

**MASTER THESIS FALL 2014
FOR
STUD. TECHN. TONE WERMUNDSEN**

**Efficient evaluation of fatigue and damage in welded joints from finite element analysis
Effektiv evaluering av utmatting og skade i sveiser fra elementmetode resultater**

The use of the Finite Element Method (FEM) for engineering purposes has been established for several decades. In their everyday work, mechanical and structural engineers utilize FEM codes to develop and improve their products. The approach is highly accepted and the results are often directly applied when evaluating a current or proposed design. As modern FE software becomes more intuitive and easy to use, having a clear understanding of the underlying methods and recognition of their limitations becomes more important than ever. One of the key aspects with mechanical design is to assess a products ability to withstand repeated loading. Understanding the factors influencing a products serviceable life is crucial to the development of improved product design, and the ability to meet increasing product requirements.

Motivation

Several industry accepted recommended practices regarding assessment of the fatigue properties of products already exists. Common to these is the fact that evaluating the fatigue properties of welds and hot spot stresses requires extrapolation of results from the base material towards the area of interest, and further processing of the results, as the numerical methods produce unphysical results at sharp geometric transitions. This often results in significant manual work and relies on the engineer's ability to locate critical areas. An algorithm that identifies these critical areas along welds or stress hot spots, automatically extracts the results at extrapolation points, and uses these to determine fatigue life or accumulated damage would be very valuable. Such a tool would reduce the work required to perform fatigue calculations of structures involving welds, and also reduce the sources of error, ultimately leading to safer design and reduced cost.

Tasks to be performed

The algorithm as described above will be implemented in ANSYS Mechanical, one of the leading finite element software packages available. ANSYS Mechanical has a development toolkit that allows users to write their own post-processing routines and make these available for other users through installation packages called extensions. The language used in the development toolkit is python and xml.

The tasks to be performed can be summarized as:

1. Study and document the recommended practice for fatigue evaluation using numerical methods from:
 - a. DNV RP C203, for high cycle fatigue
 - b. DNV RP C208, for low cycle fatigue

2. Become familiar with ANSYS ACT, the Application Customization Toolkit, used for developing extensions for ANSYS Mechanical.
3. Create a simulation model consisting of several parts held together with welds. The weld itself is not to be modelled. The model should have a known solution or simple enough such that a manual fatigue assessment of the weld is possible.
4. Develop an extension that performs weld fatigue checks according to both mentioned recommended practices, and compare the results with the known solution in 3.
5. If time permits, expand the extension to evaluate whether 'shake down' or 'ratcheting' occurs when the structure is subjected to repeated load cycles, and compare this to a known solution.

The scope of work for the thesis should be limited to:

- Straight lined and circular welds
- Shell elements with linear shape functions
- Plane and cylindrical plates


Three weeks after start of the thesis work, an A3 sheet illustrating the work is to be handed in. A template for this presentation is available on the IPM's web site under the menu "Masteroppgave" (<http://www.ntnu.no/ipm/masteroppgave>). This sheet should be updated one week before the Master's thesis is submitted.

Performing a risk assessment of the planned work is obligatory. Known main activities must be risk assessed before they start, and the form must be handed in within 3 weeks of receiving the problem text. The form must be signed by your supervisor. All projects are to be assessed, even theoretical and virtual. Risk assessment is a running activity, and must be carried out before starting any activity that might lead to injury to humans or damage to materials/equipment or the external environment. Copies of signed risk assessments should also be included as an appendix of the finished project report.

The thesis should include the signed problem text, and be written as a research report with summary both in English and Norwegian, conclusion, literature references, table of contents, etc. During preparation of the text, the candidate should make efforts to create a well arranged and well written report. To ease the evaluation of the thesis, it is important to cross-reference text, tables and figures. For evaluation of the work a thorough discussion of results is appreciated.

The thesis shall be submitted electronically via DAIM, NTNU's system for Digital Archiving and Submission of Master's thesis.

Contact person: Frode Halvorsen, EDRMedeso AS, Frode.Halvorsen@edrmedeso.com



Torgeir Welo
Head of Division



Bjørn Haugen
Professor/Supervisor

Preface

This master thesis was written in the time period August to December 2014 by Tone Wermundsen, studying Engineering Science and ICT with specialization in Product Development and Materials, at IPM NTNU.

The thesis addresses the importance of fatigue assessment and presents an implemented solution for efficient evaluation.

The study and work was done with the help and guidance of Associate Professor at NTNU Bjørn Haugen, Vice President at EDR Medeso Frode Halvorsen and Senior Technical Consultant at EDR Medeso Kjetil Rognlien.

Acknowledgment

I would like to thank Bjørn Haugen, Frode Halvorsen, Kjetil Rognlien and the other employees at EDR Medeso for this assignment, their guidance and help throughout this period.

Abstract

Fatigue analysis with non-linear loading conditions, such as wind and waves, is a complicated process that require thoroughly groundwork of the structural design. For such structures, weld lines in joints are particular hard to assess.

Two of today's practices of fatigue analysis are done according to DNVGL-RP-C203 (high cycle region) and DNV-RP-C208 (low cycle region). The recommended practices are hard to automate, and when done manually most likely inaccurate.

This thesis will provide a study of the processes described in both recommended practices, and start to present a solution to automate the processes of doing fatigue calculations, by creating extensions for the engineering simulation software ANSYS. The *DNV Fatigue Toolbar* presented provides tools for evaluating fatigue in base material and joints in both the high and low cycle region.

DNV Fatigue Toolbar is divided into main categories

- Screening
- Plane Plated Weld
- Tubular Joint

where *Screening* evaluates fatigue in the base materials and provide guidance to which weld lines need further assessment by giving un-extrapolated values at weld hot spots. When critical weld lines are identified by the help of the screening tools, *Plane Plated Weld* or *Tubular Joint* can be applied to assess hot spot fatigue through extrapolation.

The solutions provided in *DNV Fatigue Toolbar* is a start of implementing fatigue evaluations according to DNVGL-RP-C203 and DNV-RP-C208. Further development, especially on the *Tubular Joint*, is encouraged and what this implies can be found in chapter 7.

Sammendrag

Utmattingsberegninger med ikke-linære lastforhold, som bølger og vind, er komplisert og krever grundig forarbeid av det strukturelle designet. For slike strukturer er sveis i knutepunkt spesielt vanskelig å vurdere.

To av dagens praksiser for evaluering av materiell utmatting er gjort i henhold til DNVGL-RP-C203 og DNV-RP-C208. Disse anbefalte praksisene er vanskelige å automatisere, og gir mest sannsynlig unøyaktige resultater ved manuell utføring.

Denne avhandlingen presenterer en studie av prosessene beskrevet i begge anbefalte praksisene, og presenterer en start av automatisering av dem i form av applikasjoner til simuleringsprogramvaren ANSYS. *DNV Fatigue Toolbar* inkluderer verktøy for utførelse av utmattings analyser i grunnmaterialet og knutepunkt for både høye og lave sykluser.

DNV Fatigue Toolbar er delt inn i tre hovedkategorier

- Screening
- Plane Plated Weld
- Tubular Joint

hvor *Screening* evaluerer fatigue i grunnmaterialet og veileder brukere til å identifisere hvilke sveiser som krever videre vurdering ved å gi materiell utmattelse basert på verdier hentet direkte fra alle hot spot i sveis. Når kritiske sveiser er identifisert ved hjelp av screening verktøyet, kan *Plane Plated Weld* eller *Tubular Joint* verktøyene brukes til å vurdere hot spot i sveis via ekstrapolerte verdier.

Løsningen presentert i *DNV Fatigue Toolbar* er en begynnelse i automatisering av prosessene beskrevet i DNVGL-RP-C203 og DNV-RP-C208. Videre utvikling, spesielt av *Tubular Joint*, oppfordres og hva som kreves av videre arbeid er beskrevet i kapittel 7.

Contents

1	Introduction	2
1.1	Problem Statement	3
1.2	Review of Literature	4
1.3	Review of Software	4
1.4	Readers Guide	5
2	Fatigue	6
2.1	Fatigue Definition	6
2.2	DNV(GL) Fatigue	7
2.2.1	DNVGL-RP-203	7
2.2.2	DNV-RP-208	8
2.3	Base Material versus Hot Spots	9
2.3.1	Extrapolation Along Plane Plated Welds	10
2.3.2	Extrapolation Along Tubular Joints	12
3	DNV Fatigue Toolbar	16
3.1	Screening	17
3.1.1	Background	17
3.1.2	Solution	17
3.1.3	Usage	18
3.2	Plane Plated Weld	19
3.2.1	Background	19
3.2.2	Solution	19
3.2.3	Usage	20
3.3	Tubular Joint	20
3.3.1	Background	20
3.3.2	Solution	21
3.3.3	Usage	21
4	Known Issues and Limitations	22
4.1	Limitations	22
4.2	Error and Warning Messages	24

5	Solution Implementation	26
5.1	Stress/Strain Ranges	26
5.2	Classification of Geometry	30
5.2.1	Weld Definition and Directional Extrapolation in Plane Plated Welds	30
5.2.2	Hot Spot Definition and Directional Extrapolation in Tubular Joints	33
5.3	Extrapolation Points	36
5.4	Element Searching	38
5.4.1	Ray Casting Limitations and Handling of These	39
5.5	Finding Exact Value at Coordinate Point	44
5.6	Value at Hot Spot	46
5.7	Fatigue Life	46
5.7.1	High Cycle	46
5.7.2	Low Cycle	46
5.8	Combining Load Cases	48
6	Validation and Examples	50
6.1	High Cycle Screening	50
6.1.1	Stress Range	51
6.1.2	Fatigue Life	55
6.2	Low Cycle Screening	59
6.2.1	Strain Range	59
6.2.2	Fatigue Life	62
6.3	High Cycle Plane Plated Weld	66
6.3.1	Extrapolated Stress Range	67
6.4	Low Cycle Plane Plated Weld	70
6.5	High/Low Cycle Tubular Joint	70
7	Conclusion	72
7.1	Further Work	73
	Appendices	76

List of Figures

1.1	<i>A tubular joint with crack developing</i>	2
2.1	<i>Tubular joint geometry definition. Image from DNVGL-RP-C203 section 4.2</i>	13
3.1	<i>The overview GUI of DNV Fatigue Toolbar in ANSYS Mechanical</i>	16
5.1	<i>Default setup of an Element-Tensor with nodal component result</i>	26
5.2	<i>Bottom Result plot of Maximal Principal Stress on one Element</i>	27
5.3	<i>Top Result plot of Maximal Principal Stress on one Element .</i>	28
5.4	<i>Middle Result plot of Maximal Principal Stress on one Element</i>	28
5.5	<i>Top/Bottom Result plot of Maximal Principal Stress on one Element</i>	29
5.6	<i>An L shell model weld line with all potential directions. The black arrows indicate possible directions normal to the weld along the dark blue face, the pink arrows normal to the weld along the light blue surface and the green parallel to the weld line</i>	31
5.7	<i>An L shell model weld line with all potential directions after dismissal of directions pointing with the weld line in hot spots other than the weld toes</i>	32
5.8	<i>A T-joint with potential extrapolation directions. Purple lines represents direction from crown point, green from saddle point</i>	34
5.9	<i>A T-joint with vector rotated α degrees found from equation 5.1, giving the extrapolation point</i>	35
5.10	<i>Point in Polygon with ray-casting. The blue point is not within the polygons boundary, and the ray crossed the polygons edges twice. The green point is within the polygons boundary, and crossed the polygons edges once, an odd number which in the algorithm ray-casting defines as being inside the polygon</i>	38

