between the two events with which it is associated.

CheckStaticFailure

Static failure checking compares the maximum value of the *Combined Stress*, as cycle counted, with the material UTS. It also compares the *Equivalent Stress Amplitude* (after any mean stress correction) with the UTS. There is currently no criterion applied for compressive failure. There are three options:

- **Warn** —The calculation continues, but a warning message is issued. The damage for this entity is also assigned a numeric value equal to the value of the *StaticFailureDamage* property.
- **Stop** —The analysis job stops with a message.

- **False** — (Recommended) No static failure check is made. Damage is calculated up to the numerical limit defined by the *MaxDamage* property.

In the case of the *CriticalPlaneNormalandShear* combination method, the maximum of the normal and shear stresses are compared.

Note that the static failure check is simply a comparison of the maximum combined stress at each calculation point with the UTS at that point and orientation. It is an indicator of high stress, and a warning of unrealistic life predictions, but should not be interpreted as an accurate indication that the component is likely to experience static failure.

DamageFloor

This parameter sets a lower limit on calculated damage. If a cycle has less predicted damage, the damage value is set to zero. This parameter is somewhat redundant because similar functionality may be accessed using the fatigue cut-off, Nfc in the material definition.

MaxDamage

This parameter sets a numerical upper limit on the damage that can be predicted for each cycle.

StaticFailureDamage

This parameter sets a numerical value that may be set for damage to a particular calculation point in the event of the stress exceeding UTS. See "CheckStaticFailure" on page 404.

5.16 Custom Analysis Engine

5.16.1 Introduction

Fig. 5-211 Custom Analysis Engine properties page

Name	Value	Description
General		
MultiAxialAssessment	None	Whether to perform assessment of the multi-axial stress state
ScaleFactor	1	The scale factor to apply prior to damage calculation
OutputMaxMin	False	Whether to output the max and min stresses and strains
OutputMaterialNames	False	Whether to output material names to the results
OutputDistributedSource	False	Whether to output details of the distributed process that generated each result
OutputVibrationStats	False	Whether to output Vibration PSD parameters such as ExpectedZeroUpcrossings,
DutyCycle		
EventProcessing	Independent	How to process separate events in duty cycles
OutputEventResults	False	Whether to output results per event or not for duty cycle processing
Fatigue		
Method	Custom	The mode of operation for the engine
CustomEngineMethods	None	Specifies the customised method to use
CombinationMethod	Custom	The method used to combine component stresses/strains
CycleCounter	Standard	Specifies the cycle counting routine to use
DamageFloor	0	The calculated damage value below which the damage is set to zero
MaxDamage	1E30	The maximum damage value
CheckStaticFailure	False	The action to take on static failure
StaticFailureDamage	1E10	The damage value to insert on static failure
StandardCustomCombin		
CustomCombinationMethods	None	Specifies the customised combination method to use
CustomCombinationOutputs	1	Specifies the number of customised combination outputs to use

This engine allows the user to implement custom combination and damage methods, via python.

It can be used with any load provider that is appropriate for Stress, StressAnd-Strain or Vibration FE results.

A number of pre-defined combination methods are available, in addition to a custom combination method that returns a user specified number of outputs. These combinations can be passed into a cycle counter and the damage assessed from a custom routine.

Alternatively, the combination and cycle counting steps can be omitted entirely, and all aspects of the process can be specified by the user, and custom results column specified.

5.16.2 Engine Operation Summary

For most options the engine requires only stresses to be available from the finite element model. However, for certain finite element model inputs, it is advisable that orientation tensor data is also available. If both stress and strain data is available, this will be accessible for use with the custom combination and custom damage methods.

A typical analysis proceeds as follows:

1. The stress (and optionally strain) tensor histories, as a function of time or data points, are assembled from the information provided in the load provider. See the section on load providers for details.
2. (Optional) A combination is performed of the input data
3. (Optional) A multiaxial assessment is performed on the data.
4. (Optional) The combination data is cycle counted.
5. (Optional) A damage calculation is performed from a python script, based on either the raw input data, or the requested combinations

5.16.3 Material Properties

The Custom Analysis engine does not have its own material definition. It can use any of the existing material definitions, or the user can create their own.

5.16.4 Properties

The analysis engine has the following properties:

5.16.5 General Properties

The properties in this section are applicable to the engine regardless of the selected mode.

MultiaxialAssessment

Multiaxial assessment provides information about how the stress state varies throughout the loading history.

There are two options:

- None: It is not applied.
- Standard: For the basic principles of Multiaxial assessment, please refer to the appropriate part of the section on the "Standard SN Analysis Engine" on page 84. The basic principles of the Standard method are the same as in the SN engine.

ScaleFactor

This scale factor will be applied to the stresses and strains before the fatigue calculation. The default value is 1.0.

OutputMaxMin

When this property is set to True, the maximum and minimum values of the in-plane longitudinal, transverse and shear stress will be displayed. For 3d data, through thickness results will also be reported and where available, strain results will be available.

OutputMaterialNames

When this property is set to True, the material names will be reported in the results.

5.16.6 Duty Cycle Properties

The options are related to the configuration of duty cycles.

EventProcessing

A duty cycle is a sequence of events to be processed through the analysis engine. The creation and various different types of duty cycle are described in the section on Loading. Duty cycles may be processed in two different ways, which are basically the same as described in the section on the Standard SN analysis engine.

- **Independent** mode. Each unique event in the duty cycle is processed separately, and the damage from each is multiplied by the number of times it appears in the complete sequence. The total damage is the sum of the damage from all the different events. This is fast, but may miss significant cycles that cross events.
- **CombinedFull** mode. All the events in the duty cycle are concatenated, including all repeats of each event, and the resulting stress history is processed as if it were one long event. This is accurate, but may take a long time to process.

OutputEventResults

If this property is set to True, the accumulated damage (and other calculated results) associated with each individual event from the duty cycle will be output in addition to the total damage. If the "Combined" option is being used, some cycles may cross events. The damage associated with these cycles is split equally between the two events with which it is associated.

5.16.7 Fatigue Properties

CombinationMethod

There are 9 options available:

- None
- AbsMax
- CriticalPlane
- VonMises
- SignedVonMises
- PrincipleStrain
- CriticalPlaneNormal
- CriticalPlaneNormalAndShear

- Custom

None

The *None* combination method option is designed to be used with a custom damage method to allow the user to fully specify the entire analysis.

If this method is chosen in conjunction with *CycleCounter = None*, and *Method = None*, and a *time history* output is enabled, then this engine will output the stress (and strain) histories to allow for evaluation of the hysteresis loops.

Custom

The *Custom* combination method option allows the user to specify the parameter to be cycle counted, for use with the hardcoded damage methods. The chosen method should be specified through the CustomCombinationMethods option.

When used in conjunction with a custom damage method, the user is able to specify the combination method, and the damage calculation, while the cycle counting is handled by the software.

The options **AbsMax, CriticalPlane, VonMises, SignedVonMises** require **Cycle-Counter = Standard** to be chosen.

The options ***PrincipleStrain*, *CriticalPlaneNormal, CriticalPlaneNormalAnd-Shear*** require **CycleCounter = Extractor** to be chosen.

CycleCounter

There are 3 types of cycle counter currently available. These are:

- **None** – No cycle counting is performed.
- **Standard** – This is the standard cycle counter used in the SN engine. For more information, see the relevant section of the DesignLife theory guide.
  When multiple custom outputs are used, each combination will be cycle counter independently.
- **Extractor** – This is the cycle counter used by the strain energy engine. For more information, see the relevant section of the DesignLife theory guide.
  When multiple custom combination outputs are used, only the first output will be cycle counted, and the max/min values of all other parameters within that cycle will be tracked.

Method

- **None -** This method does no damage calculation. It allows the user to obtain the cycle list without requiring a damage calculation.
- **Custom -** The “Custom” option allows for a user-defined method. This option allows the user to customise the software using a python script. The script is used to define the relationship between the rainflow count and the damage, and may involve the use of custom material properties

and a user- defined mean stress correction. The chosen method should be specified through the CustomEngineMethods option.
Note; these custom methods are not CustomSNMethods.
(See the user guide for more details)

DamageFloor

This parameter sets a lower limit on calculated damage. If a cycle has less predicted damage, the damage value is set to zero. This parameter is somewhat redundant because similar functionality may be accessed using the fatigue cut-off, Nfc in the material definition.

MaxDamage

This parameter sets a numerical upper limit on the damage that can be predicted for each cycle.

CheckStaticFailure

Static failure checking compares the Stress in the local co-ordinate system to the desired failure criteria. There are three options:

- **Warn** - The calculation continues, but a warning message is issued. The damage for this entity is also assigned a numeric value equal to the value of the StaticFailureDamage property.
- **Stop** - The analysis job stops with a message.
- **False** - No static failure check is made. Damage is calculated up to the numerical limit defined by the MaxDamage property.

Note, the occurrence of static failure must be determined within the custom damage method script.

StaticFailureDamage

This parameter sets a numerical value that may be set for damage to a particular calculation point in the event of the stress exceeding UTS. See "CheckStaticFailure" above.

5.16.8 Custom Properties

CustomCombinationOutputs

This parameter specifies the number of combination outputs to be produced by a custom combination method.

When the standard cycle counter is used, each of these is treated independently.

When no cycle counter is used, then there will only ever be one line of output per location For *CycleCounter = Standard*, or *CycleCounter = Extractor*, then this defines the number of outputs, to be cycle counted.

CustomCombinationTracked

This option is only used when the extractor cycle counter is used.

For each result output, there are expected to be a number of tracked parameters (defined by CustomCombinationTracked). The cycle tracked parameters should be located at combination indices: 0, *CustomCombinationTracked*, 2* *Custom-CombinationTracked*,..., *CustomCombinationTracked *CustomCombinationOut-puts*.

The behaviour of these properties is best illustrated with an example: In order to perform a critical plane calculation with 18 planes, where the shear strain is cycle counted, and the maximum and minimum shear stress are tracked, the options would be set up as follows:

- *CustomCombinationOutputs = 18* (corresponding to the shear strain, and shear stresses)
- *CustomCombinationTracked = 2* (corresponding to the 18 critical planes)

Within the *custom combination* Python script, the outputs should be ordered as:

Table 5-19 Custom combination python script: outputs

Combination Index	Output
0	Shear Strain on first plane
1	Shear Stress on first plane
2	Shear Strain on second plane
3	Shear Stress on second plane
...	...
(LoopNumber-1)* CustomCombinationTracked Cycle Counted parameter	Cycle Counted parameter

5.17 Composite Analysis Engine

Fig. 5-212 Composite analysis engine properties: Mode = StaticFailure

Name	Value	Description
General		
MultiAxialAssessment	None	Whether to perform assessment of th
ScaleFactor	1	The scale factor to apply prior to dama
OutputMaxMin	True	Whether to output the max and min st
OutputMaterialNames	False	Whether to output material names to
OutputDistributedSource	False	Whether to output details of the distri
DutyCycle		
EventProcessing	Independent	How to process separate events in du
OutputEventResults	False	Whether to output results per event o
StaticFailure		
FailureCriteriaSource	Properties	Specifies the source of the failure crite
StaticDesignParameter	FailureIndex	How the failure criterion results are dis
OutputSeparateFailures	True	Whether to output the results of the i
FailureEnvelope		
FailureEnvelope	Conservative	How the failure envelope is defined
ConservativeFailureCriteri		
MaximumStressCriterion	True	Maximum Stress Criterion
MaximumStrainCriterion	True	Maximum Strain Criterion
FranklinMarinCriterion	False	Franklin-Marin Criterion
HoffmanCriterion	False	Hoffman Criterion
NorrisCriterion	False	Norris Criterion
NorrisMcKinnonCriterion	False	Norris-McKinnon Criterion
TsaiHillCriterion	False	Tsai-Hill Criterion
TsaiWuCriterion	False	Tsai-Wu Criterion
ChristensenCriterion	False	Christensen Criterion
HashinCriterion	False	Hashin Criterion
HashinRotemCriterion	True	Hashin-Rotem Criterion
HashinSunCriterion	False	Hashin-Sun Criterion
ModifiedNUCriterion	False	Modified NU Criterion
CustomCriterion	False	Custom Criterion
CustomFailureCriterionMethods	None	Specifies the customised failure Criteri

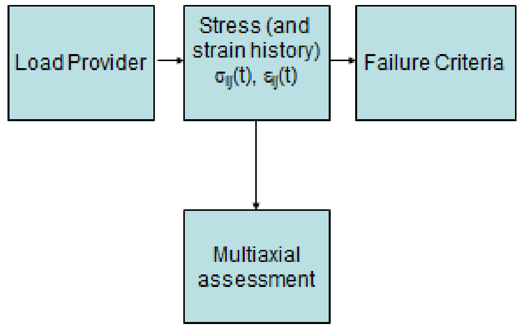
5.17.1 Engine Operation Summary

For most options the engine requires only stresses to be available from the finite element model. If both stress and strain data is available, this will be accessible for use with the custom failure criteria.

A typical analysis proceeds as:

1. The stress (and optionally strain) tensor histories, as a function of time or data points, are assembled from the information provided in the load provider. See the section on load providers for details.
2. (Optional) A multiaxial assessment is performed on the data.
3. The chosen failure criteria are evaluated.