5.11	<i>Ray Casting Problem with extrapolation along curvature. The ray must be defined in the same plane as the element that is searched for, but as this is unknown, and assumptions regarding it is difficult to generalize, this problem was not solved . .</i>	39
5.12	<i>Ray Casting limitations. Both the green and the blue point cross the polygon edges twice, and is defined as outside the polygons boundaries though both should be defined as within . .</i>	40
5.13	<i>Outline of meshed L-Shell model. The hot spot (red dot) has a desired extrapolation point located in element id 77. The first element checked will be element id 56. This element is rejected, as the point is not within its boundary. Potential elements to check further included element ids: 80, 36 and 79 [not 160 and 159 as they belong to another face]. Element id 79 centroid is closest to the point to find, and is therefore selected as next element to check</i>	42
5.14	<i>Example of mapping of an isoparametric Four-node element in physical space[6]</i>	44
6.1	<i>Loads and boundary conditions for the simple Tubular K-Joint example model</i>	50
6.2	<i>Equivalent Stress Range generated by High Cycle Screening with start step = 0 and end step = 4. Point of maximal range marked</i>	52
6.3	<i>Equivalent Stress generated by ANSYS default result object at time step = 4</i>	52
6.4	<i>Equivalent Stress generated by ANSYS default result object at time step = 3</i>	53
6.5	<i>Equivalent Stress generated by ANSYS default result object at time step = 1</i>	54
6.6	<i>Equivalent Stress Range generated by High Cycle Screening with start step = 1 and end step = 3</i>	55
6.7	<i>Details view of High Cycle Screening. Chosen output is Logarithm of Cycles to Failure to Base 10</i>	56
6.8	<i>Logarithmic Fatigue Life plotted results for High Cycle Screening result object with property settings as in figure 6.7</i>	57
6.9	<i>Accumulated Damage plotted results for High Cycle Screening result object with property settings as in figure 6.7. Number of Cycles is set to 50</i>	58
6.10	<i>Principal strain, EPTO1, plotted in ANSYS by default result object at time step = 4</i>	59

6.11	<i>Principal strain, EPTO2, plotted in ANSYS by default result object at time step = 4</i>	60
6.12	<i>Principal strain, EPTO3, plotted in ANSYS by default result object at time step = 4</i>	61
6.13	<i>Maximum Principal Strain Range generated by Low Cycle Screening with start step = 0 and end step = 4. Point of maximal value marked</i>	62
6.14	<i>Detais view of Low Cycle Screening. Chosen output is Logarithm of Cycles to Failure to Base 10</i>	63
6.15	<i>Logarithmic Fatigue Life plotted results for Low Cycle Screening result object with property settings as in figure 6.14</i>	64
6.16	<i>Accumulated Damage plotted results for Low Cycle Screening result object with property settings as in figure 6.14. Number of Cycles is set to 50</i>	65
6.17	<i>Loads and Boundary Conditions for the simple simple L-shell example model</i>	66
6.18	<i>Probed maximal principal strains generated from Low Cycle Screening result object for extrapolation of values to hot spot at end of edge, Id = 2</i>	67
6.19	<i>Probed maximal principal strains generated from Low Cycle Screening result object for extrapolation of values to hot spot in edge, Id = 28</i>	68
6.20	<i>Logarithmic Fatigue Life plotted on part edge with environment Air and S-N Curve High Strength Steel, generated by High Cycle Plane Plated Weld</i>	69
1	<i>Installing an extension in ANSYS Workbench</i>	77
2	<i>Successfully installed extension in ANSYS Workbench</i>	77
3	<i>Activate extension to use it in a project</i>	78
4	<i>Manually insert the address where the scripted version of an extension is stored</i>	79
5	<i>ACT Development Extension. Update and view Extension Log File</i>	80
6	<i>Example of string written in Extension Log File when High Cycle Screening is evaluated. Output will be "ready for HC Screening m" when the mesh unit is meters</i>	80
7	<i>Uninstall an extension in ANSYS Workbench</i>	81
8	<i>System Overview</i>	89
9	<i>Sequence diagram 1</i>	90
10	<i>Sequence diagram 2</i>	91
11	<i>Sequence diagram 3</i>	92

List of Tables

2.1	Data for Low Cycle Fatigue Analysis of Base Material	9
2.2	Read-out points for extrapolation in tubular joints	13
5.1	Stress/ Range Calculation	30
1	Details View of the <i>High Cycle Screening</i> result object	82
2	Details View of the <i>Low Cycle Screening</i> result object	83
3	Details View of the <i>High Cycle Plane Plated Weld</i> result object	84
4	Details View of the <i>Low Cycle Plane Plated Weld</i> result object	85
5	Details View of the <i>High Cycle Tubular Joint</i> result object . .	86
6	Details View of the <i>Low Cycle Tubular Joint</i> result object . .	87
7	Hot Spot Classification Table****	88

List of Algorithms

1	Extrapolation Directions	32
2	Extrapolation Locations for Tubular Joints	37
3	RayCasting	41
4	Element Searching	43
5	Newton-Raphson	47

Nomenclature

$\Delta\epsilon$	Strain range
$\Delta\epsilon'_f$	Ductility coefficient
$\Delta\sigma$	Stress range
$\Delta\sigma'_f$	Strength coefficient
ϵ	Strain
ϵ -N Curve	Strain-number of cycles to failure relation
σ	Stress
D	Accumulated damage
E	Young's modulus
$\log(N)$	Logarithmic cycles to failure to base 10
N	Cycles to failure
n	Cycles in a load block
t	Plate or beam thickness
t_{ref}	Reference thickness
ACT	ANSYS Customization Toolkit
APDL	ANSYS Parametric Design Language
API	Application Programming Interface
Body	A FE-Model consist of one or more bodies
Brace	All ingoing beams of a tubular joint
Chord	The main beam of a tubular joint

Crown Point	A point type in a tubular joint: on 'top' of welded joint
Edge	A face's boundaries are defined as edges
Element	A FE-Model is presented as a grid of elements
Extension	A plug-in for ANSYS Mechanical
Face	A body consist of one or more faces
FE	Finite Element
GUI	Graphical User Interface
Hot Spot	A point in a weld
Multibody-part	FE-Model that shares connecting edges by sharing the nodes the edge holds
Node	An element is defined by points, called nodes
Ray	A line from one point in a particular direction
Result Object	A type of extensions that evaluates and returns a result type
RP	Recommended practice
S-N Curve	Stress-number of cycles to failure relation
Saddle Point	A point type in a tubular joint: on 'bottom' of welded joint
SCF	Stress concentration factor
Toolbar	A set of clickable buttons in ANSYS Mechanical
Vertex	A corner on a FE-Model
Weld	An edge that is connected to two or more bodies
XML	Extensible Markup Language

Chapter 1

Introduction

One of the key aspects in mechanical design is to assess a products ability to withstand repeated loading [1]. Offshore installations are especially challenging as these structures are exposed to large loads from both wind and waves throughout their lifespan. Structures that are exposed to loads over time, called cyclic loads, will eventually develop microscopic cracks that grow over time [1], see figure 1.1. These crack may lead to failure of the structure before other, more predictable loading conditions do.



Figure 1.1: *A tubular joint with crack developing*

Fatigue evaluation determines how many cycles a structure can tolerate before the material fails and how large the damage distributions are [1]. Recommended practices regarding assessment of fatigue properties already exist and are excepted by several industries. Common for these recommended practices are that evaluation fatigue in welds and critical areas require spe-

cial handling with extrapolation from the base material. These are tedious and time consuming procedures and require significant amount of manual work in addition to rely on the engineers ability to locate critical areas. For assessments in welds, a highly fine mesh is also required to ensure accuracy in fatigue life, which is another time consuming process.

Creating such fine mesh for all critical areas that is demanded, and do the manual work for all possible hot spots are not only time consuming, but often leads to inaccurate results. Sources of errors are all from human mistakes to lack of possible extrapolation point with exact values accessible in a FE-model. The exceptional handling and the expected ability to locate critical areas makes the recommended practices especially hard to automate.

1.1 Problem Statement

Fatigue life is based on stress and strain ranges during a load cycle [4] [5]. Thus, the region with highest fatigue potential is not necessarily the found at the region of the maximal stress in the end configuration. For this reason, it may not always be clear where the highest fatigue potential occurs. This leads to risk of missing important regions when detailed fatigue assessment are performed. An algorithm that identifies critical areas will help solve the problem of human error in terms of locating these areas manually. By the use of a screening result object, the engineer can easily identify critical areas.

Detailed fatigue assessments of critical areas even when these are identifies can be difficult. An algorithm that automatically extract values at extrapolation points and use these to determined fatigue life can solve the time consummating problem and reduce human calculation errors, leading to safer design and reduced cost.

This master thesis has solved the two main issues described above, a screening tool that contribute to identifying critical areas and may discover that cracks will be developed at the base material before welds fail, and an extrapolation tool that evaluates fatigue life of welds in plane plated welds, and begins to solve the problem of evaluation fatigue in simple tubular joints. These solutions are implemented for both High Cycle and Low Cycle Fatigue according to recommended practices DNVGL-RP-C203 and DNV-RP-C208.

The solution has been implemented as extensions for ANSYS version 15, and is only valid for this version. Applying the solution on earlier versions will not be successful, and applying in newer versions may cause problems,

as the application programming interface provided by ANSYS, ACT, is not integrated equally in earlier versions, and is under development for newer versions. For newer versions, the developer must ensure that applied ACT updates are valid for the extensions created by recompiling and testing.

1.2 Review of Literature

The main literature study done in connection to this thesis is the study of the two recommended practices DNVGL-RP-C203 and DNV-RP-C208. Both practices are provided by the now called DNVGL, a merger of *Det Norske Veritas* and *Germanischer Lloyd*, one of the world's leading international classification societies [12]

DNVGL-RP-C203 *Fatigue design of offshore steel structure* is a thorough recommended practice that proposes fatigue life calculations methods for nominal stress, hot spot stress, fracture mechanics and local notch stress for environments air, free corrosion in seawater and cathodic protection in seawater. The RP replaces DNV-RP-C203, originally created in 2001 [4], which has been updated twice per year since. The RP has been widely accepted for years.

DNVGL-RP-C208 *Determination of Structural Capacity by Non-linear FE analysis Methods* is a newer RP and not quite as thoroughly as C203, and relies on the methodology described in DNVGL-RP-C203.

1.3 Review of Software

The solution presented, a toolbar consisting of extensions for result evaluation in ANSYS Mechanical has been created using ACT(ANSYS Customization Toolkit), which provides an API(Application Programming Interface), to several ANSYS Workbench modules. ACT makes it possible to extend the GUI(Graphical Interface), of ANSYS Mechanical with new buttons, which have customized behavior [10]. The customized behavior is dictated by source code written in python and the GUI of the customized objects are written in xml.

Before ACT was presented, ANSYS super users created customized objects by APDL(ANSYS Parametric Design Language) commands. ACT provides many advantages, including:

- Manage user input
- Consistent unit handling
- Access to Mechanical Graphic library
- Shields the source code from users that are not super users
- Customized objects presents in Mechanical GUI
- Creates installable binary files, protecting intellectual property
- Binary installable files does not demand extra licenses

ACT creates many opportunities, but as it is still under development, some limitations and disadvantages also apply:

- Mesh entities as geometry input is limited: graphic result-view is not possible
- ACT customized objects are only valid for the ANSYS version it was created for and require recompiling and validation of a super user to be applied in newer versions
- Some functionality, as property tables and graphic 3D drawing, is missing functionality
- Plotting combined layers for Shell Elements are not possible yet

1.4 Readers Guide

In this master thesis, a study of the recommended practice the implemented solution is based upon can be found in chapter 2, usage and guidance to the implemented solution is provided next along with limitations, before a thorough review of the solution itself is presented in Chapter 4. Validation and testing of the solution is done and discussed in Chapters 6 and 7.

Note that as this thesis has been done in collaboration with EDR Medeso, some of the functionality presented has been introduced to their customers, and usage and guidance Chapter 3 although mainly written by Tone Wermdsen, has been reviewed and improved by employees at EDR Medeso.

Implementation of the solution has been the main focus throughout the period of writing this master thesis, and guidance to installation of the extension can be found in Appendix *Installing the Toolbar*

Chapter 2

Fatigue

Fatigue of materials refers to the change in properties resulting from the application of cyclic loads [1]. Cycle loads, as repeated loading, may be weakening to a structure and fatigue evaluation is the study of determining whether or not these repeated loadings will lead to structural failure and how many of the cyclic loads the structure can withstand. It is common to separate between High and Low Cycle Fatigue; where High Cycle Fatigue is traditionally concerned with situation that require more than 10^4 cycles to failure[4] and Low Cycle Fatigue concerned with situation where plastic deformation occurs [5].

2.1 Fatigue Definition

Fatigue Analysis provide knowledge about fatigue life, N , and accumulated damage, D , where N is the number of cycles till failure occurs. Fatigue behavior characteristics are difficult to predict, and fatigue analysis has long been a time-consuming and cumbersome task as fatigue life is influenced by many possible factors as temperature, material elasticity, structural environment etc. When performing fatigue evaluation, environmental consideration must be taken into account, and the dangers in not executing appropriate analysis are potential catastrophic. Examples where fatigue failure has led to disasters are: the oil platform Alexander Kielland collapse in 1980 [13] and the two de Havilland Comet passenger jet crashes in 1954 [13].

Fatigue Analysis is used to ensure that a structure has a suitable lifespan and may force development changes the design phase. Design against fatigue can

ensure lifespan that meet requirements, and the important task of fatigue calculation is to discover when actions must be taken to ensure a structures perseverance.

2.2 DNV(GL) Fatigue

DNVGL is a Norwegian-German company that delivers technical standards and reliability analysis to marine, oil and gas, and energy sectors [12]. Their standards and recommended practices are widely accepted and used world-wide in offshore business. DNVGL proposes methods for fatigue evaluation in accordance to both high and low cycle situations for offshore steel constructions, the two methods are presented through recommended practice DNVGL-RP-C203 and DNV-RP-C208.

2.2.1 DNVGL-RP-203

DNVGL-RP-C203 *Fatigue design of offshore steel structures* is a recommended practice in relation to fatigue analysis in the high cycle region where stress is relative low and deformation primarily elastic[4]. The fatigue assessment calculations proposed is based on S-N Curves, which are provided, and stress ranges. The formula for cycles to failure given by equation 2.1 [4]

$$\log N = \log \bar{a} - m \log \left(\Delta \sigma \left(\frac{t}{t_{ref}} \right)^k \right) \quad (2.1)$$

where

N = number of cycles to failure for stress range $\log \Delta \sigma$

$\log \bar{a}$ = intercept of Log(N)-axis

m = negative inverse slope of the S-N Curve

$\Delta \sigma$ = stress range

t = thickness through which a crack will most likely grow

t_{ref} = reference thickness (25 mm for welded connections other than tubular joints, 32 mm for tubular joints and 25 mm for bolts)

k = thickness exponent on fatigue strength given by S-N Curve

and equation 2.2 [4]

$$D = \sum_{i=1}^j \frac{n_i}{N_i} \quad (2.2)$$

where

D = the accumulated fatigue damage

n_i = the number of cycles in load block i

N_i = the number of cycles to failure in load block i at constant stress range $\Delta\sigma$

As the formula indicates, selecting the correct S-N Curve is important for accurate and reliable results. Guidance as to how select S-N Curve based on material and geometry is provided throughout the recommended practice DNVGL-RP-C203.

2.2.2 DNV-RP-208

DNV-RP-C208 *Determination of Structural Capacity by Non-linear FE Analysis Methods*, denotes that non-linear FE-Analyses may imply that the structure is loaded beyond proportional limits, and that the structure may be weakened by subsequent load cycles by repeated yielding leading to possible cyclic failure [5]. It is understood that in such situations, fatigue check with stress range is not sufficient evaluation to determine fatigue life [5]. Cyclic plastic strains are overlooked in the process with conventional linear elastic methods such as the method described in DNVGL-RP-C203.

If repeated yielding occurs, fatigue must be evaluated on plastic strains which is proposed in [5] (section 5.2). The methodology is in principal similar to the process from DNVGL-RP-C203: material and environmental conditions are factors influencing fatigue life which is given by equation 2.3 from [5].

$$\frac{\Delta\epsilon_l}{2} = \frac{\sigma'_f}{E}(2N)^{-0.1} + \epsilon'_f(2N)^{-0.43} \quad (2.3)$$

for base material, and by equation 2.4 from [5]

$$\frac{\Delta\epsilon_{hs}}{2} = \frac{\sigma'_f}{E}(2N)^{-0.1} + \epsilon'_f(2N)^{-0.5} \quad (2.4)$$

for welded joints where

$\Delta\epsilon_l$ = the fully reversible maximal principal strain range

$\Delta\epsilon_{hs}$ = the fully reversible maximal principal hot spot strain range

E = the modulus of elasticity (material constant)

σ'_f = the fatigue strength coefficient (material constant)

ϵ'_f = the fatigue ductility coefficient (material constant)

N = the number of cycles to failure

Elasticity Modulus, E-Module, is accessible in ANSYS through ACT, and Fatigue strength and ductility coefficient is found in [5] and rendered in table 2.1

Table 2.1: Data for Low Cycle Fatigue Analysis of Base Material

Data for low cycle fatigue analysis of base material		
Environment	σ'_f [MPa]	ϵ'_f
Air	175	0.091
Seawater with cathodic protection	160	0.057

2.3 Base Material versus Hot Spots

Both DNVGL-RP-C203 and DNV-RP-C208 proposes methods to handle critical areas, denoted as hot spots. Hot Spots are typical located in weld lines or notches and are points in the structure where fatigue cracks may initiate due to the combined effect of structural stress fluctuation and the weld geometry or a similar notch [4]. In finite element models where the weld is not modeled the weld geometry will often present a singularity which will result in FE analysis giving too large stress values at the hot spot. These values will give over-conservative fatigue results.

To avoid over-conservative results where structures are underestimated one can:

1. Model the weld with a sufficient radius, smooth edges and fine mesh
2. Use stress concentrations factors to dimension the stress value found at the hot spot
3. Extrapolate stress values towards the hot spot

Solution 1, modelling the weld with a sufficient radius, smooth radius and fine mesh is an extremely time consuming task, and most often avoided by engineers. This method requires the engineer to be able to make reasonable assumptions regarding the weld and project a realistic weld model construction of a weld that in reality will be uneven. Meshing the weld after modeling is the most time consuming part of this solution, the mesh must be unusually fine, and this mesh is individual for each weld. Manually meshing such fine mesh for each weld in a structure is difficult and demands time in both preparation and analysis.

Using stress concentration factors (a stress concentration factor may be defined as the ratio of hot spot stress range over nominal stress range [4] (section 3.1.1)) is a function of the nominal stress and the geometry of the hot spot. The values found by SCF's are often different than values found through Finite Element Analysis, and the geometry dependency often makes this method unpractical in terms of automatizing the method.

The extrapolation method is often used as it demands less preparation. Mesh requirements are not unacceptable: element size of *plate thickness * plate thickness* is sufficient. The method is direct, extract values from certain points and use these instead of the hot spots value itself.

But the extrapolation method has limitations: it requires precision in collecting extrapolation values as well as require a set of manual calculations for each hot spot desired to inspect.

Though most of the focus lies on fatigue in hot spots, there are situations where cracks can initiate and grow in the base material. Base material does not represent singularities in geometry, and extrapolation is not needed to evaluate fatigue. Note that base material fatigue check is limited to certain S-N Curves.

2.3.1 Extrapolation Along Plane Plated Welds

Extrapolation along plane surfaces should be done at read out points $0.5t$ and $1.5t$, where t is the plate thickness [4]. This demands a mesh size of $t \times t$ in shell models with mid-side nodes, for a regular user to be able to read out values at the exact locations. If the mesh is so perfect, performing manual extrapolation is simple enough. But most often, mesh is not, and read out points are not necessary located on a node.

DNVGL-RP-203 proposes two methods for extrapolating:

Method A Is valid for modeling with shell elements without any weld included in the model, and for modeling with solid elements with the weld included[4]. A linear extrapolation of the hot spot stress can be performed from read out points $0.5*t$ and $1.5*t$, and the effective hot spot stress is given by 2.5 from [4]

$$\sigma_{eff} = \max \begin{cases} \sqrt{\Delta\sigma_{\perp}^2 + 0.81\Delta\tau_{\parallel}^2} \\ \alpha|\Delta\sigma_1| \\ \alpha|\Delta\sigma_2| \end{cases} \quad (2.5)$$

where

$\alpha = 0.90$ if the detail is classified as C2 with stress parallel to the weld at the hot spot [4]

$\alpha = 0.82$ if the detail is classified as C1 with stress parallel to the weld at the hot spot [4]

$\alpha = 0.72$ if the detail is classified as C with stress parallel to the weld at the hot spot [4]

and the principal stresses is calculated as 2.6 and 2.7 from [4]

$$\Delta\sigma_1 = \frac{\Delta\sigma_{\perp} + \Delta\sigma_{\parallel}}{2} + \frac{1}{2}\sqrt{(\Delta\sigma_{\perp})^2 + 4\Delta\tau_{\parallel}^2} \quad (2.6)$$

$$\Delta\sigma_2 = \frac{\Delta\sigma_{\perp} + \Delta\sigma_{\parallel}}{2} - \frac{1}{2}\sqrt{(\Delta\sigma_{\perp})^2 + 4\Delta\tau_{\parallel}^2} \quad (2.7)$$

Method B Is quite similar to method A, with effective hot spot stress given by 2.8 [4]

$$\sigma_{eff} = \max \begin{cases} 1.12\sqrt{\Delta\sigma_{\perp}^2 + 0.81\Delta\tau_{\parallel}^2} \\ 1.12\alpha|\Delta\sigma_1| \\ 1.12\alpha|\Delta\sigma_2| \end{cases} \quad (2.8)$$

These methods depends on knowledge about the stress-components direction in regards to the weld, and take into account directional loading changes in a load cycle. For load cycles without directional loading changes, a simplification can be performed transforming equations 2.5 or 2.8 to equation 2.9:

$$\Delta\sigma_{hs} = \Delta\sigma_a - \left(\frac{a}{b-a}\right)(\Delta\sigma_b - \Delta\sigma_a) \quad (2.9)$$

where

a = the distance from read out point $0.5t$ to the hot spot $= 0.5 \times t$

b = the read out point $1.5t$ to the hot spot $= 1.5 \times t$

$\Delta\sigma_a$ = the stress/strain range at read out point a (absolute maximal principal stress/strain range or equivalent stress range)

$\Delta\sigma_b$ = the stress/strain range at read out point b (absolute maximal principal stress/strain range or equivalent stress range)

2.3.2 Extrapolation Along Tubular Joints

Read out points for extrapolation in tubular joints differ from the static read out points in plane plated welds. The geometry of the tubular joint must be defined, where the different beams are classified as chord or brace, as indicated in figure 2.1