Fig. 5-213 Operation of the Static Failure Composite Engine

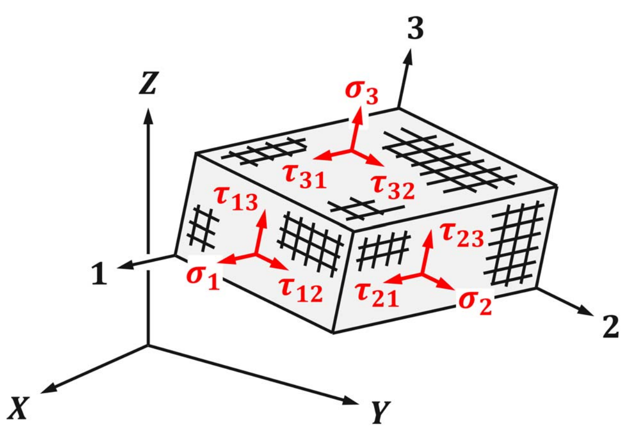


5.17.2 Material properties

The Composite Analysis engine uses its own material definition, which can be found in the MXD material database. These consist of a set of generic properties, that define the static material properties, and a child set of failure criteria properties.

Figure 5-214 is an overview of the 1-2-3 principal material axes used for defining orthotropic composite material properties along with the associated 3D stress tensor components.

Fig. 5-214 Overview of the 1-2-3 principal material axes...



$$S_i^c \le \sigma_i \le S_i^t$$
$$S_{ij} \le \tau_{ij} \le S_{ij}$$
$$(i, j = 1, 2, 3)$$

In the current version, all the properties must be defined in the 1-2-3 rectangular Cartesian principal material axes as depicted in Figure 5-214. It is important to note that these properties must be either orthotropic or transversely isotropic with respect to the 1-axis. They are detailed in Tables 5-20 and 5-21.

Table 5-20 Elastic, strength and ultimate strain orthotropic properties

Type	Property	Description
Young's moduli	E1	Young's modulus in the 1-direction
	E2	Young's modulus in the 2-direction
	E3	Young's modulus in the 3-direction
Shear moduli	G12	Shear modulus in the 1-2 plane
	G13	Shear modulus in the 1-3 plane
	G23	Shear modulus in the 2-3 plane
Poisson's ratios	Nu12	Major Poisson's ratio in the 1-2 plane
	Nu13	Major Poisson's ratio in the 1-3 plane
	Nu23	Major Poisson's ratio in the 2-3 plane
Strengths	S1t	Tensile strength in the 1-direction
	S2t	Tensile strength in the 2-direction
	S3t	Tensile strength in the 3-direction
	S1c	Compressive strength in the 1-direction (absolute value)
	S2c	Compressive strength in the 2-direction (absolute value)
	S3c	Compressive strength in the 3-direction (absolute value)
	S12	Shear strength in the 1-2 plane
	S13	Shear strength in the 1-3 plane
	S23	Shear strength in the 2-3 plane
Maximum strains and distortions	Epsilon1tu	Maximum tensile strain in the 1-direction
	Epsilon2tu	Maximum tensile strain in the 2-direction
	Epsilon3tu	Maximum tensile strain in the 3-direction
	Epsilon1cu	Maximum compressive strain in the 1-direction (absolute value)
	Epsilon2cu	Maximum compressive strain in the 2-direction (absolute value)
	Epsilon3cu	Maximum compressive strain in the 3-direction (absolute value)
	Gamma12u	Maximum distortion in the 1-2 plane
	Gamma13u	Maximum distortion in the 1-3 plane
	Gamma23u	Maximum distortion in the 2-3 plane

Table 5-21 Franklin-Marin, Hashin and Tsai-Wu parameters

Criterion	Parameters	Description
Franklin-Marin	FranklinMarin_tt_Sig1p	σ_{1p} coordinate of the third point in the tension-tension quadrant $(\sigma_{1p} > 0)$
	FranklinMarin_tt_Sig2p	σ_{2p} coordinate of the third point in the tension-tension quadrant $(\sigma_{2p} > 0)$
	FranklinMarin_ct_Sig1p	σ_{1p} coordinate of the third point in the compression-tension quadrant $(\sigma_{1p} < 0)$
	FranklinMarin_ct_Sig2p	$\sigma_{2(1)p}$ coordinate of the third point in the compression-tension quadrant $(\sigma_{2p} > 0)$
	FranklinMarin_cc_Sig2p	σ_{1p} coordinate of the third point in the compression-compression quadrant $(\sigma_{1p} < 0)$
	FranklinMarin_cc_Sig2p	σ_{2p} coordinate of the third point in the compression-compression quadrant $(\sigma_{2p} < 0)$
	FranklinMarin_tc_Sig2p	σ_{1p} coordinate of the third point in the tension-compression quadrant $(\sigma_{1p} > 0)$
	FranklinMarin_tc_Sig2p	σ_{2p} coordinate of the third point in the tension-compression quadrant $(\sigma_{2p} < 0)$
Hashin	Hashin_Alpha	Contribution, a, of the shear stress(es) to the fibre-dominated failure criterion $(0 \leq a \leq 1)$
Hashin-Sun	HashinSun_Mu	Enhancement parameter, μ, analogous to an internal friction coefficient when transverse compression occurs $\mu \geq 0$
Tsai-Wu	TsaiWu_F12_Star	Dimensionless interaction coefficient, $F_{12}*$, in the 1-2 plane $(-1 < F_{12}* < 1)$
	TsaiWu_F13_Star	Dimensionless interaction coefficient, $F_{13}*$, in the 1-2 plane $(-1 < F_{13}* < 1)$
	TsaiWu_F23_Star	Dimensionless interaction coefficient, $F_{23}*$, in the 1-2 plane $(-1 < F_{23}* < 1)$

Note The 3D orthotropic elastic properties must satisfy the following stability conditions [11]:

$$E_1, E_2, E_3, G_{12}, G_{13}, G_{23} > 0$$
$$|\nu_{12}| < (E_1/E_2)^{1/2}$$
$$|\nu_{13}| < (E_1/E_3)^{1/2}$$
$$|\nu_{23}| < (E_2/E_3)^{1/2}$$
$$(1-\nu_{12}\nu_{21}), (1-\nu_{13}\nu_{31}), (1-\nu_{23}\nu_{32}) > 0$$
$$1-\nu_{12}\nu_{21}-\nu_{13}\nu_{31}-\nu_{23}\nu_{32}-2\nu_{21}\nu_{32}\nu_{13} > 0 \text{ where } \nu_{ji} = (\nu_{ij}E_j)/E_i$$

5.17.3 Analysis Engine Properties

The properties in this section are applicable to the engine regardless of the selected mode.

MultiaxialAssessment

Multiaxial assessment provides information about how the stress state varies throughout the loading history.

There are two Multiaxial assessment options:

- None
- Standard

For the basic principles of Multiaxial assessment, please refer to the appropriate part of the section on the "Standard SN Analysis Engine" on page 84. The basic principles of the Standard method are the same as in the SN engine.

ScaleFactor

This scale factor will be applied to the stresses and strains before the fatigue calculation. The default value is 1.0.

OutputMaxMin

When this property is set to True, the maximum and minimum values of the in-plane longitudinal, transverse and shear stress will be displayed. For 3d data, through thickness results will also be reported and where available, strain results will be available.

OutputMaterialNames

When this property is set to True, the material names will be reported in the results.

The options are related to the configuration of duty cycles.

EventProcessing

A duty cycle is a sequence of events to be processed through the analysis engine. The creation and various different types of duty cycle are described in the section on Loading. Duty cycles may be processed in two different ways, which are basically the same as described in the section on the Standard SN analysis engine.

- **Independent** mode: each unique event in the duty cycle is processed separately, and the damage from each is multiplied by the number of times it appears in the complete sequence. The total damage is the sum of the damage from all the different events. This is fast, but may miss significant cycles that cross events.
- **CombinedFull** mode: all the events in the duty cycle are concatenated, including all repeats of each event, and the resulting stress history is processed as if it were one long event. This is accurate, but may take a long time to process.

OutputEventResults

If this property is set to True, the accumulated damage (and other calculated results) associated with each individual event from the duty cycle will be output in addition to the total damage. If the "Combined" option is being used, some cycles may cross events. The damage associated with these cycles is split equally between the two events with which it is associated.

These properties determine the failure criteria to be evaluated.

FailureCriteriaSource

There are two options for this. These are:

- **Properties**: the failure criteria are specified by the properties on the analysis engine. These criteria are applied across all materials on the model.
- **FromMaterialMap**: the failure criteria are specified via the MaterialMapLoader. The requested properties must be chosen as additional properties on that menu. In this case, the failure criteria can be specified on a per material basis.

StaticDesignParameter

There are four options to specify the form of the return output of the failure criteria. These are:

- **FailureIndex**: This is the value supplied by direct evaluation of the failure function defining the failure criterion.
- **StrengthRatio**: Local measure of the failure margin.
- **StressExposureFactor**: Local measure of the fracture risk.
- **MarginOfSafety**: StrengthRatio -1.

OutputSeparateFailures

When this property is set to true, the maximum returned value of each failure criterion is shown separately.

Failure Envelope

FailureEnvelope

There are two different methods of combining the failure criteria. These are:

- **Single**: Only one failure criterion is evaluated.
- **Conservative**: The worst case is always taken, so that if any criterion is violated then static failure will occur.

Single Failure Criteria

FailureCriterion

This property determines which failure criterion to use when only a single criterion is requested for the failure envelope.

Conservative Failure Criteria

MaximumStressCriterion

When this is set to true, the maximum stress criterion is evaluated.

MaximumStrainCriterion

When this is set to true, the maximum strain criterion is evaluated.

NorrisCriterion

When this is set to true, the Norris criterion is evaluated.

HoffmanCriterion

When this is set to true, the Hoffman criterion is evaluated.

TsaiHillCriterion

When this is set to true, the Tsai-Hill criterion is evaluated.

TsaiWuCriterion

When this is set to true, the Tsai-Wu criterion is evaluated.

FranklinMarinCriterion

When this is set to true, the Franklin-Marin criterion is evaluated. This criterion is only defined for plane stress conditions.

HashinCriterion

When this is set to true, the Hashin criterion is evaluated.

HashinRotemCriterion

When this is set to true, the Hashin-Rotem criterion is evaluated. This criterion is only defined for plane stress conditions.

ChristensenCriterion

When this is set to true, the Christensen criterion is evaluated.

NorrisMcKinnonCriterion

When this is set to true, the Norris-McKinnon criterion is evaluated. This criterion is valid for 2d materials only.

HashinSunCriterion

When this is set to true, the Hashin-Sun criterion is evaluated. This criterion is valid for 2d materials only.

ModifiedNUCriterion

When this is set to true, the NU criterion is evaluated. This criterion is valid for 2d materials only.

CustomCriterion

The *Custom* option allows for a user-defined composite method. This option allows the user to customise the software using a Python script. The script is used to define the failure criteria. It may involve the use of custom material properties.

Note If the static design parameter is set to Strength Ratio, or Margin of Safety, then the lowest value will be reported, otherwise the maximum will be returned. Beyond that it is down to the user to provide the appropriate definitions.

5.17.4 Strength Assessment

Overview

When used for structural applications, composite materials are generally subjected to complex loading conditions leading to a multiaxial state of stress. Predicting the loss of their load carrying capabilities is then achieved by using a mathematical function (i.e. criterion), F, combining the most damaging stress tensor or strain tensor components. F is generally defined under the following form:

$$F = \sum_i L_i + \sum_{i,j} Q_{ij}$$

where $\sum_i L_i$ and $\sum_{i,j} Q_{ij}$ are the sums of linear and quadratic stress/strain components respectively. By denoting $\{\sigma\}$ and $\{\varepsilon\}$, the vector representations (Voigt's notation) of the these tensors, failure is expected to occur when one of the following conditions is fulfilled:

$F(\{\sigma\}) \geq 1$ or $F(\{\sigma\}) \geq 1$

There are numerous failure theories in the literature and no general consensus has yet been found that establishes a unique and universal version [9]. This is why, several of them have been made available in the Composite Analysis Engine, see Table 5-22. It is the user's responsibility to select the most suitable theory based on their needs.

Table 5-22 Intra Laminar Failure Theories

Type	Failure theory	3D orthotropic	3D transversely isotropic	2D orthotropic	2D transversely isotropic	Supports Strain
Non interactive theories	Maximum stress	✓	✓	✓	✓	
	Maximum strain	✓	✓	✓	✓	✓
Fully interactive theories	Franklin-Marin			✓	✓	
	Hoffman	✓	✓	✓	✓	
	Norris	✓	✓	✓	✓	
	Norris-McKinnon			✓	✓	
	Tsai-Hill	✓	✓		✓	
	Tsai-Wu	✓	✓	✓	✓	
Partially interactive theories	Christensen		✓		✓	
	Hashin		✓		✓	
	Hashin-Rotem				✓	
	Hashin-Sun				✓	
	Nu				✓	
Custom	Python-based	✓	✓	✓	✓	✓

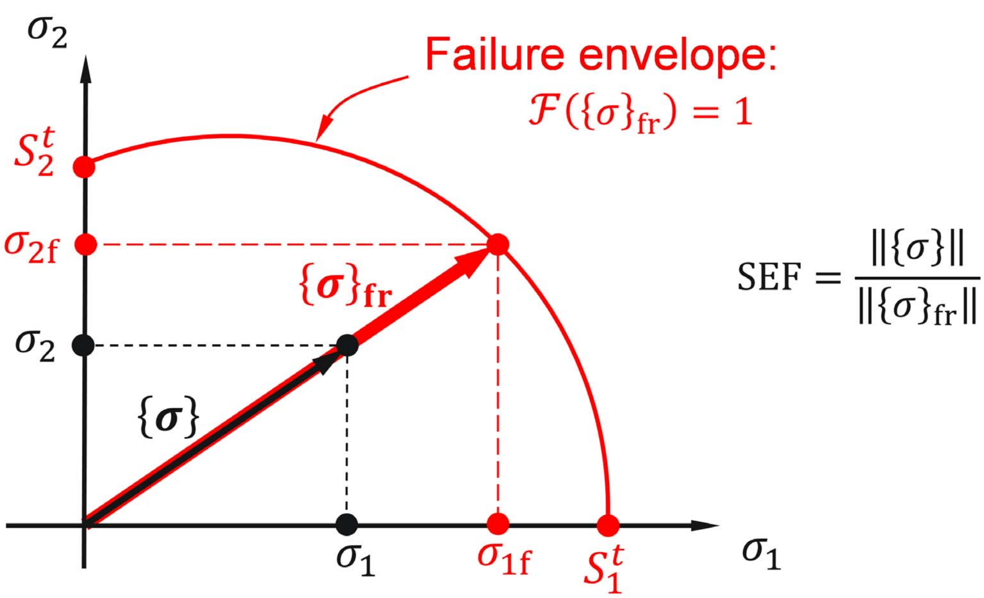
Engineering design parameters

For each failure theory, four different design parameters can be computed, see Table 5-23.