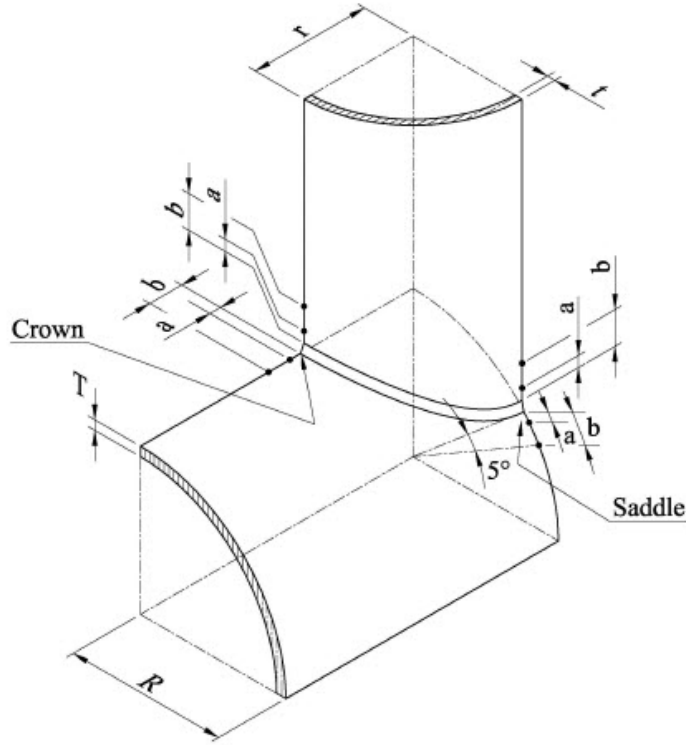


Figure 2.1: *Tubular joint geometry definition. Image from DNVGL-RP-C203 section 4.2*

The read out points are defined from [4] and rendered in table 2.2:

Table 2.2: Read-out points for extrapolation in tubular joints

Extrapolation situation	Read Out Points a and b
Along Brace Surface normal to the hot spot	$a = 0.2\sqrt{rt}$ $b = 0.65\sqrt{rt}$
Along the chord surface normal to the hot spot at crown position	$a = 0.2\sqrt{rt}$ $b = 0.4\sqrt[4]{rtRT}$
Along the chord surface normal to the hot spot at saddle position	$a = 0.2\sqrt{rt}$ $b = 2\pi R \frac{5}{360} = \frac{\pi R}{36}$

where

r = brace radius

t = brace thickness

R = chord radius

T = chord thickness

Chapter 3

DNV Fatigue Toolbar

When automating the process of fatigue evaluation according to DNVGL-RP-C203 and DNV-RP-C208, the best solution found in the time provided was to divide the problem in three subproblems:

- Performing fatigue evaluation on base material
- Performing fatigue evaluation on plane plated welds with the use of extrapolation method
- Performing fatigue evaluation on tubular joints with the use of extrapolation method

Which resulted in a toolbar extension for ANSYS Mechanical, *DNV Fatigue Toolbar*

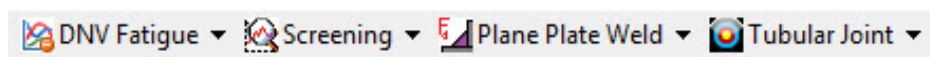


Figure 3.1: *The overview GUI of DNV Fatigue Toolbar in ANSYS Mechanical*

The first clickable option, *DNV Fatigue* opens a drop down menu that contains objects of information about the toolbar, where selecting *Help* will open this document. The other three buttons represent the result objects *Screening*, *Plane Plated Weld* and *Tubular Joint*, each consisting of result objects for both Low and High Cycle Fatigue evaluation.

Note that the base material fatigue evaluator gives results also in hot spots and is therefore classified as a screening tool. This means that the results

given can be used to detect which critical areas and welds that require further investigation. The extrapolating technique implemented includes many heavy operations, such as a search for elements based on coordinates.

Note also that for extrapolation, the simplified method described in Chapter 2 given by equation 2.9 is used

3.1 Screening

3.1.1 Background

Fatigue life is calculated based on the stress/strain range during a load cycle. Thus, the region with highest fatigue potential is not necessarily found at the region of the maximal stress/strain in the end configuration. For this reason, it may not always be obvious where the highest fatigue potential occurs, especially for complex structures with many different loading patterns. When the points for detailed assessment are selected, there is a risk that some points are missed, if not a very high number of points are selected.

The two screening result objects can therefore be used to locate the critical areas and to check for initial crack growth in base material. Note that in the *High Cycle Fatigue Screening Tool*, the same S-N Curve may not apply for both base material and weld screening.

3.1.2 Solution

The two screening tools can be used to plot the number of cycles to failure, or accumulated damage, with the use of S-N Curves or ϵ -N Curves found in DNVGL-RP-C203 or DNV-RP-C208 (section 5.2.6) for a single, or combination of, load cases. The used stress/strain ranges are the stress/strain ranges found at the evaluated node. The two screening tools also provides plot of the stress/strain range used if no combination of load cases is selected.

Note that for *High Cycle Screening*, it is possible to select between absolute maximal principal stress range and equivalent stress range (von-mises), which provide different results.

3.1.3 Usage

The two screening tools can be applied in any Static Structural and Transient Structural analysis. The properties have to be selected or entered regarding how the results should be calculated. All properties are defined in tables 1 and 2 in Appendix *Property Input Guidance* if guidance should be required. The result can be scoped to any type of geometry, but not to mesh entities (directly or through named selections).

The available result plots are:

- *Accumulated Damage*
- *Cycles To Failure*
- *Logarithm Of Cycles To Failure To Base 10*
- *Stress/Strain Range*

The selected result will be plotted directly on the model. If the geometry contains shell elements, the result at the selected shell location will be used as a uniform contour through the thickness. The results can be viewed as *Averaged* or *Unaveraged*.

The result can be calculated for a single, or a combination of, load cases by selecting *Yes* or *No* for *Load Case Combination*. In case of load case combination, the *Load Case Table* will have to be filled in. Refer to Chapter 4 if the table does not pop up. A load case is defined as a range from one load step to another load step in the same analysis. Any analysis sharing the Model cell in Workbench can be used as a load case, but only solved analysis are valid.

Start Step and *End Step*, specified by the user, defines one load cycle, or a stress/strain range. A stress/strain range is the two extremes of a cycle and the stress/strain amplitude is one half of the stress/strain range. Thus, the simulation must include a full load cycle. Note that the time for a chosen step is the steps end time. Step 0 is used to define the initial time 0.

The S-N/ ϵ -N data can be the same for all bodies of the geometry selection or body dependent by selecting *No* for *S-N/ ϵ -N Data Equal For All Bodies*. In the case of body dependent S-N/ ϵ -N data, the *S-N/ ϵ -N Data Table* has to be filled in. The various S-N curves in [4] (Tables 2-1, 2-2 and 2-3) are available, as well as S-N curves for high strength steel as given in [4] (section 2.4.10). The various ϵ -N Curves from [5] (table 5-5) are available.

3.2 Plane Plated Weld

3.2.1 Background

Singularities in geometry, such as sharp edges, represents difficult areas in a geometric FE model. The singularities cause an underestimation in fatigue life as the extracted stress/strain values at the locations are higher than in reality. These critical areas often represents un-modeled welds, and evaluation fatigue life without underestimating these hot spots demands extrapolation of stress/strain values towards the weld. When extrapolating, the points need to be exact for correct and conservative results, and extrapolation must be done in all possible directions as knowledge about which direction lead to the most conservative extrapolated stress/strain value may not always be plausible.

The two *Plane Plated Weld* result objects can be used to find the correct fatigue life in plane plated welds according to [4] and [5]. By selecting whole bodies, the result objects define welds based on geometry and extrapolates values to all nodes in the defined weld, and use this to evaluate fatigue in these hot spots. Note that a weld is defined as an edge that share two or more bodies. If it is desirable to evaluate upon an edge that the two *Plane Plated Weld* result objects does not define as a weld, it is possible to scope geometry to that edge. When geometry is scoped to edges, all these edges will be defined as welds.

3.2.2 Solution

The two *Plane Plated Weld* tools can be used to plot the number of cycles to failure, or accumulated damage, with the use of S-N Curves or ϵ -N Curves found in [4] and [5] (section 5.2.6) for a single, or combination of, load cases. The applied stress/strain ranges are the largest ranges found by extrapolation along all faces connected to the defined weld. It is also possible to display the extrapolated stress/stress range if combination of load cases is not selected.

The extrapolated value is extrapolated from distance $0.5t$ and $1.5t$ perpendicular to the hot spot [4]. The values gathered are linear interpolated values from the surrounding nodes, and therefore the exact simplified-extrapolated value.

3.2.3 Usage

The two *Plane Plated Weld* tools can be applied in any Static Structural and Transient Structural analysis, and the setup is quite similar to the setup in the two *Screening* tools. For property input guidance, please see tables 3 and 4 in Appendix *Property Input Guidance*. Note that for *High Cycle Plane Plated Weld*, the thickness modification factor t/t_{ref} has been divided into two properties *Plate Thickness*, t , and *Reference Thickness*, t_{ref} , and for *Low Cycle Plane Plated Weld* these two properties are additional. Also, in geometry scope, vertices are no longer valid as geometry scope.

The *Plate Thickness* is initiated to the first body's actual thickness, but it is possible to change this property if desired. The plate thickness indicates where the extrapolation is done, and must follow the guidance in DNVGL-RP-C203. Please note that the recommended practice also specifies the mesh size to be no larger than $t \times t$, where t = Plate thickness.

3.3 Tubular Joint

3.3.1 Background

Tubular structures are often welded, and the welds are most often the most critical areas in such structures. In reality, welds are randomly uneven structures, which is near impossible to project into a FE model. By not modeling the weld, the FE model does not represent the welds structural strength sufficiently. Performing fatigue evaluation on such welds, even when the geometry do not consist of sharp edges, requires specific handling. Extrapolating along tubular geometries proves more difficult than extrapolating along plane plates, as the location of extrapolation is dependent on the tubular geometry to be the chord or the brace, and where the hot spot in question is located on the circular weld.

The two *Tubular Joint* result object tries to solve the difficulties in performing fatigue evaluation on simple tubular joints, meaning tubular joints that are created by one chord and one or more ingoing braces. The tools are concerned with the most critical hot spots, which are the crown and saddle point.

3.3.2 Solution

The two *Tubular Joint* tools can be used to plot the number of cycles to failure, or accumulated damage, with the use of S-N Curves or ϵ -N Curves found in [4] and [5] (section 5.2.6) for a single, or combination of, load cases. The applied stress/strain ranges are the largest ranges found by extrapolation along the connected chord and brace of the hot spot in question. It is also possible to see the extrapolated stress/strain range if combination of load cases is not selected.

Note that as these result objects geometry scope is limited to mesh node selection, the results will not be plotted on the model directly, as this is a limitation in ANSYS ACT, refer to Chapter 4.

Also, note that these result object are not finished products, performing evaluation will not give result in most cases.

3.3.3 Usage

The two *Tubular Joint* tools can be applied in any Static Structural and Transient Structural analysis, and the setup is quite similar to the setup in the two *Plane Plated Weld* tools with a few additions. Firstly, the geometry scope is limited to mesh nodes only. The geometry input must also be defined, and a property group *Hot Spot Classification* is added for that purpose. All properties are defined in tables 5 and 6 in the Appendix *Property Input Guidance* if guidance should be required. Please note that each beam must be its own body and end or start edge of each beam must be a single edge (not split edge).

This result object has several limitations regarding the geometry. Please read limitations (Chapter 4) carefully.

Chapter 4

Known Issues and Limitations

4.1 Limitations

DNVGL-RP-C203 Limitations

1. The High Cycle result objects has been made with the purpose of assessing fatigue damage in the high cycle region (DNVGL-RP-C203 section 1.2.3). High cycle loading is normally understood as high cycles more than 10 000, see DNVGL-RP-C203 1.2.3.
2. High Cycle Fatigue analysis is based on elastic stress and therefore linear-elastic materials. See DNVGL-RP-C203 section 1.2.3
3. Mean stresses are neglected, see DNVGL-RP-C203 section 2.3.1
4. Detailed mesh recommendations are given in DNV-RP-C203 section 4.1, 4.2 and 4.3.2. Too large elements sizes may provide non-conservative results.

DNV-GL-C208 Limitations

1. The geometry scope must have plastic behaviour defined to be valid

Common Limitations

1. The High Cycle and Low Cycle result objects are limited to *Static Structural* and *Transient Structural* analyses, 3D geometries and surface bodies.
2. Model must be a multibody part, where nodes are shared between connected, separate bodies

3. Updating invalid results in load case tables from other analysis then the one the result object in inserted in is only done after the result object is clicked.
4. Stopping evaluation by clicking *Stop Solution* in the ANSYS Workbench Solution Status pop-up window is not possible
5. Faces and edges that have been created as *Virtual Topology* are not selectable as geometry input
6. Duplicating a result object with results are not available. Duplicated result objects need to be evaluated
7. Pop-up tables in the details view are not visible on other screens than the main screen
8. Pop-up tables can only be closed by clicking *Apply* or *Cancel*
9. Pop-up tables may not be visible if the *Mechanical* window is scaled too small
10. For geometry scoped to mesh entities, elements or nodes, result cannot be displayed on the model
11. The *Body Id* used in the result objects are not the same as *Reference__ID* found through *Information* in the main toolbar in *Mechanical* details view under body. To be sure which body is which, provide unique names for each body in model
12. Mid side nodes are not included in calculations as these does not hold values in ANSYS (linear approximation between nodes are conducted)
13. The result objects are valid for use in ANSYS 15
14. Result objects cannot be applied on line bodies

Screening Limitations

1. The result objects do not produce accurate results in weld hot spots where stress/strain extrapolation is required.
2. *see Chapter 4* objects can be used on solid models, but this might be a slow process.

Plane Plated Weld Limitations

1. The weld must be a straight line and all connected faces must be plane plates with 4-noded shell elements

2. Extrapolation method is a simplified version of the one described in DNVGL-RP-C203, see 2 equation 2.9

Tubular Limitations

1. All tubular structures connected to the evaluation weld must be defined as separate bodies
2. Extrapolation method is a simplified version of the one described in DNVGL-RP-C203, see 2 equation 2.9
3. Evaluation can only be done on surface models, meaning models created by shell elements
4. Result objects are not finished products and will most likely not provide results

4.2 Error and Warning Messages

Number of cycles to failure exceeds 1.0e+10 for some areas. Use results with caution: For some areas, cycles to failure exceeds the cutoff limit at 10^{10} . Low Cycle Fatigue is not defined for such high life spans, and the results displayed at 10^{10} may be even higher.

Result in Analysis is outdated: 'Load Case Table' contains one or more analysis which have been modified: When combining load cases from different analysis than the analysis that the result object is created in, the additional analysis must be solved. When such an analysis is changed after evaluation, results are outdated and evaluation must be run anew. Result objects will be suppressed and user must unsuppress and evaluate again.

Please select geometry and fill in classification table after selecting 'yes' at property Geometry is Selected: The classification table is dependent on the geometry scope, and cannot be complete before geometry input is selected.

Invalid input: The selected scoped geometry is not valid. Please select different geometry.

Could not recognize any hot spots. Try changing geometry: For *Tubular Joint* result objects, the geometry scope could not be defined as nodes and is not valid. Please try different geometry.

Could not recognize any welds. Try changing geometry to type edge:
For *Plane Plated Weld* result object. No edges in the geometry scope is connected to two or more bodies, no welds are defined and no result will be given. Select edge manually and run anew.

Initial Guess for iteration method Newton-Raphson is not within the methods convergence interval. Please try a different value as some results are not accurate Newton-Raphson iteration for calculation cycles to failure did not converge. Try changing the *Initial Guess*. guidance can be found in an error log file created in your working directory.

Unfinished result object, result may not apply or be viewed at all:
Tubular Joint result object are unfinished, and this warning serves as a reminder.

Chapter 5

Solution Implementation

The solution implemented consist of a toolbar with six result objects, divided into three categories. These result objects have many similarities in solution, and share methods. In this chapter, the implemented solution will be explained with algorithms, equations and illustrations.

5.1 Stress/Strain Ranges

Collecting values through ANSYS ACT is done by having access to the analysis that results should be extracted from. Extracting values is limited to retrieving values from elements only, which hold result components for each of its connected corner-nodes, a tensor, with data structure as shown in figure 5.1.

$$Tensor = < node1_{bottom}, node2_{bottom}, node3_{bottom}, node4_{bottom}, \\ node1_{top}, node2_{top}, node3_{top}, node4_{top} \\ node1_{middle}, node2_{middle}, node3_{middle}, node4_{middle} >$$

Figure 5.1: *Default setup of an Element-Tensor with nodal component result*

The tensor holds six result components (x,y,z,xy,xz,yz) for each connected node, and for shell elements, each of these components consist of three values per location (bottom, top, middle), as shown in figure 5.1. By selecting to

see *Thick Shells and Beams* in Mechanical Toolbar *View*, the affect of these layer-oriented values can be seen as in figures 5.2, 5.3, 5.4 and 5.5

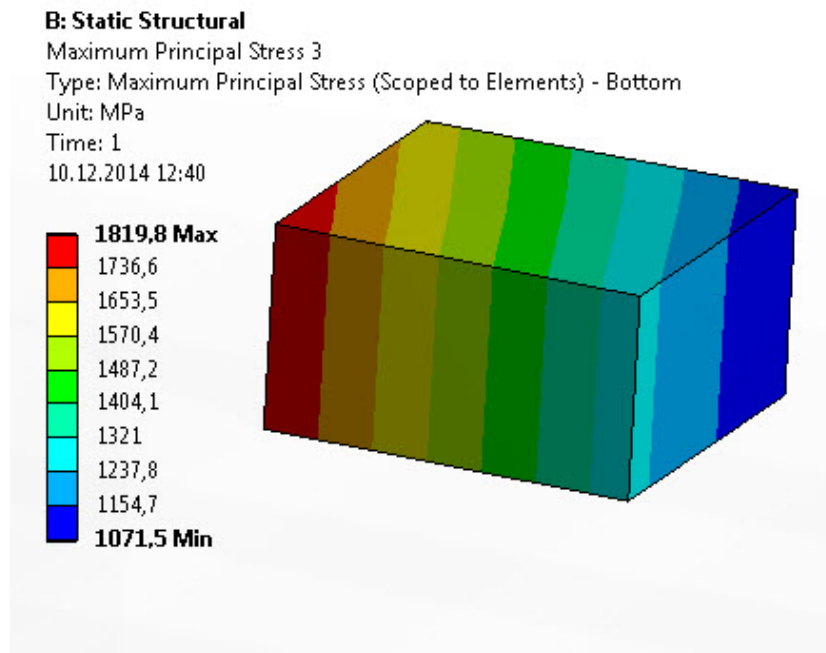


Figure 5.2: *Bottom Result plot of Maximal Principal Stress on one Element*

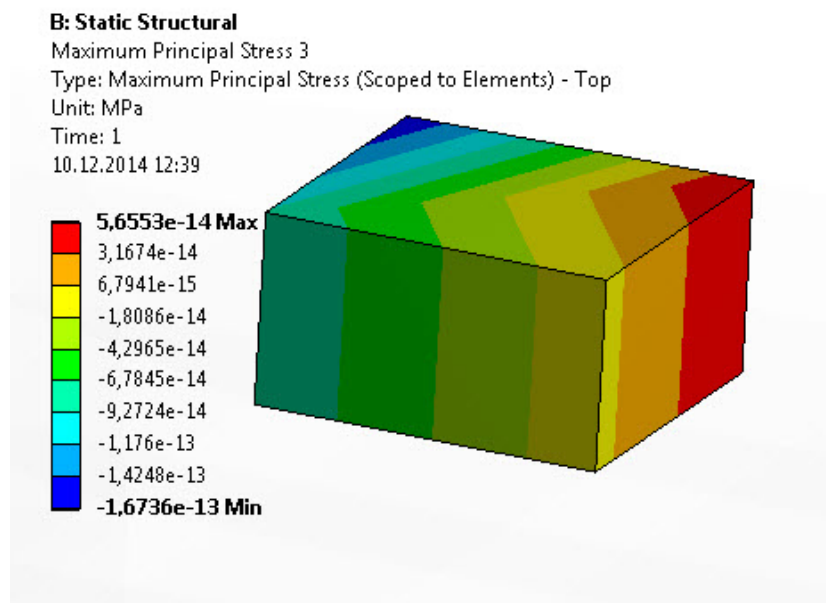


Figure 5.3: *Top Result plot of Maximal Principal Stress on one Element*

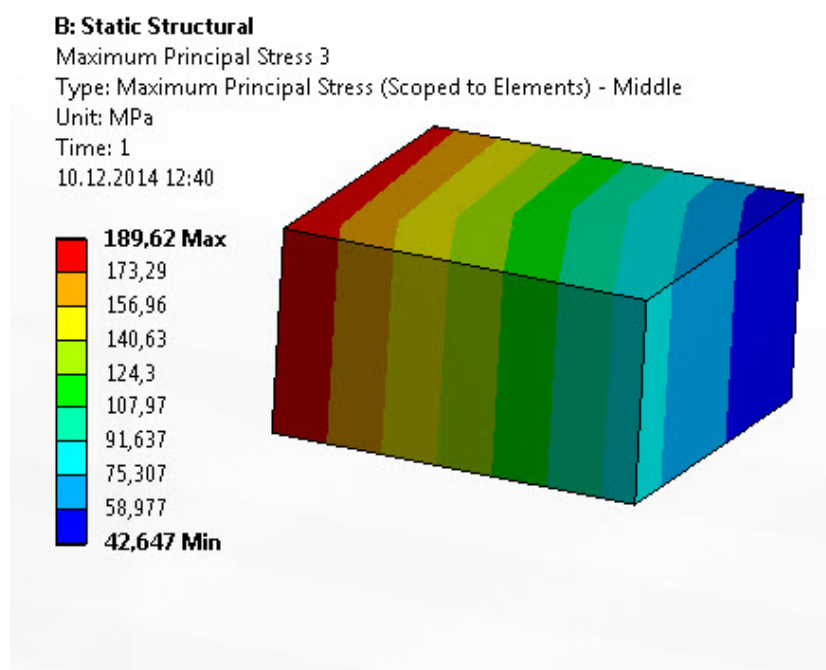


Figure 5.4: *Middle Result plot of Maximal Principal Stress on one Element*

Plotting only for one layer will give a uniform contour plot through the

imaginable thickness, while selecting a combined layered plot will give plot that varies in the thickness direction as well, as shown in figure 5.5, which displays a stress result plot with Shell option *Top/Bottom*. Note that this ability is not available in the implemented solution, as ACT only provides options for plotting one layer at the time for shell elements.

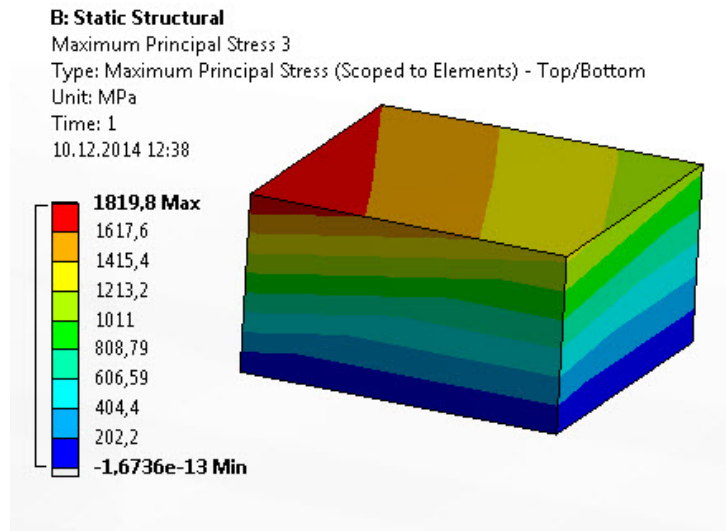


Figure 5.5: *Top/Bottom Result plot of Maximal Principal Stress on one Element*

Gathering components is an ACT method-call where the time on which the values should be retrieved from is defined firstly. For fatigue calculations, where stress/strain ranges is needed, components are retrieved twice: one for the cycle start time, and once for the cycle end time. These components are then used to calculate component ranges, which are used to eventually calculate the stress/strain range used in fatigue evaluation.