Table 5-23 Design Parameters f or each Failure Theory

Parameter	Acronym	Description
Failure index	FI	Directly evaluates the failure function F
Strength/stress ratio	SR	Provides a local measure of the margin to failure
Stress exposure factor	SEF	Measures the risk of fracture
Margin of safety	MoS	Indicates the margin to failure

Fig. 5-215 Definition of the stress exposure factor (SEF)



Failure Index (FI)

FI represents the value supplied by the direct evaluation of the failure function F introduced in section Overview.
Typical values are:

- $FI < 1$ Non-critical stresses or strains - failure has not occurred.
- $FI = 1$ $\{\sigma\}$ or $\{\varepsilon\}$ are just high enough to cause failure.
- $FI > 1$ $\{\sigma\}$ or $\{\varepsilon\}$ are too high - failure has occurred.

Note

- FI does not supply any quantitative information about the risk of fracture, i.e. tell design and stress engineers to which extent they can increase the load before fracture occurs.

Strength/stress ratio (SR)

SR is a local measure of the margin to failure. It is defined as [16]:

$$SR = \frac{\left(-\sum_i L_i + \sqrt{\left(\sum_i L_i\right)^2 + 4\sum_{i,j} Q_{ij}}\right)}{2\sum_{i,j} Q_{ij}}$$

...where $\sum_i L_i$ and $\sum_{i,j} Q_{ij}$ are the sums of linear and quadratic stress terms respectively. Typical SR values are:

- $SR > 1$ Positive margin to failure - failure has not occurred.
- $SR = 1$ Stresses are just high enough to cause failure.
- $SR < 1$ Negative margin to failure - failure has already occurred.

Graphically, SR can be interpreted as:

$$SR = \frac{\|\{\sigma\}_{fr}\|}{\|\{\sigma\}\|}$$

where $\|\{\sigma\}\|$ is the norm of the stress vector $\{\sigma\}$, and $\|\{\sigma\}_{fr}\|$ is the norm of the same vector extended up or down to the failure surface as shown in Fig. 5-215 for a basic two dimensional case.

Note

- When using this parameter, proportional loading is assumed.
- This parameter is sometimes called Stretch Factor.
- At the laminate (structural) level, SR is often called Reserve Factor (RF).

Stress Exposure Factor (SEF)

SEF provides a local measure of the risk of fracture. It is defined as [18]:

$$SEF = \frac{1}{2}\left(\sum_i L_i + \sqrt{\left(\sum_i L_i\right)^2 + 4\sum_{i,j} Q_{ij}}\right)$$

where $\sum_i L_i$ and $\sum_{i,j} Q_{ij}$ are the sums of linear and quadratic stress terms respectively. Typical SEF values are:

- $SEF < 1$ Non-critical stresses - failure has not occurred.
- $SEF = 1$ Stresses $\{\sigma\}$ are just high enough to cause failure.
- $SEF > 1$ Stresses $\{\sigma\}$ are too high - failure has already occurred.

Graphically, SEF can be interpreted as:

$$SEF = \frac{\|\{\sigma\}\|}{\|\{\sigma\}_{fr}\|}$$

where $\|\{\sigma\}\|$ is the norm of the stress vector $\{\sigma\}$, and $\|\{\sigma\}_{fr}\|$ is the norm of the same vector extended up or down to the failure surface as shown in Fig. 5-215 for a basic two dimensional case.

Note

- When using this parameter, proportional loading is assumed.
- At the laminate (structural) level, SEF is often called Inverse Reserve Factor (IRF).

Margin of Safety (MoS)

MoS indicates the margin to failure (usually in percent). It is defined as:

$$MoS = SR - 1$$

Typical values are:

- $MoS > 0$ Stresses $\{\sigma\}$ can be increased by a factor MoS before failure occurs.
- $MoS = 0$ Stresses $\{\sigma\}$ are just high enough to cause failure.
- $MoS < 0$ Indicates how much $\{\sigma\}$ should be decreased to reach the limit state.

Intra-laminar failure criteria

The next sub-sections detail the failure criteria listed in Table 5-22. To illustrate the shape of their envelopes, the lamina-level properties in Table 5-24 are used throughout. It is important to note that these do not represent a real material system but are only representative values arbitrary chosen for the sake of plotting failure envelopes in a meaningful manner.

Table 5-24 Failure Criteria Lamina-level Properties

Elastic properties	Strength properties
$E_1 = 120$ GPa	$S_1^t = 2000$ MPa
$E_2 = E_3 = 10$ GPa	$S_1^c = 1000$ MPa
$\nu_{12} = \nu_{13} = 0.3$	$S_2^t = S_3^t = 50$ MPa

Table 5-24 Failure Criteria (Continued)Lamina-level Properties

Elastic properties	Strength properties
$\nu_{23} = 0.5$	$S_2^c = S_3^c = 200$ MPa
$G_{12} = G_{13} = 5$ GPa	$S_{12} = S_{13} = 100$ MPa
	$S_{23} = 50$ MPa

Maximum stress criterion

Failure is expected to occur once one of the stress tensor components along the principal material axes exceeds the corresponding strength in the same direction. In 3D, the failure function F is defined as:

$$F = max(F_1, F_2, F_3, F_4, F_5, F_6)$$

...where:

$$F_1 = \begin{cases} \dfrac{\sigma_1}{S_1^t} \text{ if } (\sigma_1 \geq 0) \\ \dfrac{-\sigma_1}{S_1^c} \text{ if } (\sigma_1 < 0) \end{cases} \qquad F_4 = \frac{|\tau_{12}|}{S_{12}}$$

$$F_2 = \begin{cases} \dfrac{\sigma_2}{S_2^t} \text{ if } (\sigma_2 \geq 0) \\ \dfrac{-\sigma_2}{S_2^c} \text{ if } (\sigma_2 < 0) \end{cases} \qquad F_5 = \frac{|\tau_{23}|}{S_{23}}$$

$$F_3 = \begin{cases} \dfrac{\sigma_3}{S_3^t} \text{ if } (\sigma_3 \geq 0) \\ \dfrac{-\sigma_3}{S_3^c} \text{ if } (\sigma_3 < 0) \end{cases} \qquad F_6 = \frac{|\tau_{13}|}{S_{13}}$$

In 2D or for plane stress conditions, the stress function reduces to:

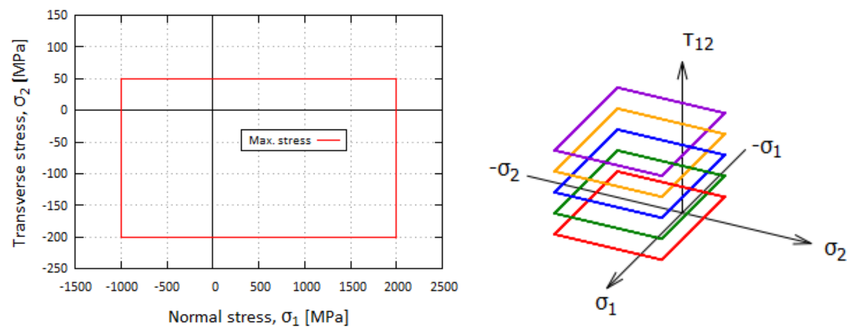
$$F = max(F_1, F_2, F_4) \geq 1$$

Figure 5-216 shows failure envelope produced by the 2D version of the maximum stress criterion in the σ_1-σ_2 stress space.

Figure 5-216a shows the failure envelope generated by the 2D version of this criterion. Figure 5-216b shows the same envelope for several positive shear stress values, τ_{12}, ranging 0–100% of the shear strength value, S_{12}.

Failure envelope produced by the 2D version of the maximum stress criterion in the σ_1 - σ_2 - τ_{12} stress space:

Fig. 5-216 2D version of the maximum stress criterion



a) In the σ_1-σ_2 strain space

(b) In the σ_1-σ_2-τ_{12} strain space

Maximum strain criterion

Failure is expected to occur once one of the strain tensor components defined in the principal material axes exceeds the corresponding ultimate value in the same direction. Two options can be used depending on the availability of the stress or strain tensor components from preliminary FE results.

Option 1: Strain-based version

When chosen, the standard definition is used. Therefore, the strain-based failure function F is defined in 3D as:

$$F = max(F_1, F_2, F_3, F_4, F_5, F_6)$$

...where:

$$F_1 = \begin{cases} \dfrac{\varepsilon_1}{\varepsilon_{1t}^u} \text{ if } (\varepsilon_1 \geq 0) \\ \dfrac{-\varepsilon_1}{\varepsilon_{1c}^u} \text{ if } (\varepsilon_1 < 0) \end{cases} \qquad F_4 = \frac{|\gamma_{12}|}{\gamma_{12}^u} = 2\frac{|\varepsilon_{12}|}{\gamma_{12}^u}$$

$$F_2 = \begin{cases} \dfrac{\varepsilon_2}{\varepsilon_{2t}^u} \text{ if } (\varepsilon_2 \geq 0) \\ \dfrac{-\varepsilon_2}{\varepsilon_{2c}^u} \text{ if } (\varepsilon_2 < 0) \end{cases} \qquad F_5 = \frac{|\gamma_{23}|}{\gamma_{23}^u} = 2\frac{|\varepsilon_{23}|}{\gamma_{23}^u}$$

$$F_3 = \begin{cases} \dfrac{\varepsilon_3}{\varepsilon_{3c}^u} \text{ if } (\varepsilon_3 \geq 0) \\ \dfrac{-\varepsilon_3}{\varepsilon_{3c}^u} \text{ if } (\varepsilon_3 < 0) \end{cases} \qquad F_6 = \frac{|\gamma_{13}|}{\gamma_{13}^u} = 2\frac{|\varepsilon_{13}|}{\gamma_{13}^u}$$

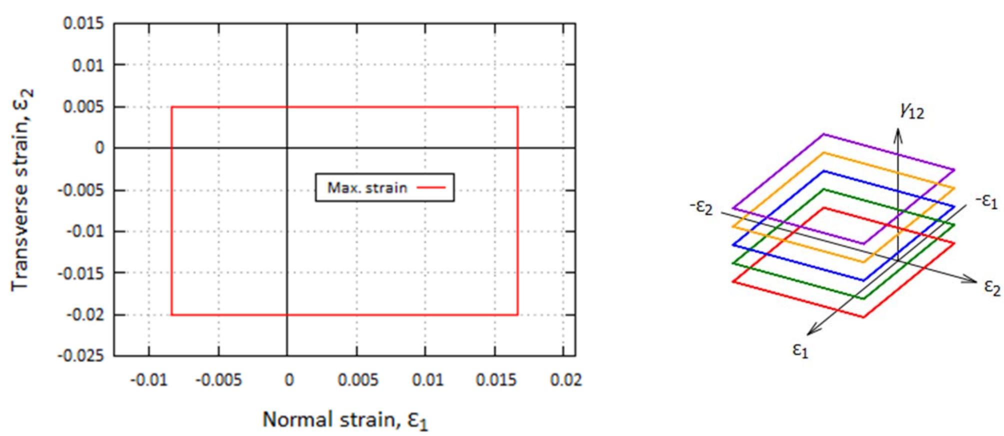
In 2D or for plane stress conditions, the failure function reduces to:

$$F = max(F_1, F_2, F_4)$$

Figure 5-217 shows failure envelopes produced by the 2D version of the maximum strain criterion in the strain space.

Figure 5-217a shows the failure envelope generated by the 2D version of this criterion, and Figure 5-217b shows the same envelope for several positive shear stress values, γ_{12}, ranging 0-100% of the shear strength value, γ_{12}^u.