Table 5.1: Stress/ Range Calculation

$\Delta\sigma_x = \sigma_x^{EndStep} - \sigma_x^{StartStep} $		
$\Delta\sigma_y = \sigma_y^{EndStep} - \sigma_y^{StartStep} $		
$\Delta\sigma_z = \sigma_z^{EndStep} - \sigma_z^{StartStep} $	$\rightarrow \Delta\sigma_1, \Delta\sigma_2, \Delta\sigma_3 \rightarrow$	$\Delta\sigma = \max \Delta\sigma_1, \Delta\sigma_2, \Delta\sigma_3 $
$\Delta\sigma_{xy} = \sigma_{xy}^{EndStep} - \sigma_{xy}^{StartStep} $		
$\Delta\sigma_{xz} = \sigma_{xz}^{EndStep} - \sigma_{xz}^{StartStep} $	\longrightarrow	$\Delta\sigma = \Delta\sigma_{von-Mises}$
$\Delta\sigma_{yz} = \sigma_{yz}^{EndStep} - \sigma_{yz}^{StartStep} $		

Note that strain range calculations are done similar to the stress calculations as shown in table 5.1, and that only maximum principal strain ranges are an option.

5.2 Classification of Geometry

Each of the result objects are dependent on some geometry classifications. Firstly, the bodies included in the geometry scope must be defined for special cases where different S-N Data will be used for different bodies. ANSYS ACT provides geometry type information, and based on the geometry scope type, all bodies included can be found and classified.

5.2.1 Weld Definition and Directional Extrapolation in Plane Plated Welds

For plane plated weld result objects, when geometry is scoped to face or body, welds must be defined. A weld is in this solution defined as an edge that is connected to at least two bodies. When geometry is scoped to edge, this edge will automatically be defined as a weld. All welds will have knowledge about faces connected to it, which is used as a basis for extrapolation. Further, all nodes connected to welds will be defined as hot spots. Extrapolation

directions is decided from the welds connected faces, and directions depends on the hot spot itself:

- From hotspot to the left along the weld line
- From hotspot to the right along the weld line
- Normal from hotspot along all connected faces, positive direction
- Normal from hotspot along all connected faces, negative direction

For an L-Shell model, all possible directions are shown in figure 5.6

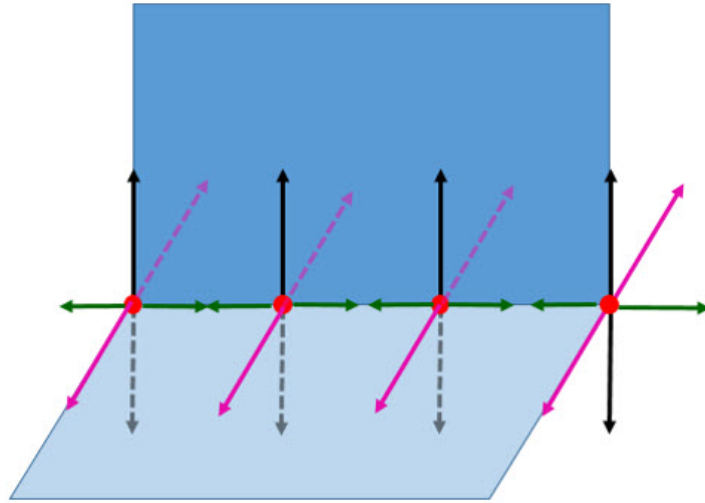


Figure 5.6: *An L shell model weld line with all potential directions. The black arrows indicate possible directions normal to the weld along the dark blue face, the pink arrows normal to the weld along the light blue surface and the green parallel to the weld line*

Based on knowledge about where hot spots are located in a weld, some directions are eliminated, as it is not conservative to extrapolate values from the weld itself as shown in figure 5.7

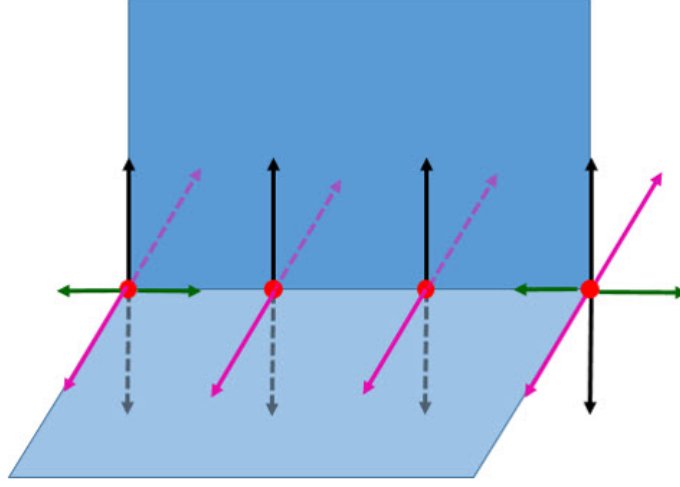


Figure 5.7: *An L shell model weld line with all potential directions after dismissal of directions pointing with the weld line in hot spots other than the weld toes*

The algorithm presented beneath provides accurate simplified description of how the potential directions are found.

Algorithm 1 Extrapolation Directions

```

procedure GETEXTDIRECTION(Mesh, geometry, hotspotId, faceId,
edgeId, weldline)
    directions = []
    faceNormal = geometry(faceId).Normal
    direction1 = crossProduct(faceNormal, weldline)
    direction2 = -direction1
    directions.append(unitVector(direction1))
    directions.append(unitVector(direction2))
    if isEndVertex(hotspot,edgeId) or isStartVertex(hotspot,edgeId) then
        direction3 = weldLine
        direction4 = -weldLine
        directions.append(unitVector(direction3))
        directions.append(unitVector(direction4))
    end if
    return directions
end procedure

```

5.2.2 Hot Spot Definition and Directional Extrapolation in Tubular Joints

In the *Tubular Joint* result objects, where input is mesh-nodes, each mesh node given as geometry input must be classified as either saddle or crown points before evaluation. Knowledge about the hot spots surrounding bodies are also needed, and with the limited time provided to solve this problem partially, this is the users responsible. Hot spot classification also demands the user to specify which of the hot spots connected bodies is the chord, and which is the brace. With this known, extrapolation directions and locations depend on the hot spot type being a crown or a saddle point, and which body the extrapolation should be done along. This can be divided into categories (see also figure 2.1):

- Crown Points
 - Extrapolation along Brace: direction is a straight line along the brace surface
 - Extrapolation along Chord: direction is a straight line along the chord surface
- Saddle Points
 - Extrapolation along Brace: direction is a straight line along the brace surface
 - Extrapolation along Chord: direction follow the curvature of the chord

Giving extrapolation directions as shown in figure 5.8

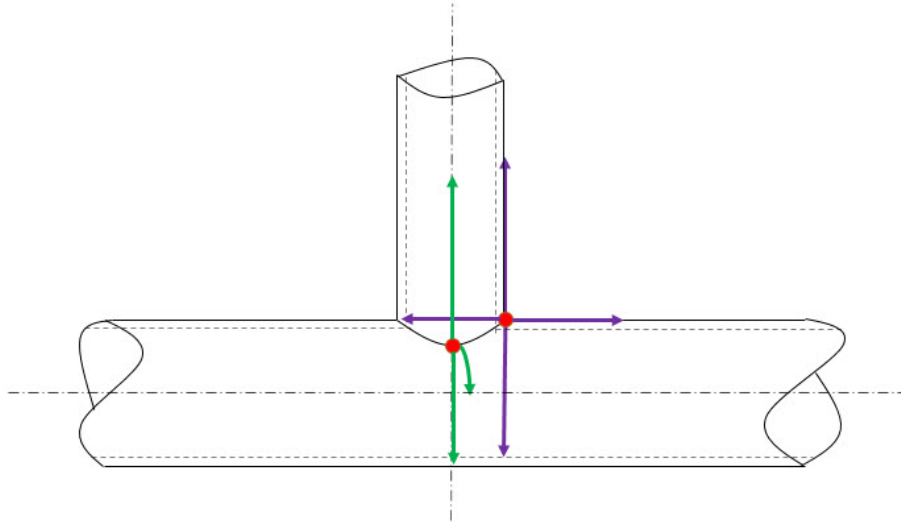


Figure 5.8: A T-joint with potential extrapolation directions. Purple lines represents direction from crown point, green from saddle point

This means that saddle point extrapolation along the chord surface is the extrapolation that differ from the straight line extrapolation method used for plane plates. This introduces a new challenge, without a straight line direction to extrapolate from, the coordinates that is desired to locate must be found otherwise.

For all other, extrapolation is conducted as before, by finding the direction in which the extrapolation should be done towards. For tubular structures, the direction of the beam can be found, and this is used as the direction

Regarding the issue of extrapolating along the chords curvature, a geometric solution is found. The arc-length between the hot spot and the desired extrapolation point is known, as well as the line from hot spot to the middle of the chord is equal to the radius, the angle (see figure 5.9) the arc-length represents can be found by equation 5.1:

$$\alpha = \frac{180^\circ \text{ArcLength}}{\pi \text{radius}} \quad (5.1)$$

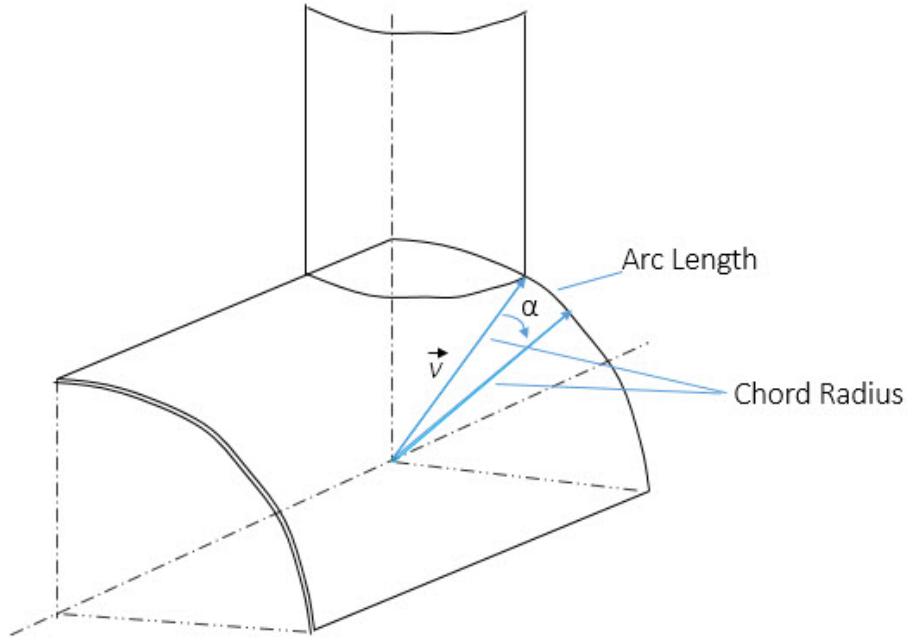


Figure 5.9: A T-joint with vector rotated α degrees found from equation 5.1, giving the extrapolation point

By rotating the vector pointing from the centerline in the chord to the hot spot, \vec{v} , α degrees the desired extrapolation location is found by rotation formula 5.2.

$$\vec{v}_{rot} = \vec{v} \cos \alpha + (\vec{k} \times \vec{v}) \sin \alpha + \vec{k} (\vec{k} \cdot \vec{v}) (1 - \cos \alpha) \quad (5.2)$$

where

\vec{v} is the vector between brace centerline and hot spot (see figure 5.9)

α is the angle to rotate vector \vec{v} (see figure 5.9)

\vec{k} is the unit vector describing the axis to rotate about

5.3 Extrapolation Points

Finding extrapolation points for straight line directional extrapolation along plane surfaces is done easily by equations:

$$pointX = hotSpot.X + a * direction.X$$

$$pointY = hotSpot.Y + a * direction.Y$$

$$pointZ = hotSpot.Z + a * direction.Z$$

where

$a = 0.5thickness$ or $1.5thickness$ for extrapolation along plane plates

$a = 0.2\sqrt{rt}$ or $0.65\sqrt{rt}$ for extrapolation along brace surface

$a = 0.2\sqrt{rt}$ or $0.4\sqrt[4]{rtRT}$ for extrapolation along chord surface from crown position

For extrapolation along chord curvature surface from saddle point, the extrapolation coordinates is found by adding the center point to the rotated vector found by equation 5.2.

Algorithm 2 describes the process of finding extrapolation locations for a hot spot in a tubular geometry.

Algorithm 2 Extrapolation Locations for Tubular Joints

procedure GETTUBULARLOCATIONS(distanceType, hotspotId, hotspot-Data, bodyData, Mesh, Geometry)

 locations =

if hotspot.Type == 'Crown Point' **then**

 braceDirection1 = braceDirection

 braceDirection2 = -braceDirection

 chordDirection1 = chordDirection

 chordDirection2 = -chordDirection

if distanceType == 'small' **then**

 chordA = 0.2 SQRT(braceRadius*braceThickness)

 braceA = chordA

else

 chordA = 0.4 SQRT(SQRT(braceRadius*braceThickness*chordRadius*chordThickness)

 braceA = 0.65 SQRT(braceRadius*braceThickness)

end if

 braceP1 = getExtCoordinates(hotspotId, braceDirection1, braceA)

 braceP2 = getExtCoordinates(hotspotId, braceDirection2, braceA)

 chordP1 = getExtCoordinates(hotspotId, chordDirection1, chordA)

 chordP2 = getExtCoordinates(hotspotId, chordDirection2, chordA)

 locations[chordId] = [chordP1, chordP2]

 locations[braceId] = [braceP1, braceP2]

else

 braceDirection1 = braceDirection

 braceDirection2 = -braceDirection

if distanceType == 'small' **then**

 chordA = chordA = 0.2 SQRT(braceRadius*braceThickness)

 braceA = chordA

else

 chordA = π *chordRadius/36

 braceA = 0.65 SQRT(braceRadius*braceThickness)

end if

 braceP1 = getExtCoordinates(hotspotId, braceDirection1, braceA)

 braceP2 = getExtCoordinates(hotspotId, braceDirection2, braceA)

 chordP1 = getExtCoordCurve(chordA, chord.Data, hotspotId, 1)

 chordP2 = getExtCoordCurve(chordA, chord.Data, hotspotId, 2)

 locations[chordId] = [chordP1, chordP2]

 locations[braceId] = [braceP1, braceP2]

end if

end procedure

5.4 Element Searching

When coordinates of the potential extrapolation point is known, and which face that location is expected to be located in, the key is to find out which element this point belongs to. By knowing this, the value at the extrapolation point can be found.

Not all point are located within the models boundary, but all points are treated similar as knowledge about whether or not they are more than potential points are unknown. Searching is done by an algorithm called ray-casting [7]

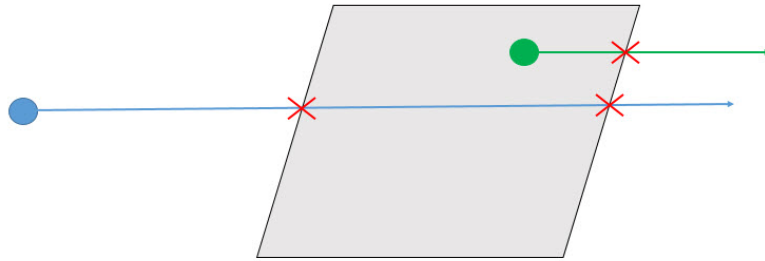


Figure 5.10: *Point in Polygon with ray-casting. The blue point is not within the polygons boundary, and the ray crossed the polygons edges twice. The green point is within the polygons boundary, and crossed the polygons edges once, an odd number which in the algorithm ray-casting defines as being inside the polygon*

Point in polygon with ray-casting is simple enough, but is only valid for 2D geometry[7]. Since shell elements do not have any thickness modelled, the 2D requirement is met as long as the ray used is in the same plane as the shell element. The ray is defined from the weld line to the point, which is in the same plane as the model if, and only if, the plate in question is entirely plane.

For tubular joints, this is not the case. The face to extrapolate from consist of several elements that are not in the same plane, though this issue arise, the direction for extrapolation is valid for all elements in the face as long as the beam has a constant radius and the mesh is fine, meaning the beam consist of elements lined in same planed-rows perpendicular to the welded joint. The challenge presented is to define a weld that is in the same plane as the element row that the element in search for is expected to be found.

For tubular joints extrapolation along the curvature, the elements between the hot spot and the desired extrapolation point do not form a same-planed row, and defining a suitable ray proved difficult. Several attempts in ray defining were done, but no general solution were found.

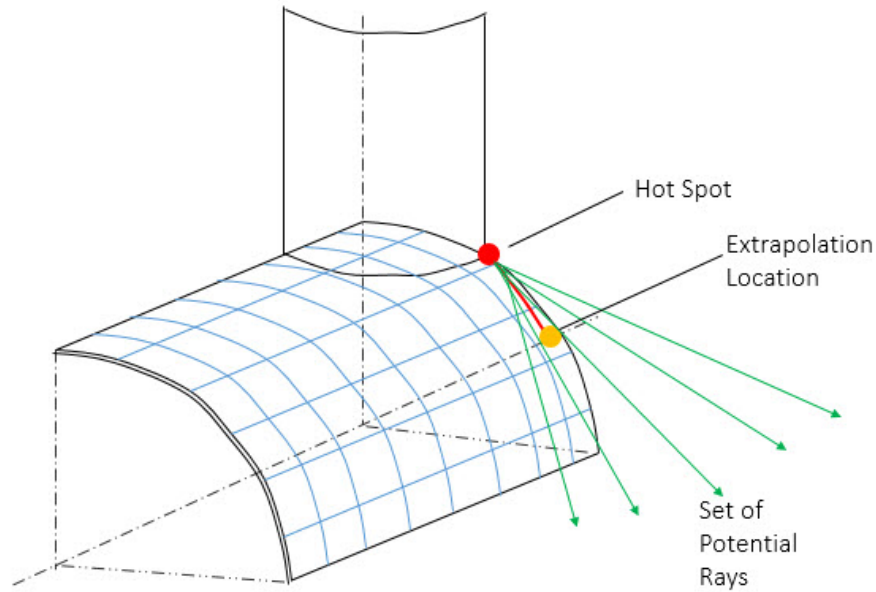


Figure 5.11: *Ray Casting Problem with extrapolation along curvature. The ray must be defined in the same plane as the element that is searched for, but as this is unknown, and assumptions regarding it is difficult to generalize, this problem was not solved*

5.4.1 Ray Casting Limitations and Handling of These

Ray Casting is an efficient algorithm, but it has problems handling cases where the point in search for is located in a corner, the ray will cross two edges in the polygon, and even though the crossing is at the same point, it will be counted twice and the algorithm will define the point as not within that polygon.

The same problem arise when the point is located on the edge of the polygon. See figure 5.12.

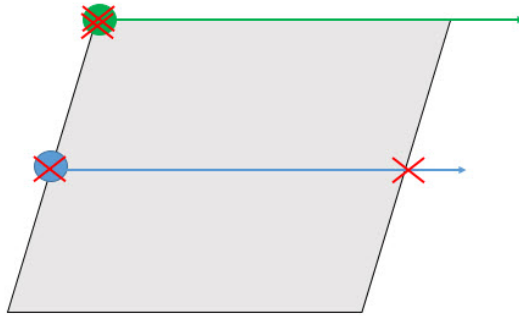


Figure 5.12: *Ray Casting limitations. Both the green and the blue point cross the polygon edges twice, and is defined as outside the polygons boundaries though both should be defined as within*

The solution is to check if the point is on one of the polygons lines before ray casting is omitted. See algorithm 3 for full procedure.

Algorithm 3 RayCasting

```
procedure GETEXTELEMENT(Mesh, int Id, point extPoint, point
rayStart)
    ray = extPoint - rayStart
    startElementId = someConnectedElementId
    searched = []
    while haveFoundElement == False do
        list[vector] elementEdges = getElementEdges(startElementId)
        if extPoint isOnLine(elementEdge) then
            haveFoundElement = True
            break
        else
            intersects = rayCasting(ray, elementEdges)
            if isOddNumber(intersects) then
                haveFoundElement = True
                break
            else
                searched.append(startElementId)
                startElementId = findNextElement(searched)
            end if
        end if
    end while
end procedure
```

When an element is rejected, the search for next potential element must be done. By using the connected elements for the element just rejected, and the distance from these elements to the point to find, searches are conducted as a combination of *Breadth-first-search*[8] and *Depth-first-search*[8]. See figure 5.13 for example of the searching procedure, and algorithm 27 for pseudo-code explanation of it.

132	123	122	121
160	159	158	157
80	56	49	50
79	36	37	38
78	35	24	11
77	34	1	3
76	33	2	4

Figure 5.13: *Outline of meshed L-Shell model. The hot spot (red dot) has a desired extrapolation point located in element id 77. The first element checked will be element id 56. This element is rejected, as the point is not within its boundary. Potential elements to check further included element ids: 80, 36 and 79 [not 160 and 159 as they belong to another face]. Element id 79 centroid is closest to the point to find, and is therefore selected as next element to check*

Algorithm 4 Element Searching

```
procedure FINNEXTELEMENTID(Mesh, int previousId, list searchedIds,
int faceId, point extPoint)
    potentialIds = []
    for cornerNodes in previousId.CornerNodes do
        elementIds = cornerNode.ConnectedElement
        for elementId in elementIds do
            if elementId not in searchedIds then
                if elementId not in potentialIds then
                    potentialIds.append(elementId)
                end if
            end if
        end for
    end for
    if potentialIds.Size != 0 then
        elementCentroid = Centroid(potentialIds[0])
        distance = SQRT(elementCentroid - extPoint)2
        nextElementId = potentialIds[0]
        for elementId in potentialIds do
            centroid = Centroid(elementId)
            thisDistance = SQRT(centroid - extPoint)2
            if thisDistance < distance then
                distance = thisDistance
                nextElementId = elementId
            end if
        end for
    else
        nextElementId = randomElementInFace
    end if
end procedure
```

Since not all potential extrapolation points are within the models boundary, a cut off at searching for correct elements has been given. When this cut off is reached, this location will be discarded and used no further.

5.5 Finding Exact Value at Coordinate Point

ANSYS ACT provides results only for nodal locations. Most often, the extrapolation point is not located on a node, and result are not directly accessed. By the use of mapping and linear shape functions[6], values at coordinate points, regardless of where in an element it is located can be found exact.