Fig. 5-217 Failure envelopes: 2D version of maximum strain criterion (strain space)



a) In the ε_1-ε_2 strain space (b) In the ε_1-ε_2-γ_{12} strain space

Option 2: Stress-based version

If this option is chosen, the generalised Hooke's law for orthotropic materials is used to derive stress-based failure indices, F_i (i=1,...,6). In this context, a stress-based failure function, F, is defined as:

$$F=\max(F_1,F_2,F_3,F_4,F_5,F_6)$$

...where:

$$\mathcal{F}_1 = \begin{cases} [\sigma_1 - \nu_{12}\sigma_2 - \nu_{13}\sigma_3]/S_1^t & (\sigma_1 - \nu_{12}\sigma_2 - \nu_{13}\sigma_3 \geq 0) \\ -[\sigma_1 - \nu_{12}\sigma_2 - \nu_{13}\sigma_3]/S_1^c & (\sigma_1 - \nu_{12}\sigma_2 - \nu_{13}\sigma_3 < 0) \end{cases}$$

$$\mathcal{F}_2 = \begin{cases} [\sigma_2 - \nu_{21}\sigma_1 - \nu_{23}\sigma_3]/S_2^t & (\sigma_2 - \nu_{21}\sigma_1 - \nu_{23}\sigma_3 \geq 0) \\ -[\sigma_2 - \nu_{21}\sigma_1 - \nu_{23}\sigma_3]/S_2^c & (\sigma_2 - \nu_{21}\sigma_1 - \nu_{23}\sigma_3 < 0) \end{cases}$$

$$\mathcal{F}_3 = \begin{cases} [\sigma_3 - \nu_{31}\sigma_1 - \nu_{32}\sigma_2]/S_3^t & (\sigma_3 - \nu_{31}\sigma_1 - \nu_{32}\sigma_2 \geq 0) \\ -[\sigma_3 - \nu_{31}\sigma_1 - \nu_{32}\sigma_2]/S_3^c & (\sigma_3 - \nu_{31}\sigma_1 - \nu_{32}\sigma_2 < 0) \end{cases}$$

$$\mathcal{F}_4 = |\tau_{12}|/S_{12}$$

$$\mathcal{F}_5 = |\tau_{23}|/S_{23}$$

$$\mathcal{F}_6 = |\tau_{13}|/S_{13}$$

In 2D or for plane stress conditions, the failure function reduces to:

$$F=\max(F_1,F_2,F_4)$$

...where:

$$\mathcal{F}_1 = \begin{cases} [\sigma_1 - \nu_{12}\sigma_2]/S_1^t & (\sigma_1 - \nu_{12}\sigma_2 \geq 0) \\ -[\sigma_1 - \nu_{12}\sigma_2]/S_1^c & (\sigma_1 - \nu_{12}\sigma_2 < 0) \end{cases}$$

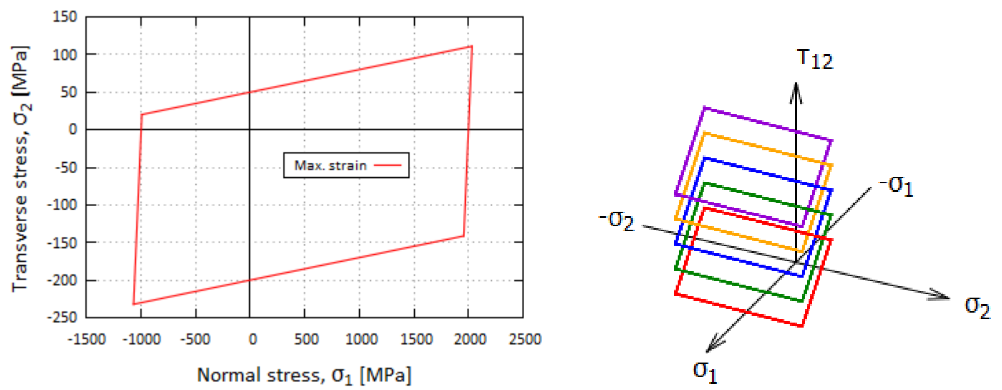
$$\mathcal{F}_2 = \begin{cases} [\sigma_2 - \nu_{21}\sigma_1]/S_2^t & (\sigma_2 - \nu_{21}\sigma_1 \geq 0) \\ -[\sigma_2 - \nu_{21}\sigma_1]/S_2^c & (\sigma_2 - \nu_{21}\sigma_1 < 0) \end{cases}$$

$$\mathcal{F}_4 = |\tau_{12}|/S_{12}$$

Figure 5-218 shows failure envelopes produced by the 2D version of the maximum strain criterion in the stress space.

Fig. 5-218a shows the failure envelopes generated by the 2D version of this criterion. In Fig. 5-218b, they are plotted for several positive shear stress values, τ_{12}, ranging 0-100% of the shear strength value, S_{12}.

Fig. 5-218 Failure envelopes: 2D version of maximum strain criterion (stress space)



(a) In the σ_1-σ_2 stress space (b) In the σ_1-σ_2-τ_{12} stress space

Franklin-Marin criterion (1968)

The *Franklin-Marin* theory, also referred to as the *modified Marin* theory, was introduced by Franklin [4]. He generalised Marin's theory [11] since the latter is restricted to having the principal stresses aligned with the principal material axes, which is too restrictive for composites under general loading conditions.

Franklin formulated his theory for orthotropic materials under plane stress conditions. Within this framework, the failure function is defined as:

$$\mathcal{F} = \frac{\sigma_1^2 - \bar{K}_2\sigma_1\sigma_2}{S_1^t S_1^c} + \frac{\sigma_2^2}{S_2^t S_2^c} + \left(\frac{\tau_{12}}{S_{12}}\right)^2 + \left(\frac{1}{S_1^t} - \frac{1}{S_1^c}\right)\sigma_1 + \left(\frac{1}{S_2^t} - \frac{1}{S_2^c}\right)\sigma_2$$

In the above equation, $\bar{K}_2$ is a "floating constant" evaluated from a predefined biaxial stress condition in each of the four quadrants. It is found by solving the above equation in each quadrant when considering a third point of coordinates (σ_1-σ_2-τ_{12})=(σ_{1p}-σ_{2p}-0) not belonging to any of the σ_1- and σ_2-axes. This gives:

$$\bar{K}_2 = \frac{S_{1t}S_{1c}}{\sigma_{1p}\sigma_{2p}}\left[\frac{\sigma_{1p}^2}{S_{1t}S_{1c}} + \frac{\sigma_{2p}^2}{S_{2t}S_{2c}} + \left(\frac{1}{S_{1t}} - \frac{1}{S_{1c}}\right)\sigma_{1p} + \left(\frac{1}{S_{2t}} - \frac{1}{S_{2c}}\right)\sigma_{2p} - 1\right]$$

Figure 5-219a shows the failure envelopes generated by this criterion when using the following third points shown in red in each quadrant:

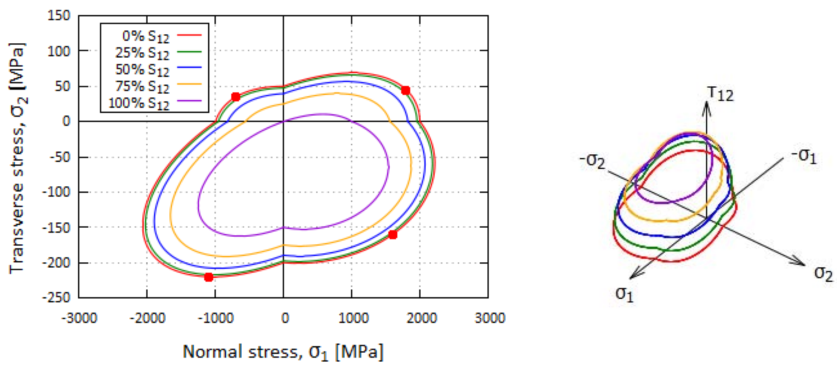
- (σ_{1p},σ_{2p})=(0.9 S_1^t , 0.9 S_2^t) in the tension-tension quadrant.
- (σ_{1p},σ_{2p})=(-0.7 S_1^c , 0.7 S_2^t) in the compression-tension quadrant.
- (σ_{1p},σ_{2p})=(-1.1 S_1^c , -1.1 S_2^c) in the compression-compression quadrant.
- (σ_{1p},σ_{2p})=(0.8 S_1^t , -0.8 S_2^c) in the tension-compression quadrant.

Figure 5-219b shows several positive shear stress values, τ_{12}, ranging from 0-100% of the shear strength value, S_{12}.

Note

- $\bar{K}_2$ must verify $\Delta = [1/S_2^t S_2^c - \bar{K}_2^2/(4S_1^t S_1^c)]/(S_1^t S_1^c) > 0$ in each quadrant for "stability" reasons.
- The plane stress version of the Hoffman criterion is found when $\bar{K}_2 = 1$ in each quadrant.
- The above third points are not real stress points. They were used to illustrate the distorting effect induced by the floating constant, $\bar{K}_2$, on the failure envelopes in Figure 5-219.

Fig. 5-219 Failure envelopes produced by the Franklin-Marin criterion



(a) In the σ_1-σ_2 stress space

(b) In the σ_1-σ_2-τ_{12} stress space

Hoffman criterion (1967)

The Hoffman theory is a fully interactive stress-based theory for composites. In 3D, its failure function is defined as [9]:

$$\mathcal{F} = C_1(\sigma_2 - \sigma_3)^2 + C_2(\sigma_3 - \sigma_1)^2 + C_3(\sigma_1 - \sigma_2)^2$$
$$+ C_4\sigma_1 + C_5\sigma_2 + C_6\sigma_3 + C_7\tau_{23}^2 + C_8\tau_{31}^2 + C_9\tau_{12}^2$$

...where:

$$C_1 = \frac{1}{2}\left(\frac{1}{S_2^t S_2^c} + \frac{1}{S_3^t S_3^c} - \frac{1}{S_1^t S_1^c}\right) \qquad C_4 = \frac{1}{S_1^t} - \frac{1}{S_1^c} \qquad C_7 = \frac{1}{S_{23}^2}$$

$$C_2 = \frac{1}{2}\left(\frac{1}{S_3^t S_3^c} + \frac{1}{S_1^t S_1^c} - \frac{1}{S_2^t S_2^c}\right) \qquad C_5 = \frac{1}{S_2^t} - \frac{1}{S_2^c} \qquad C_8 = \frac{1}{S_{31}^2}$$

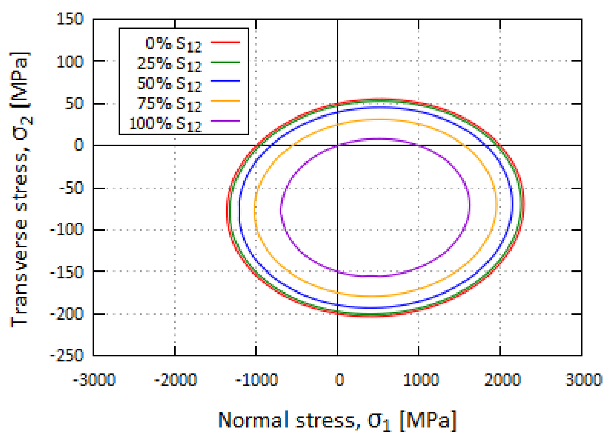
$$C_3 = \frac{1}{2}\left(\frac{1}{S_1^t S_1^c} + \frac{1}{S_2^t S_2^c} - \frac{1}{S_3^t S_3^c}\right) \qquad C_6 = \frac{1}{S_3^t} - \frac{1}{S_3^c} \qquad C_9 = \frac{1}{S_{12}^2}$$

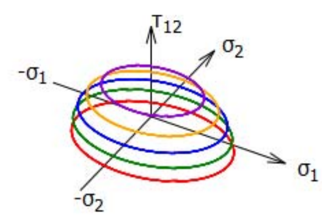
For orthotropic materials and plane stress conditions, the stress function reduces to:

$$\mathcal{F} = \frac{\sigma_1^2}{S_1^t S_1^c} - \left(\frac{1}{S_1^t S_1^c} + \frac{1}{S_2^t S_2^c} - \frac{1}{S_3^t S_3^c}\right)\sigma_1\sigma_2 + \frac{\sigma_2^2}{S_2^t S_2^c} + \left(\frac{1}{S_1^t} - \frac{1}{S_1^c}\right)\sigma_1 + \left(\frac{1}{S_2^t} - \frac{1}{S_2^c}\right)\sigma_2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2$$

Figure 5-220 shows the failure envelopes generated by the plane stress version of this criterion. They are plotted for several positive shear stress values, τ_{12}, ranging 0-100% of the shear strength value, S_{12}.

Fig. 5-220 Failure envelopes produced by the plane stress version of the Hoffman criterion





(a) In the σ_1-σ_2 stress space

(b) In the σ_1-σ_2-τ_{12} stress space

Norris criterion (1962)

The Norris failure theory is a stress-based formulation for 3D orthotropic materials derived from the Hencky-von Mises theory of energy due to change of shape [12]. The failure function is defined as:

$$\mathcal{F} = \max(\mathcal{F}_1, \mathcal{F}_2, \mathcal{F}_3)$$

...where:

$$\mathcal{F}_1 = \left(\frac{\sigma_1}{S_1}\right)^2 - \frac{\sigma_1\sigma_2}{S_1S_2} + \left(\frac{\sigma_2}{S_2}\right)^2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2$$

$$\mathcal{F}_2 = \left(\frac{\sigma_2}{S_2}\right)^2 - \frac{\sigma_2\sigma_3}{S_2S_3} + \left(\frac{\sigma_3}{S_3}\right)^2 + \left(\frac{\tau_{23}}{S_{23}}\right)^2$$

$$\mathcal{F}_3 = \left(\frac{\sigma_3}{S_3}\right)^2 - \frac{\sigma_3\sigma_1}{S_3S_1} + \left(\frac{\sigma_1}{S_1}\right)^2 + \left(\frac{\tau_{13}}{S_{13}}\right)^2$$

In these equations, the strength parameters, S_1, S_2, and S_3, are defined as:

$$S_1 = S_1^t + \mathcal{H}(-\sigma_1)\,(S_1^c - S_1^t)$$
$$S_2 = S_2^t + \mathcal{H}(-\sigma_2)\,(S_2^c - S_2^t)$$
$$S_3 = S_3^t + \mathcal{H}(-\sigma_3)\,(S_3^c - S_3^t)$$

...where H is the Heaviside step function defined as $H(x \geq 0)=1$ and $H(x<0)=0$. In 2D or for plane stress conditions, the stress function reduces to:

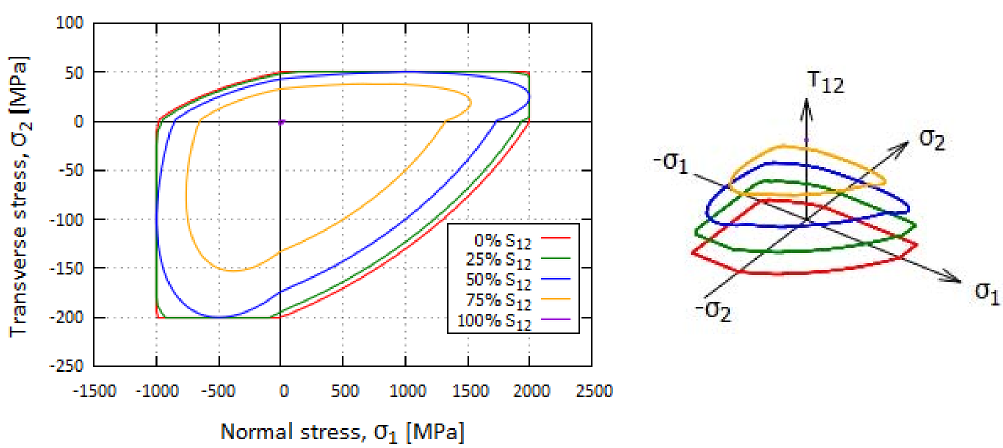
$$\mathcal{F} = \max(\mathcal{F}_1, \mathcal{F}_2, \mathcal{F}_3)$$

...where:

$$\mathcal{F}_1 = \left(\frac{\sigma_1}{S_1}\right)^2 - \frac{\sigma_1 \sigma_2}{S_1 S_2} + \left(\frac{\sigma_2}{S_2}\right)^2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2$$
$$\mathcal{F}_2 = \left(\frac{\sigma_2}{S_2}\right)^2$$
$$\mathcal{F}_3 = \left(\frac{\sigma_1}{S_1}\right)^2$$

Figure 5-221 shows the failure envelopes generated by the 2D version of this criterion. They are plotted for several positive shear stress values, τ_{12} ranging 0-100% of the shear strength value, S_{12}.

Fig. 5-221 Failure envelopes produced by the plane stress version of the Norris criterion



(a) In the σ_1-σ_2 stress space (b) In the σ_1-σ_2-τ_{12} stress space

Norris-McKinnon criterion (1962)

This stress-based failure criterion was proposed by Norris and McKinnon for composites under plane stress loading conditions [14]. They postulated that the failure function is defined as a simple second order interaction formula:

$$F = \left(\frac{\sigma_1}{S_1}\right)^2 + \left(\frac{\sigma_2}{S_2}\right)^2 + \left(\frac{\sigma_{12}}{S_{12}}\right)^2$$

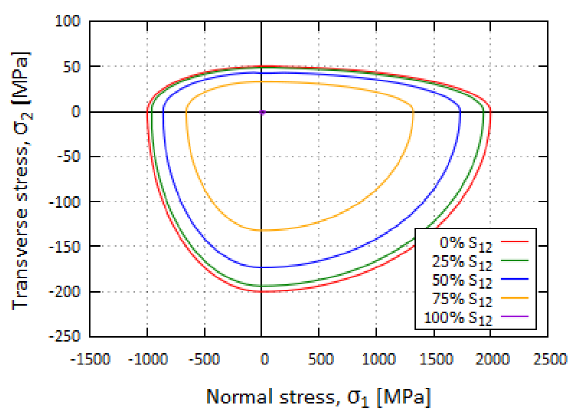
....where the strength parameters, S_1 and S_2 are defined as:

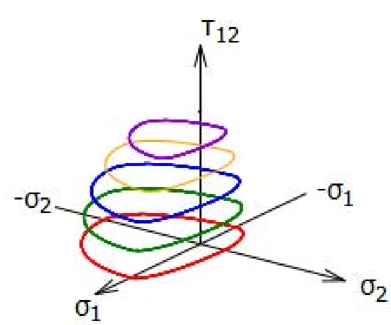
$$S_1 = S_1^t + H(-\sigma_1)(S_1^c - S_1^t)$$

$$S_2 = S_2^t + H(-\sigma_2)(S_2^c - S_2^t)$$

In these equations, H is the Heaviside step function defined as $H(x \geq 0) = 1$ and $H(x < 0) = 0$. shows the failure envelopes generated by this criterion. They are plotted for several positive shear stress values, τ_{12} ranging 0-100% of the shear strength value S_{12}.

Fig. 5-222 Failure envelopes produced by the Norris-McKinnon criterion





(a) In the σ_1-σ_2 stress space

(b) In the σ_1-σ_2-τ_{12} stress space

Tsai-Hill criterion (1948)

The Hill criterion, also known as the Tsai-Hill criterion in the composite community, is a fully interactive quadratic stress-based criterion. It was initially defined to predict anisotropic yielding of metals [7]. It can be used for composites when regarding the yield stresses as failure strength parameters. In 3D, its failure function is defined as:

$$\mathcal{F} = F(\sigma_2 - \sigma_3)^2 + G(\sigma_3 - \sigma_1)^2 + H(\sigma_1 - \sigma_2)^2 + 2L\tau_{12}^2 + 2M\tau_{23}^2 + 2N\tau_{13}^2$$

...where:

$$F = \frac{1}{2}\left(\frac{1}{S_2^2} + \frac{1}{S_3^2} - \frac{1}{S_1^2}\right) \qquad L = \frac{1}{2S_{12}^2}$$

$$G = \frac{1}{2}\left(\frac{1}{S_3^2} + \frac{1}{S_1^2} - \frac{1}{S_2^2}\right) \qquad M = \frac{1}{2S_{23}^2}$$

$$H = \frac{1}{2}\left(\frac{1}{S_1^2} + \frac{1}{S_2^2} - \frac{1}{S_3^2}\right) \qquad N = \frac{1}{2S_{13}^2}$$

In these equations, the strength parameters, S_1, S_2 and S_3, are defined as:

$$S_1 = S_1^t + \mathcal{H}(-\sigma_1)\,(S_1^c - S_1^t)$$

$$S_2 = S_2^t + \mathcal{H}(-\sigma_2)\,(S_2^c - S_2^t)$$

$$S_3 = S_3^t + \mathcal{H}(-\sigma_3)\,(S_3^c - S_3^t)$$

...where H is the *Heaviside* step function defined such that $H(x \geq 0)=1$ and $H(x < 0)=0$. For plane stress conditions, the stress function for orthotropic materials reduces to:

$$\mathcal{F} = \left(\frac{\sigma_1}{S_1}\right)^2 + \left(\frac{\sigma_2}{S_2}\right)^2 - \left(\frac{1}{S_1^2} + \frac{1}{S_2^2} - \frac{1}{S_3^2}\right)\sigma_1\sigma_2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2$$

Azzi and Tsai [1] and Tsai [15] subsequently applied Hill's criterion to unidirectional composite laminates. Within the framework of transversely isotropic materials with respect to the 1-direction, the previous failure function reduces to:

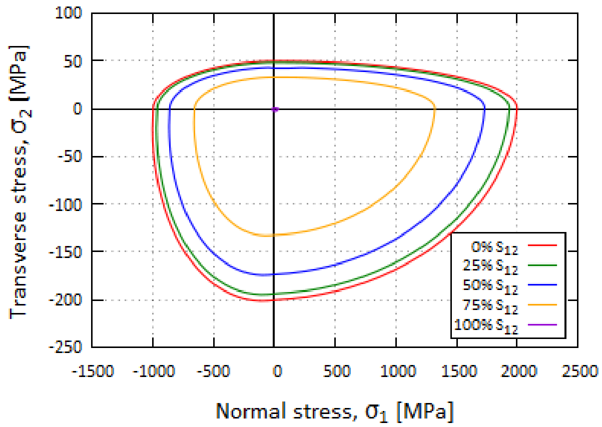
$$\mathcal{F} = \frac{\sigma_1^2 - \sigma_1\sigma_2}{S_1^2} + \left(\frac{\sigma_2}{S_2}\right)^2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2$$

Figure 5-223 shows the failure envelopes generated by this criterion. They are plotted for several positive shear stress values, τ_{12}, ranging 0-100% of the total shear strength value, S_{12}.

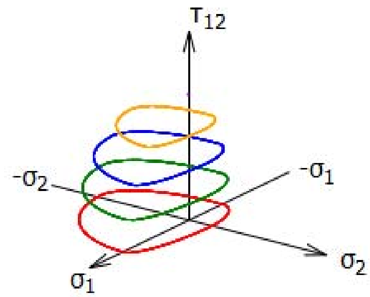
Note

- Tsai-Hill criterion for orthotropic materials under plane stress is not available as there is no through-thickness stress, σ_3, in the formulation making it possible to distinguish between $S_3 = S_3^t$ and $S_3 = S_3^c$.
- The failure index, FI, may be negative for very particular stress combinations. When this happens, failure is not expected to occur and the design parameters, namely SR, SEF and MoS, are each given a value by default.

Fig. 5-223 Failure envelopes produced by the plane stress version of Tsai-Hill criterion



(a) In the σ_1-σ_2 stress space



(b) In the σ_1-σ_2-τ_{12} stress space

Tsai-Wu criterion (1971)

This criterion is a tensor polynomial fully interactive stress-based failure criterion proposed by Tsai and Wu for anisotropic materials [14]. It accounts for both tensile and compressive strengths as well as shear strengths in a single equation. For 3D orthotropic materials, its failure function is defined as:

$$\begin{aligned}\mathcal{F} = {} & F_{11}\sigma_1^2 + F_{22}\sigma_2^2 + F_{33}\sigma_3^2 + F_{44}\tau_{12}^2 + F_{55}\tau_{23}^2 + F_{66}\tau_{13}^2 \\ & +2F_{12}\sigma_1\sigma_2 + 2F_{23}\sigma_2\sigma_3 + 2F_{13}\sigma_1\sigma_3 \\ & +F_1\sigma_1 + F_2\sigma_2 + F_3\sigma_3\end{aligned}$$

...where F_i and F_{ij} (i,j=1,2,3) are the components (Voigt's notation) of the strength tensors of the second and fourth rank respectively. They are defined as:

$$F_1 = \frac{1}{S_1^t} - \frac{1}{S_1^c} \qquad F_{11} = \frac{1}{S_1^t S_1^c} \qquad F_{44} = \frac{1}{S_{12}^2}$$

$$F_2 = \frac{1}{S_2^t} - \frac{1}{S_2^c} \qquad F_{22} = \frac{1}{S_2^t S_2^c} \qquad F_{55} = \frac{1}{S_{23}^2}$$

$$F_3 = \frac{1}{S_3^t} - \frac{1}{S_3^c} \qquad F_{33} = \frac{1}{S_3^t S_3^c} \qquad F_{66} = \frac{1}{S_{13}^2}$$

The remaining components are interactive parameters defined as:

$$F_{12} = F_{12}^* \sqrt{F_{11}F_{22}}$$

$$F_{23} = F_{23}^* \sqrt{F_{22}F_{33}}$$

$$F_{13} = F_{13}^* \sqrt{F_{11}F_{33}}$$

...where F_{12}^* , F_{13}^* , and F_{23}^* are dimensionless empirical coefficients ranging from -1 to 1. For orthotropic materials under plane stress, the failure function reduces to:

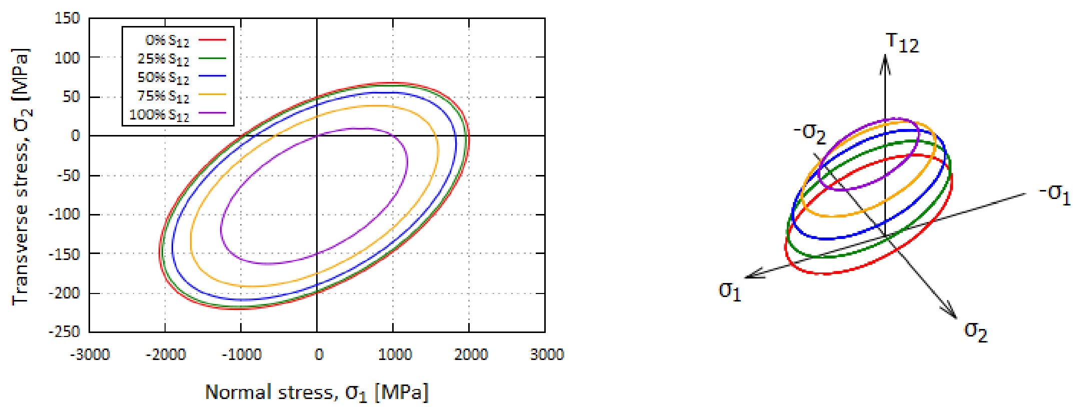
$$\mathcal{F} = \frac{\sigma_1^2}{S_1^t S_1^c} + \frac{\sigma_2^2}{S_2^t S_2^c} + \left(\frac{\tau_{12}}{S_{12}}\right)^2 + 2F_{12}^*\sqrt{F_{11}F_{22}}\sigma_1\sigma_2 + \left(\frac{1}{S_1^t} - \frac{1}{S_1^c}\right)\sigma_1 + \left(\frac{1}{S_2^t} - \frac{1}{S_2^c}\right)\sigma_2$$

Figure 5-224 shows the failure envelopes generated by the plane stress version of this criterion when F_{12}^* = -1/2. They are plotted for several positive shear stress values, τ_{12}, ranging 0-100% of the shear strength value, S_{12}.

Notes

- The strength parameters must verify the "stability conditions": $F_{ii}F_{jj}-F_{ij}^2 > 0 \quad (i,j=1,2,3)$
- These conditions are equivalent to having $-1 < F_{ij}^* < 1 \quad (ij=12,23,13)$
- The Hoffman criterion is found when $F_{12}^* = -F_{11}/\left(2\sqrt{F_{11}F_{22}}\right)$
- The Tsai-Hill criterion is found in the very particular case when $S_1^t = S_1^c = S_1$, $S_2^t = S_2^c = S_2$, and $F_{12}^* = -S_2/(2S_1)$

Fig. 5-224 Failure envelopes produced by the plane stress version of the Tsai-Wu criterion



(a) In the σ_1-σ_2 stress space

(b) In the σ_1-σ_2-τ_{12} stress space

Christensen criterion (1997)

By introducing different transversely isotropic stress invariants with respect to the 1-axis from that used by Hashin [6], Christensen derived a 3D criterion for aligned fibre composite materials distinguishing fibre-dominated and matrix-dominated failure modes in tension and compression [2]. In 3D, the failure function is defined as:

$$\mathcal{F} = \max\left(\mathcal{F}_f, \mathcal{F}_m\right)$$

...where F_f is the fibre-dominated failure function defined as the maximum stress criterion:

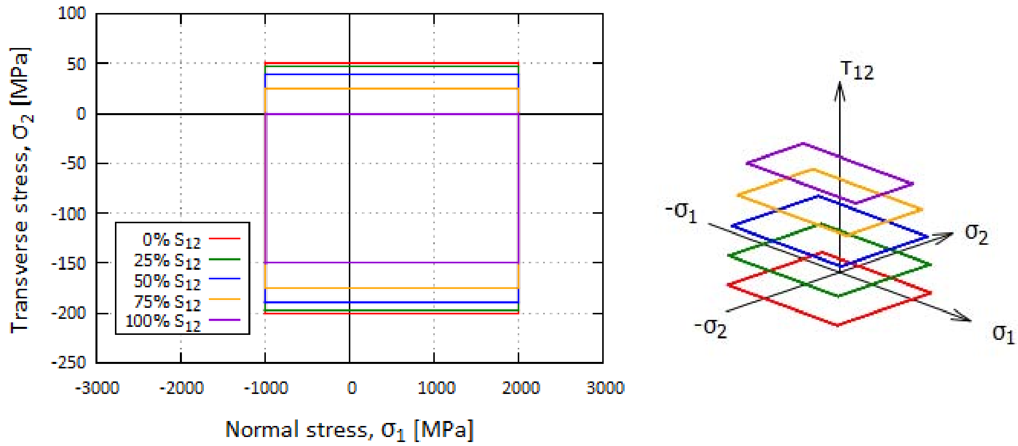
$$\mathcal{F}_f = \begin{cases} \sigma_1/S_1^t & (\sigma_1 \geq 0) \\ -\sigma_1/S_1^c & (\sigma_1 < 0) \end{cases}$$

...and F_m is the matrix-dominated failure function defined as:

$$\mathcal{F}_m = \frac{1}{S_2^t S_2^c}\left[(\sigma_2 - \sigma_3)^2 + 4\tau_{23}^2\right] + \frac{\tau_{12}^2 + \tau_{13}^2}{S_{12}^2} + \left(\frac{1}{S_2^t} - \frac{1}{S_2^c}\right)(\sigma_2 + \sigma_3)$$

Figure 5-225 shows the failure envelopes generated by the plane stress version of this criterion. They are plotted for several positive shear stress values, τ12, ranging 0-100% of the shear strength value, S_{12}.

Fig. 5-225 Failure envelopes: plane stress version of the Christensen criterion



(a) In the σ_1-σ_2 stress space

(b) In the σ_1-σ_2-τ_{12} stress space

Hashin criterion (1980)

Hashin derived his criterion for 3D transversely isotropic materials after observing that the Tsai-Wu criterion predicted failure under biaxial tensile stress by accounting for compressive strength values, which is physically unrealistic [6]. He introduced a set of transversely isotropic stress invariants with respect to the fibre 1-direction to derive a set of four criteria making the distinction between fibre and matrix failure modes in tension and compression. Within this framework, Hashin's failure function is defined as:

$$\mathcal{F} = \max\left(\mathcal{F}_f, \mathcal{F}_m\right)$$

...where F_f is the fibre-dominated failure function defined as:

$$\mathcal{F}_f = \begin{cases} \left(\frac{\sigma_1}{S_1^t}\right)^2 + \alpha\,\frac{\tau_{12}^2 + \tau_{13}^2}{S_{12}^2} & (\sigma_1 \geq 0) \\ -\frac{\sigma_1}{S_1^c} & (\sigma_1 < 0) \end{cases}$$

and F_m is the matrix-dominated failure function defined as:

$$\mathcal{F}_m^t = \left(\frac{\sigma_2+\sigma_3}{S_2^t}\right)^2 + \frac{{\tau_{23}}^2-\sigma_2\sigma_3}{S_{23}^2} + \frac{\tau_{12}^2+\tau_{13}^2}{S_{12}^2}$$

$$\mathcal{F}_m^c = \frac{1}{S_{23}^2}\left[\left(\frac{\sigma_2-\sigma_3}{2}\right)^2 + {\tau_{23}}^2\right] + \frac{\tau_{12}^2+\tau_{13}^2}{S_{12}^2} + \frac{1}{S_2^c}\left[\left(\frac{S_2^c}{2S_{23}}\right)^2 - 1\right](\sigma_2+\sigma_3)$$

...where:

For plane stress conditions, the failure function reduces to:

$$\mathcal{F} = \max(\mathcal{F}_f, \mathcal{F}_m)$$

...where:

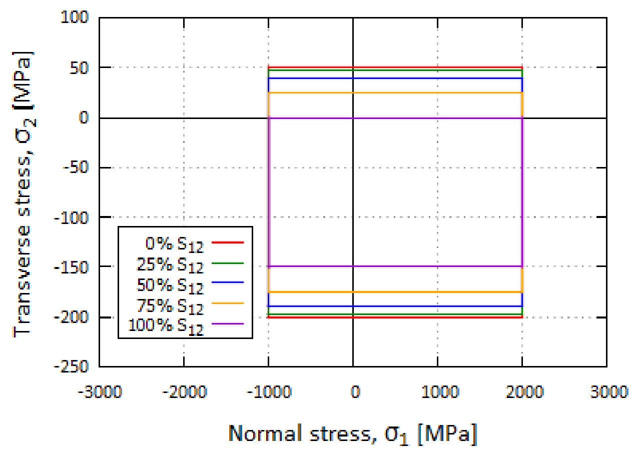
$$\mathcal{F}_f = \begin{cases} \left(\frac{\sigma_1}{S_1^t}\right)^2 + \alpha\left(\frac{\tau_{12}}{S_{12}}\right)^2 & (\sigma_1 \geq 0) \\ -\frac{\sigma_1}{S_1^c} & (\sigma_1 < 0) \end{cases}$$

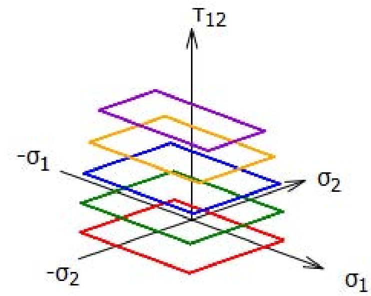
$$\mathcal{F}_m = \begin{cases} \left(\frac{\sigma_2}{S_2^t}\right)^2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2 & (\sigma_2 \geq 0) \\ \left(\frac{\sigma_2}{2S_{23}}\right)^2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2 + \left[\left(\frac{S_2^c}{2S_{23}}\right)^2 - 1\right]\frac{\sigma_2}{S_2^c} & (\sigma_2 < 0) \end{cases}$$

Figure 5-226 and Figure 5-227 show the failure envelopes generated by the plane stress version of this criterion when α = 0 and 1 respectively. They are plotted for several positive shear stress values, τ_{12}, ranging 0-100% of the shear strength value, S_{12}.

Note	• α is a positive dimensionless parameter ranging from 0 to 1. It determines the contribution of the shear stress to the fibre tensile criterion. • When α=0, the fibre dominated tensile stress function reduces to that of the maximum stress criterion in tension.

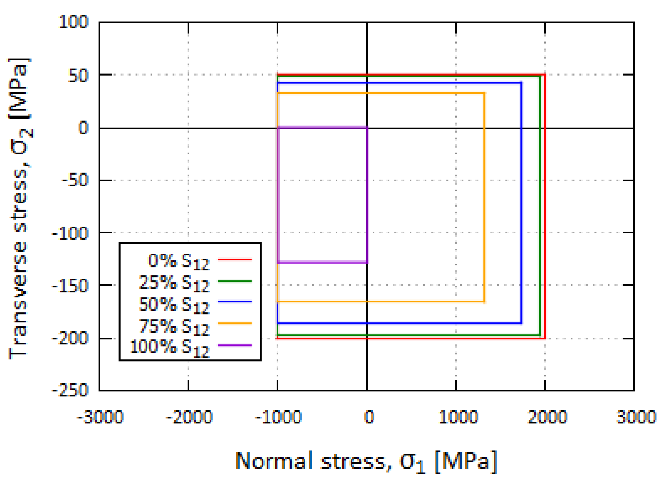
Fig. 5-226 Failure envelopes: plane stress version of the Hashin criterion when α=0

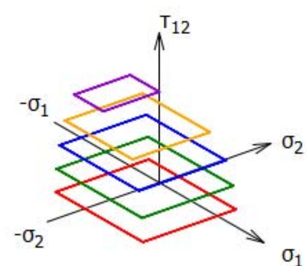




(a) In the σ_1-σ_2 stress space (b) In the σ_1-σ_2-τ_{12} stress space

Fig. 5-227 Failure envelopes: plane stress version of the Hashin criterion when α=1





(a) In the σ_1-σ_2 stress space (b) In the σ_1-σ_2-τ_{12} stress space

Hashin-Rotem criterion (1973)

After several post-mortem examinations of unidirectional composite specimens loaded at various angles from their fibre orientations, Hashin and Rotem established a set of four criteria to make the distinction between fibre and matrix failure modes in tension and compression [5]. By assuming plane stress conditions, they formulated the failure function as:

$$\mathcal{F} = \max(\mathcal{F}_f, \mathcal{F}_m)$$

...where F_f is the fibre-dominated failure function defined as the maximum stress criterion:

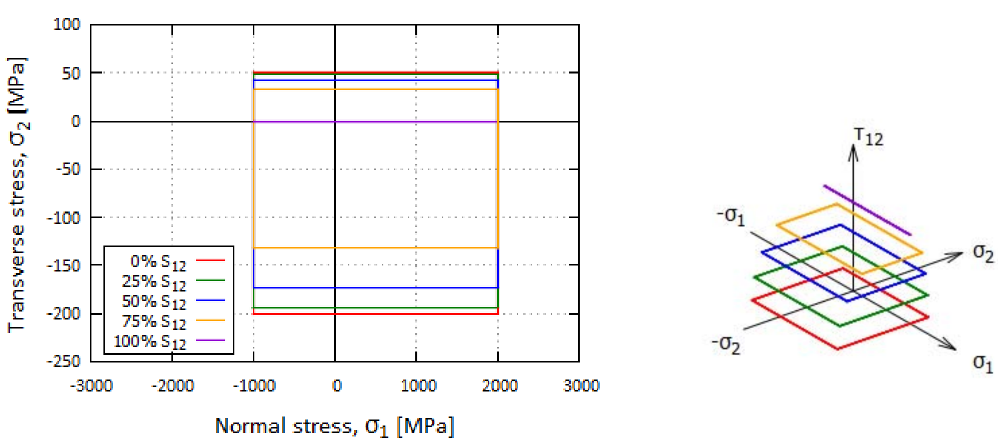
$$\mathcal{F}_f = \begin{cases} \sigma_1 / S_1^t & (\sigma_1 \geq 0) \\ -\sigma_1 / S_1^c & (\sigma_1 < 0) \end{cases}$$

...and F_m is the matrix-dominated failure function defined as:

$$\mathcal{F}_m = \begin{cases} \left(\frac{\sigma_2}{S_2^t}\right)^2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2 & (\sigma_2 \geq 0) \\ \left(\frac{\sigma_2}{S_2^c}\right)^2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2 & (\sigma_2 < 0) \end{cases}$$

Figure 5-228 shows the failure envelopes generated by this criterion. They are plotted for several positive shear stress values, τ_{12}, ranging 0-100% of the shear strength value, S_{12}.

Fig. 5-228 Failure envelopes produced by the Hashin-Rotem criterion



(a) In the σ_1-σ_2 stress space (b) In the σ_1-σ_2-τ_{12} stress space

Hashin-Sun criterion (1996)

Since measuring the shear strength in the 2-3 plane (S_{23}) experimentally may be difficult, Sun et al. [15] proposed an empirically-modified version of the plane stress version of Hashin's criterion [7] capable of capturing the apparent material shear strength enhancement when transverse compression is applied. They introduced an empirical parameter, μ, playing a role similar to an internal friction coefficient. Their stress function is formulated as:

F=max(F_f,F_m)

...where F_f is the fibre-dominated failure function defined as:

$$\mathcal{F}_f = \begin{cases} \sigma_1/S_1^t & (\sigma_1 \geq 0) \\ -\sigma_1/S_1^c & (\sigma_1 < 0) \end{cases}$$

...and F_f is the matrix-dominated failure function defined as:

$$\mathcal{F}_m = \left(\frac{\sigma_2}{S_2^c}\right)^2 + \left(\frac{\tau_{12}}{S_{12} + \mu\,\mathcal{H}(-\sigma_2)}\right)^2$$

In above equation, H is the Heaviside step function defined as $H(x \geq 0) = 1$ and H (x < 0) = 0. Figure 5-228 shows two examples of Hashin-Sun failure envelopes plotted in the σ_2 - τ_{12} stress space when μ = 0 and 0.5. The highest possible enhanced shear strength is denoted S_{12}^*. When μ = 0, S_{12}^* = S_{12} as shown in Figure 5-228.

Fig. 5-229 Matrix-dominated Hashin-Sun failure envelopes when μ = 0 and 0.5

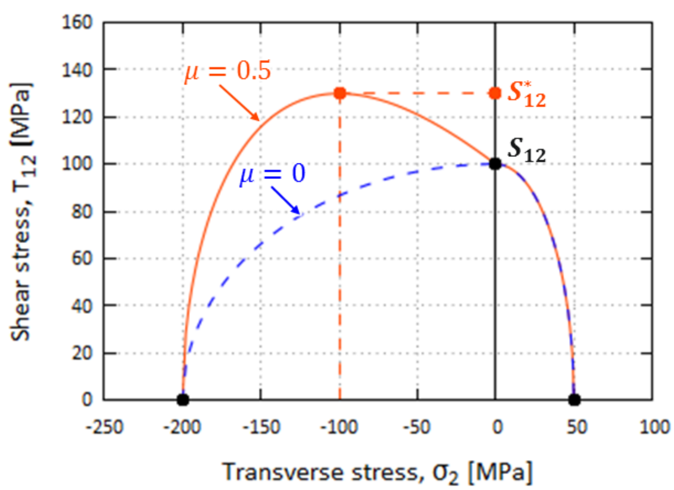
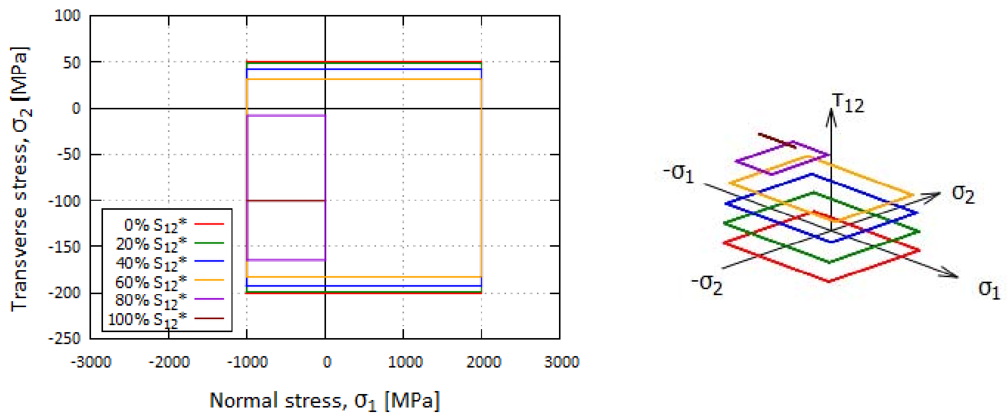


Figure 5-230 shows the failure envelopes plotted for several positive shear stress values, τ_{12}, ranging 0 - 100% of the highest possible enhanced shear strength, S_{12}^*.

Fig. 5-230 Failure envelopes produced by the Hashin-Sun criterion when μ =0.5.



a) In the σ_1-σ_2 stress space (b) In the σ_1-σ_2-τ_{12} stress space

Note

- μ is a positive empirically-defined parameter chosen such that the Hashin-Sun failure envelope fits the experimental one in the σ_2-τ_{12} stress space.
- When $\mu = 0$, the Hashin-Sun criterion becomes the Hashin-Rotem criterion.

Modified NU criterion (2009)

The NU theory was developed by Daniel et al. at Northwestern University [4]. Based on micromechanical matrix-dominated failure mechanisms, it is expressed in terms of measured macro-mechanical lamina and strength properties. This made them derive three sub-criteria for three different failure modes:

Compression-dominated failure:

$$\mathcal{F}_m^{(a)} = \left(\frac{\sigma_2}{S_2^c}\right)^2 + \left(\alpha\,\frac{\tau_{12}}{S_2^c}\right)^2 \quad \text{(NUa)}$$

Shear-dominated failure:

$$\mathcal{F}_m^{(b)} = \left(\frac{\tau_{12}}{S_{12}}\right)^2 + \frac{2}{\alpha}\frac{\sigma_2}{S_{12}} \quad \text{(NUb)}$$

Tension-dominated failure:

$$\mathcal{F}_m^{(c)} = \frac{\sigma_2}{S_2^t} + \left(\frac{\alpha}{2}\frac{\tau_{12}}{S_2^t}\right)^2 \quad \text{(NUc)}$$

In the above equations, $\alpha = E^2/G_{12}$ is the ratio of the transverse Young's modulus, E_2, to the in-plane shear modulus, G_{12}.

In its initial version, the NU theory does not include the influence of σ_1 in the fibre direction. This is why it has been enriched in DesignLife with the maximum stress criterion to predict fibre failure. For plane stress conditions, the failure function is therefore defined as:

$$F = \max(F_f, F_m)$$

In this equation:

$$\mathcal{F}_f = \begin{cases} \sigma_1/S_1^t & (\sigma_1 \geq 0) \\ -\sigma_1/S_1^c & (\sigma_1 < 0) \end{cases}$$

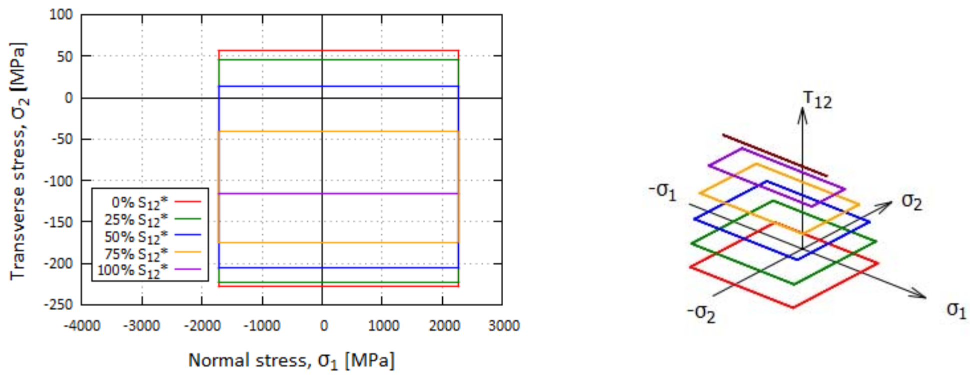
...and:

$$\mathcal{F}_m = \begin{cases} \mathcal{F}_m^{(c)} & (\sigma_2 \geq 0) \\ \mathcal{F}_m^{(b)} & (\sigma_2 < 0 \;\wedge\; \sigma_2/\sigma_2^* \leq |\tau_{12}|/S_{12}^*) \\ \mathcal{F}_m^{(a)} & (\sigma_2 < 0 \;\wedge\; \sigma_2/\sigma_2^* > |\tau_{12}|/S_{12}^*) \end{cases}$$

...where $\sigma_2^* = \alpha S_{12} - S_2^c$ is the stress at which the failure envelope switches from NUa to NUb and S_{12}^* is the highest possible enhanced shear strength when $\sigma_2 = \sigma_2^*$.

Figure 5-231 shows the failure envelopes for several positive shear stress values, τ_{12}, ranging 0 100% of the highest possible enhanced shear strength, S_{12}^*. They have been plotted using AS4/3506-1 material data [9] to easily visualise the contribution of each of the 3 sub-criteria, namely NUa, NUb, and NUc.

Fig. 5-231 Failure envelopes produced by the modified NU criterion



a) In the σ_1-σ_2 stress space (b) In the σ_1-σ_2-τ_{12} stress space

Note

- The NU theory is relevant for stiff and strong unidirectional carbon fibre-reinforced plastic composites and assumes that no fibre micro-buckling occurs in longitudinal compression.
- The original NU theory provides a continuous failure envelope when $S_{12} = 2S_2^t / \alpha$
- To ensure continuity in the overall failure envelope, both the NUb and NUc failure functions, $F_m^{(b)}$ and $F_m^{(c)}$, are reduced to:

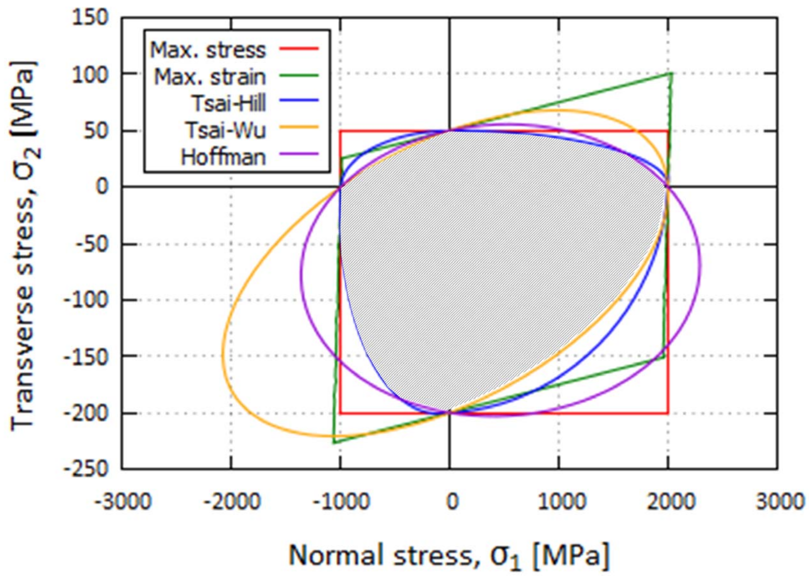
$$F_m^{(d)} = \frac{\sigma_2}{S_2^t} + \left(\frac{\tau_{12}}{S_{12}}\right)^2$$

Applicability of intra-laminar failure criteria

The validity and applicability of a given failure theory depends on the convenience of application and, most importantly, on the agreement with an experimental failure envelope [3]. If the user is not able to select the most suitable failure criterion, they can consider adopting a conservative approach. In this case, all the pre-selected failure theories are combined so that they define the smallest possible failure envelope.

Figure 5-232 is an example of conservative failure envelope (hatched region) produced under bi-axial normal loading (τ_{12}=0) after selecting four failure theories Figure 5-232 shows the resulting envelope indicated by the hatched region when using four failure theories for example.

Fig. 5-232 Example of conservative failure envelope after selecting 5failure theories



References

1. Azzi VD, Tsai SW. Anisotropic strength of composites. Experimental Mechanics 1965; 5(9): 283-288.
2. Christensen RM. Stress based yield/failure criteria for fiber composites. International Journal of Solids and Structures 1997; 34(5): 529-543.
3. Daniel IM, Ishai O. Engineering Mechanics of Composite Materials. Oxford University Press, 1994, 410 p.
4. Daniel IM, Luo J-J, Schubel PM, Werner BT. Interfiber/interlaminar failure of composites under multi-axial states of stress. Composites Science and Technology 2009; 69: 764-771.
5. Franklin HG. Classic theories of failure of anisotropic materials. Fibre Science and Technology 1968; 1(2): 137-150.
6. Hashin Z, Rotem A. A fatigue failure criterion for fiber reinforced materials. Journal of Composite Materials 1973; 7: 448-464.
7. Hashin Z. Failure criteria for unidirectional fiber composites. ASME Journal of Applied Mechanics 1980; 47: 329-334.
8. Hill R. A theory of the yielding and plastic flow of anisotropic metals. Proceedings of the Royal Society of London A 1948; 193(1033): 281-297.
9. Hinton MJ, Kaddour AS, Soden PD. Failure Criteria in Fibre Reinforced Polymer Composites: The World-Wide Failure Exercise. Elsevier, 2004, 1269 p.

10. Hoffman O. The brittle strength of orthotropic materials. Journal of Composite Materials 1967; 1: 200-206.

11. Lempriere BM. Poisson's ratio in orthotropic materials. AIAA Journal 1968; 6(11): 2226-2227.

12. Marin J. Theories of strength for combined stresses and nonisotropic materials. Journal of the Aeronautical Sciences 1957; 24(4): 265-268.

13. Norris CB. Strength of orthotropic materials subjected to combined stresses. U.S. Department of Agriculture report No.1816, May 1962, 42 p.

14. Norris CB, McKinnon PF. Compression, tension, and shear tests on yellow-poplar plywood panels of sizes that do not buckle with tests made at various angles to the face grain. Forest Products Laboratory Report No. 1328, 1956.

15. Sun et al. U.S. Comparative evaluation of failure analysis methods for composite laminates. Department of Transportation report Ref. DOT/FAA/AR-95/109, May 1996.

16. Tsai SW, Wu EM. A general theory of strength for anisotropic materials. Journal of Composites Materials 1971; 5(1): 58-80.

17. Tsai SW. Strength characteristics of composite materials. NASA CR-224, April 1965.

18. Tsai SW. Theory of Composite Design. In: Strength & Life of Composites, Tsai SW, Ed. JEC Composites, 2008.

5.18 WholeLife Analysis Engine

WholeLife Weld CAE Fatigue performs fatigue analysis of welds from finite element models. Bending and Tension stress profiles are taken directly from FE results and these are used within the WholeLife analysis. The WholeLife model combines aspects of both the strain-life (EN) and fracture mechanics (LEFM) models into a unified model for fatigue life estimation.

Fig. 5-233 WholeLife analysis engine properties

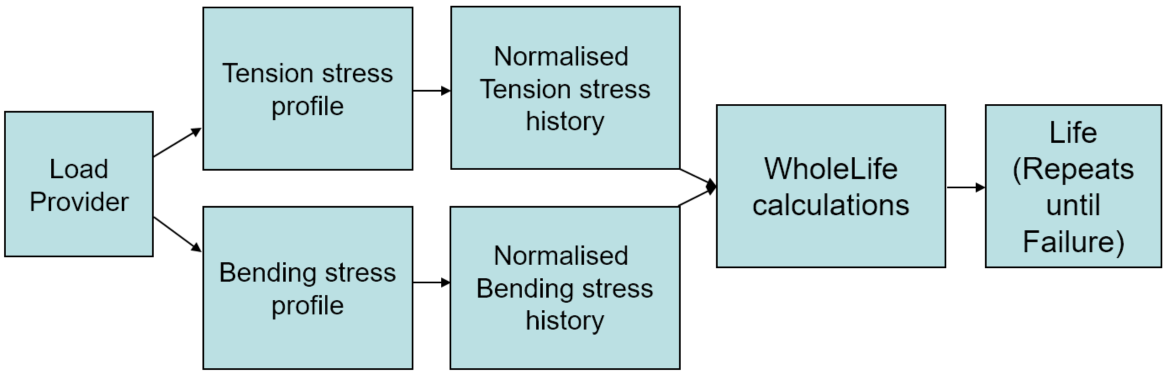
Object Name: WholeLifeEngine (WholeLife analysis engine)

Name	Value	Description
General		
OutputMaterialNames	False	Whether to output material names to the results
OutputDistributedSource	False	Whether to output details of the distributed process that generated each result
ScaleFactor	1	The scale factor to apply prior to damage calculation
DutyCycle		
EventProcessing	Independent	How to process separate events in duty cycles
OutputEventResults	False	Whether to output results per event or not for duty cycle processing
Analysis		
WeldTypeSource	Properties	The source for the weld type definition and weld properties
InitialCrackLengthA		Enter the initial crack length in the units defined on the FEModel
InitialCrackLengthB		Enter the initial crack length in the units defined on the FEModel
FinalCrackLengthA		Enter the final crack length in the units defined on the FEModel
FinalCrackLengthB		Enter the final crack length in the units defined on the FEModel
MaxCycles	1000000	Maximum number of cycles used in the iterative crack growth calculation.
ReportInterval	1000	Specifies the number of cycles between each line in the output report
WeldType		
WeldType	Fillet	Type of weld
FilletWeldProperties		
t		Sheet thickness (horizontal part) in the units defined on the FEModel
r		Radius at the weld toe in the units defined on the FEModel
theta		Angle of the weld face in degrees
TerminationCondition		
TerminateOnCrackLength	All	Crack Length Termination Condition
TerminateOnAppliedStress	All	Applied Stress Termination Condition
Advanced		

5.18.1 Engine Operation Summary

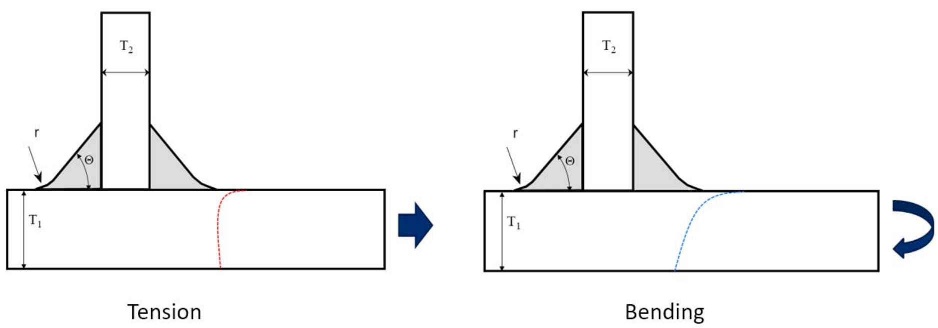
The basic operation of the WholeLife analysis engine can be summarized in Figure 5-234. The engine requires the tension and bending stress profiles down through the component at each calculation point. The stress profiles are obtained by using the SolidWeld Solution location as used in a standard seam weld analysis from solid elements, see the DesignLife theory guide ("Seam Weld Analysis Engine" on page 264).

Fig. 5-234 Basic WholeLife analysis engine steps



The stress profile contains a stress concentration factor at the imitation site and these stress profiles are a function of the loading mode as well as the component geometry.

Fig. 5-235 Stress profiles dependent on loading mode



The un-cracked stress profile is factored by the universal weight functions to produce a stress profile for the cracked plate. This calculation is moderately CPU intensive and a non-proportional load history would require this calculation at each step.

The structural stress approach in DesignLife use the membrane and bending structural stresses at the weld toe and these are used here to create two structural stress histories. In the WholeLife solver the two un-cracked stress profiles (pure bending and pure tension) are combined with the universal weight functions to give two cracked stress profiles. These resulting cracked stress profiles can be superimposed.

5.18.2 Analysis Engine Properties

The analysis engine has a number of properties which are fully documented in the *DesignLife Programmer's Manual*. The following properties that directly control the analysis process are discussed here.

5.18.3 EventProcessing

This property controls the handling of duty cycles across events. In principle, and specifically as regards rainflow counting, duty cycles are handled in the same way as for the Standard S-N analysis engine.

Table 5-25 EventProcessing properties

Parameter Name	Description
Independent	Each unique event in the duty is calculated separately.
CombinedFull	The whole duty cycle is logically concatenated and processed as if it were one long event, including all repeats of all events in the correct sequence.

5.18.4 Analysis Parameters

WeldTypeSource

Defines the source for weld type definition and properties. There are two possible options:

Table 5-26 WeldTypeSource properties

Parameter Name	Description
Properties	The weld properties are read from properties on the analysis engine. When this option is selected all the welds locations present in the analysis must use the same properties.
FromMaterialMap	The weld details are read from the bill of materials defined in the material map. This allows the weld properties to vary for each weld location. The following properties can be read from a bill of materials : *WeldType*, *r*, *t*, *theta*, *InitialCrackLengthA*, *InitialCrackLengthB*, *FinalCrackLengthA*, *FinalCrackLengthB*, *TerminateOnCrackLength*, *TerminateOnAppliedStress*. To enable this feature they must be selected via the additional datacolumns option on the material map loader.

InitialCrackLengthA

This is the initial crack length for a semi-elliptical crack and is the dimension through the thickness. This should be entered in the same units as the defined on the FEModel. If the length entered for initial crack length A or B is less than the material property Rho* then an analysis including crack initiation will be carried out.

InitialCrackLengthB

This is the initial crack length for a semi-elliptical crack and is the dimension along the surface. This should be entered in the same units as the defined on the FEModel. If the length entered for initial crack length A or B is less than the material property Rho* then an analysis including crack initiation will be carried out.

FinalCrackLengthA

This is the final crack length for a semi-elliptical crack and is the dimension through the thickness. This should be entered in the same units as the defined on the FEModel.

FinalCrackLengthB

This is the final crack length for a semi-elliptical crack and is the dimension along the surface. This should be entered in the same units as the defined on the FEModel.

MaxCycles

This is the maximum number of cycles allowed in the iterative crack growth calculation. The analysis will stop when it reaches this number of cycles and report the results at this point.

ReportInterval

This specifies the number of cycles between each line in the output report. It is not usually necessary to report the results for every cycle because these can become very long and time consuming to process. This value sets the desired reporting interval, for example 1 cycle in every 1000.

Note: If more than 2^16 iterations are set to be returned, this value will be increased to reduce the number of reported cycles to within this range. A warning message is provided when this occurs.

5.18.5 CustomWholeLifeWeldType

This specifies the customised WholeLife Weld to use. This property is used only if WeldType is Custom or the WeldTypeSource is FromMaterialMap. Custom methods are defined in the CustomWholeLifeWelds.sys file.

The list of options for this property is generated by reading this configuration file. There will always be a default value of None, which is intended to be used when WholeLifeWeldType is not set to Custom. This denotes the fact that no custom method is selected.

There must be a materials data set in a materials database with a type name that matches the selected method name to be able to analyse using the method.

5.18.6 TerminationCondition

TerminateOnCrackLength

This selects which termination condition to use for the crack length.

Table 5-27 TerminateOnCrackLength properties

Parameter Name	Description
None	The calculation will not terminate due to the crack length exceeding FinalCrackLengthA or FinalCrackLengthB.
All	Uses both crack dimensions.
DimensionAOnly	Only checks for failure based on the crack length through the thickness (FinalCrackLengthA).
DimensionBOnly	Only checks for failure based on the crack length along the surface (FinalCrackLengthB).

TerminateOnAppliedStress

Table 5-28 TerminateOnAppliedStress properties

Parameter Name	Description
None	The calculation will not terminate due to the applied stress exceeding the Fracture Toughness.
All	Check for the applied stress exceeding the Fracture Toughness in both crack dimensions.
DimensionAOnly	Only checks for the applied stress exceeding the Fracture Toughness in the through the thickness direction.

Table 5-28 TerminateOnAppliedStress properties

Parameter Name	Description
DimensionAOnly	Only checks for the applied stress exceeding the Fracture Toughness in the along the surface direction.

5.18.7 WeldType

This is the source for the weld type definition and weld properties and has two possible options:

Table 5-29 WeldType properties

Parameter Name	Description
Fillet	MIG/MAG fillet weld.
Custom	User defined weld.

5.18.8 Theoretical Background

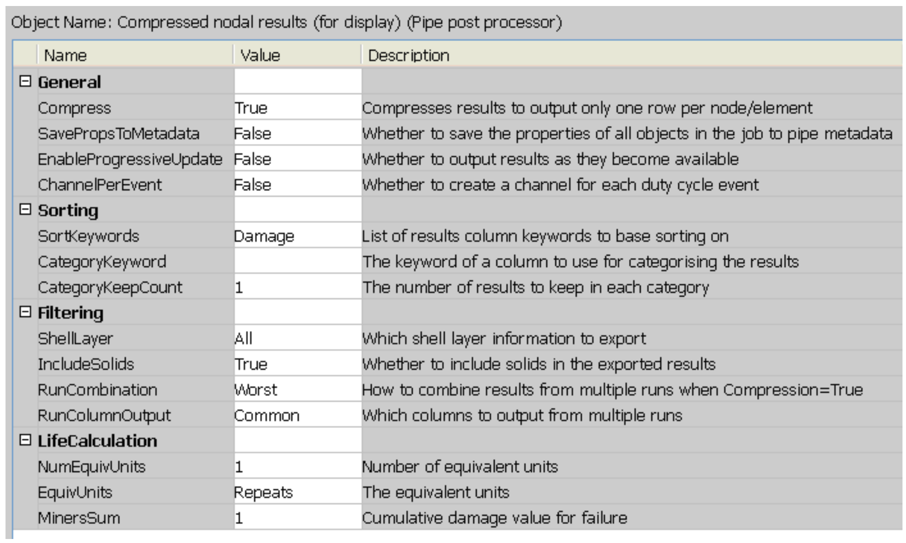
The WholeLife model combines aspects of both the strain-life (EN) and fracture mechanics (LEFM) models into a unified model for fatigue life estimation. The model can produce fatigue life estimates that are considerably more accurate than the classical approaches. This is particularly apparent with welded structures and lightweight structures, such as the fatigue of aluminium panels and riveted joints etc. In these cases, a significant period of time is spent in both initiating and propagating the crack to failure. In many cases the crack will actually propagate into a low stressed region and this can result in the crack arresting and never propagating to failure. In this case it is important to recognise that these cracks do no compromise the durability of the structure and may be acceptable in-service.

The WholeLife model is based on the work of Glinka and Mikheevskiy [1] and this is covered in detail in “WholeLife Theory Guide”.

6 Results

As regards the Results processing options, most of the outputs have been discussed in the above analysis engine discussions. Additional information on the Miner's sum and Equivalent units settings on the results post-processors is provided below.

Fig. 6-1 Pipe post-processor settings



Object Name: Compressed nodal results (for display) (Pipe post processor)

Name	Value	Description
⊟ General		
Compress	True	Compresses results to output only one row per node/element
SavePropsToMetadata	False	Whether to save the properties of all objects in the job to pipe metadata
EnableProgressiveUpdate	False	Whether to output results as they become available
ChannelPerEvent	False	Whether to create a channel for each duty cycle event
⊟ Sorting		
SortKeywords	Damage	List of results column keywords to base sorting on
CategoryKeyword		The keyword of a column to use for categorising the results
CategoryKeepCount	1	The number of results to keep in each category
⊟ Filtering		
ShellLayer	All	Which shell layer information to export
IncludeSolids	True	Whether to include solids in the exported results
RunCombination	Worst	How to combine results from multiple runs when Compression=True
RunColumnOutput	Common	Which columns to output from multiple runs
⊟ LifeCalculation		
NumEquivUnits	1	Number of equivalent units
EquivUnits	Repeats	The equivalent units
MinersSum	1	Cumulative damage value for failure

The fatigue analysis engines (with the exception of the Standard Assessment, Safety Factor, and the Dang Van engines) calculate **fatigue damage**. The translation of damage into life is left to the postprocessor (e.g., pipe postprocessor), which provides the output for contour plotting or tabulation. By default:

Life = 1/Damage and is expressed in Repeats

If MinersSum is set to another value

Life(Repeats) = (MinersSum)/Damage

The ability to change the Miner's sum provides a simple way to make the life prediction more or less conservative. For example, some calculation methods require the Miner's sum to be set to 0.3 or 0.5 for variable amplitude loadings.

Note that Chaboche creep results have two damage values reported. The damage is reported as 1/Life, but this is in fact the average damage for a repeat as the damage can be non-linear. Therefore, the ChabocheDamage is also reported. This is the damage for the first repeat and may not be equal to 1/Life.

We have seen that by default, nCodeDT provides fatigue analysis life predictions in units of Repeats, i.e., it predicts how often the applied loading, whether a time history or duty cycle, etc., could be repeated before failure. Sometimes it may be convenient to get results in units that are more meaningful to the user, e.g., laps of a test track, equivalent hours, years, etc. This can be achieved by setting equivalent units.

If NumEquivUnits (number of equivalent units) and EquivUnits (equivalent units) are set, then in addition to the life in Repeats, an additional column of results—Eq Life (equivalent life) is output in the Equivalent Units.

Eq Life = Life(Repeats) x (NumEquivUnits) expressed in EquivUnits.

HBM - nCode

26555 Evergreen Road - Suite 700 -
Southfield, MI 48076 - USA
Tel. +1 248 350 8300 - Fax: +1 248 350 1678
Email: info@hbmncode.com - www.hbm.com/ncode

measure and predict with confidence