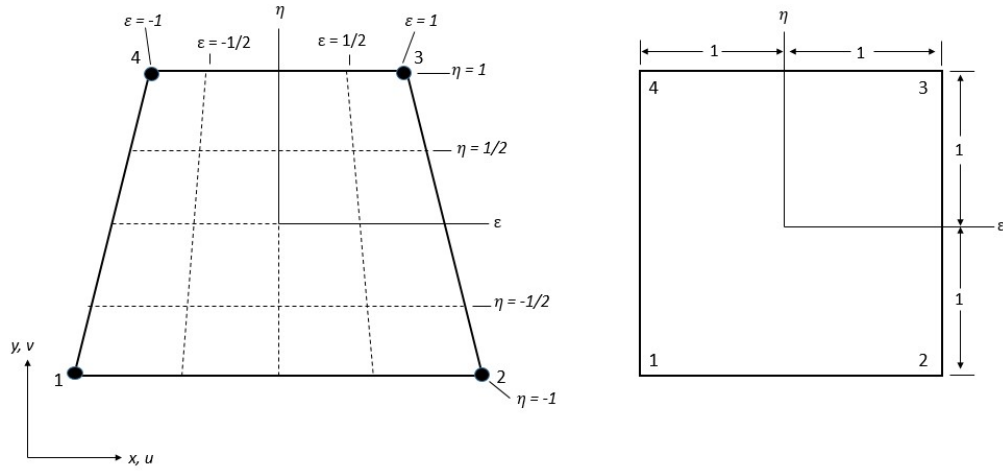


Figure 5.14: *Example of mapping of an isoparametric Four-node element in physical space[6]*

By creating a new, special defined two-dimensional coordinate system for each element we can manipulate the basic shape functions from[6]:

$$N1 = \frac{(a - x)(b - y)}{4ab} \quad (5.3)$$

$$N2 = \frac{(a + x)(b - y)}{4ab} \quad (5.4)$$

$$N3 = \frac{(a + x)(b + y)}{4ab} \quad (5.5)$$

$$N4 = \frac{(a - x)(b + y)}{4ab} \quad (5.6)$$

to[6]

$$N1 = \frac{(1 - \xi)(1 - \eta)}{4} \quad (5.7)$$

$$N2 = \frac{(1 + \xi)(1 - \eta)}{4} \quad (5.8)$$

$$N3 = \frac{(1 + \xi)(1 + \eta)}{4} \quad (5.9)$$

$$N4 = \frac{(1 - \xi)(1 + \eta)}{4} \quad (5.10)$$

where $N1, N2, N3$ and $N4$ decides corner node 1, 2, 3 and 4's influence of the coordinate set value[6]. Which will be[6]:

$$ExtValue = node_{value}^1 \times N1 + node_{value}^2 \times N2 + node_{value}^3 \times N3 + node_{value}^4 \times N4$$

where

$$N1 + N2 + N3 + N4 = 1$$

in all cases.

The mapped values, ξ and η are found by solving the system[6]:

$$\begin{pmatrix} x \\ y \end{pmatrix} = \begin{pmatrix} \sum N_i x_i \\ \sum N_i y_i \end{pmatrix} = \begin{pmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{pmatrix} * \begin{pmatrix} x_1 \\ y_1 \\ x_2 \\ y_2 \\ x_3 \\ y_3 \\ x_4 \\ y_4 \end{pmatrix} \quad (5.11)$$

where

N_1, N_2, N_3, N_4 is given by equations 5.7, 5.8, 5.9 and 5.10

x, y is the extrapolation 2D-mapped coordinates

x_i, y_i is the coordinates of the respective corner nodes

5.6 Value at Hot Spot

The extrapolation points values are known, and must be combined to get the extrapolated value at the hot spot:

$$\Delta\sigma_{hotspot} = \Delta\sigma_a - \frac{a}{b-a}(\Delta\sigma_b - \Delta\sigma_a) \quad (5.12)$$

Note that this is a simplification to the extrapolation methods described in Chapter 2 equation 2.9.

5.7 Fatigue Life

Calculating fatigue life is done according to equation 2.1 for High Cycle result objects, and equation 2.3 for Low Cycle result objects. For High Cycle result objects, $\log\bar{a}$ and m may vary depending on S-N Curve and stress range. For Low Cycle, the equation 2.3 is not linearly solvable.

5.7.1 High Cycle

For some S-N Curves, DNVGL-RP-C203 provide two $\log\bar{a}$ and two m , which is specified to be used depending on the stress range being above or below a limit. Note that this limit is not the one provided in [4] (Tables 2-1, 2-2 and 2-3, row 5 *Fatigue Limit at 10^7 cycles*) but a limit that is found by equation 5.13, which described where the two curves $\log N(\log\bar{a}1, m1)$ and $\log N(\log\bar{a}2, m2)$ intersect:

$$\log N_{limit} = \frac{\log\bar{a}2 - \log\bar{a}1}{m2 - m1} \quad (5.13)$$

When calculation fatigue, a check of the stress range is always conducted to determine which fatigue curve to use.

5.7.2 Low Cycle

Equation 2.3 is not linear, and to solve for N, some iteration-method must be applied. The solution provided use Newton-Raphson for solving the equation, see algorithm 5.

Algorithm 5 Newton-Raphson

```
procedure NEWTONRAPHSON(strainRange, emodule, strength, ductility,
N0)
    delta = 1e-4; maxIter = 100; difference = 1.; excepter = 0
    if function(10e9) >= 0 then
        N1 = 10e9
        highLife = True
    else
        if function(N0) > 0 then
            Initial Guess N0 is too low
            N0 = N0*10
        else if function(N0) < 0 then
            Initial Guess N0 is too high
            if function(N0/10) < 0 then
                Initial Guess N0 is still too high
                N0 = N0/100
            else
                N0 = N0/10
            end if
        while abs(difference) > delta do
            iter += 1
            if iter > maxIter then:
                excepter += 1
                writeToLog()
                break
            end if
            N1 = N0 - function(N0)/dFunction(N0)
            relaxationIter = 0
            while N1 < 0 do
                relaxer = 0.1-((relaxationIter*0.1)/100)
                relaxationIter += 1
                N1 = N0 - relaxer*(function(N0)/dFunction(N0))
            end while
            difference = N1-N0
            N0 = N1
        end while
    end if
    end if
    return N1, excepter, highLife
end procedure
```

Since the fatigue life calculation equation for Low Cycle is poorly defined for high strain ranges, a cut off limit at 10^9 has been set, if the strain range will give a fatigue life higher than this limit, no calculations will be performed and N is set to this cut off value.

Performing Newton-Raphson on the fatigue life equation for Low Cycle is limited in terms of inserting negative values in the equation, which can happen for high or low strain ranges where the function has a very steep slope or where the function flattens. For these values, where the function may not be convergent for the initial guess, a relaxation factor has been introduced to ensure consistent values.

As algorithm 5 indicates, the Newton-Raphson iteration method maintains an overview of results and reports to the controller when lack of convergence has occurred in addition to reporting that the cut off limit has been reached.

5.8 Combining Load Cases

Load Case combination is done quite easily, for each load case, cycles to failure(N), is calculated and eventually summed up according to equation 5.14:

$$N_{combined} = \left(\sum_k \frac{1}{N_i} \right)^{-1} \quad (5.14)$$

For each load case, damage is summed up by equation 5.15[4]:

$$Damage_{combined} = \sum_k^{i=0} damage_i \quad (5.15)$$

Chapter 6

Validation and Examples

6.1 High Cycle Screening

This example shows the steps required to use the *High Cycle Screening* result object in an analysis, and verifies the result. The geometry consist of a simple tubular K-joint as a shell element meshed multibody part, as shown in figure 6.1

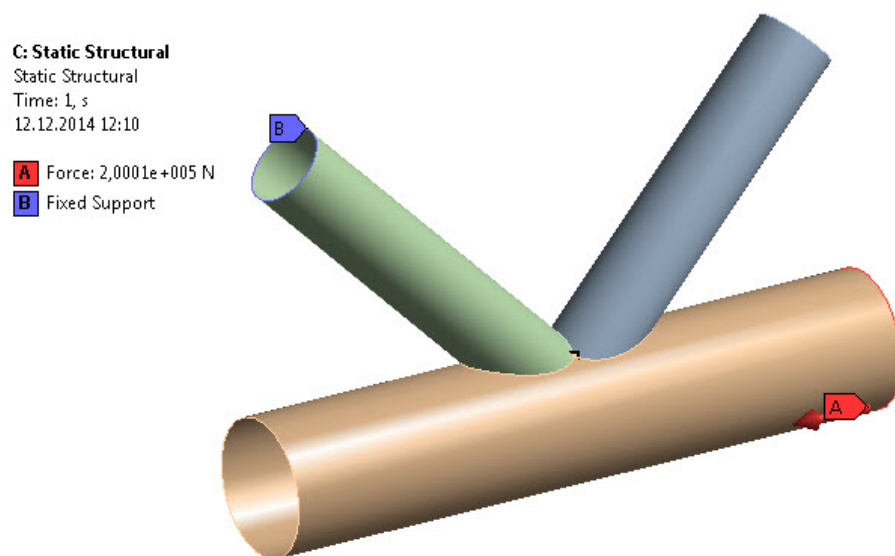


Figure 6.1: *Loads and boundary conditions for the simple Tubular K-Joint example model*

Material properties:

Material Property	Structural Steel Material
Elastic Modulus $[GPa]$	200
Poisson's ration $[-]$	0.3
Density $[kg/M^3]$	7850

And applied loading:

Load	Applied in timestep			
	1	2	3	4
Force on chord start edge	$F_x, F_y, F_z = -2e4, 0, 2e3$	$F_x, F_y, F_z = -2e4, 2e3, 0$	$F_x, F_y, F_z = -2e5, 0, 2e4$	$F_x, F_y, F_z = -2e6, 2e4, 2e4$
Fixed Support on brace end	Applied	Applied	Applied	Applied

High Cycle Screening result object is created and geometry consist of the whole model (default setting).

6.1.1 Stress Range

Looking at the stress ranges used is a effective way to establish certainty that the correct parameters is used to establish fatigue life. The result view option *Stress Range* can be chosen and compared against ANSYS default result objects. Fatigue life of the tubular K-joint van be evaluated upon both stress options *Equivalent Stress* and *Absolute Maximal Principal Stress*.

To start off, the stress ranges given will be checked. An easy check will be to evaluate a *High Cycle Screening* tool with equivalent stress as *result view* with start step 0. The stress range plotted should be equal to ANSYS default equivalent stress plot at step equal to the *High Cycle Screening* end step. Comparison of figures 6.2 and 6.3 validated the stress range used:

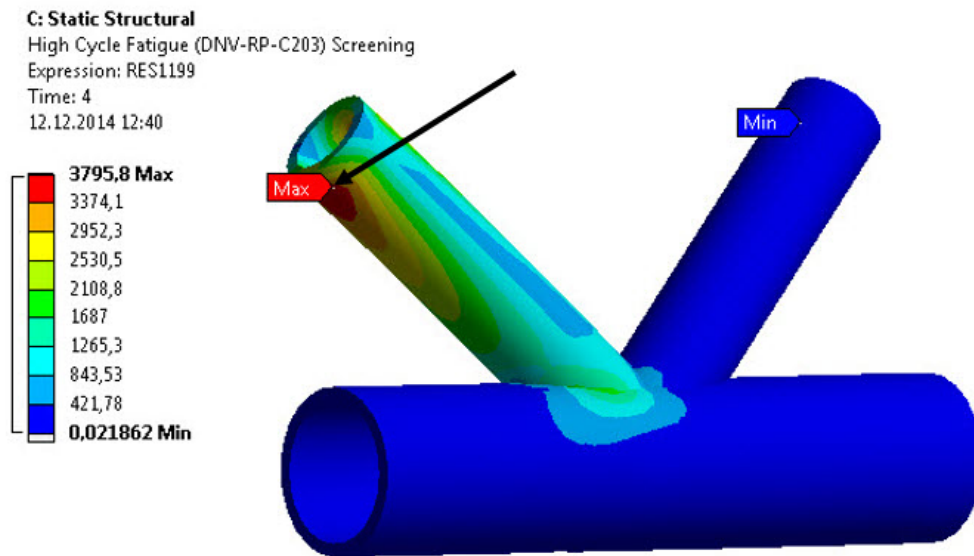


Figure 6.2: *Equivalent Stress Range generated by High Cycle Screening with start step = 0 and end step = 4. Point of maximal range marked*

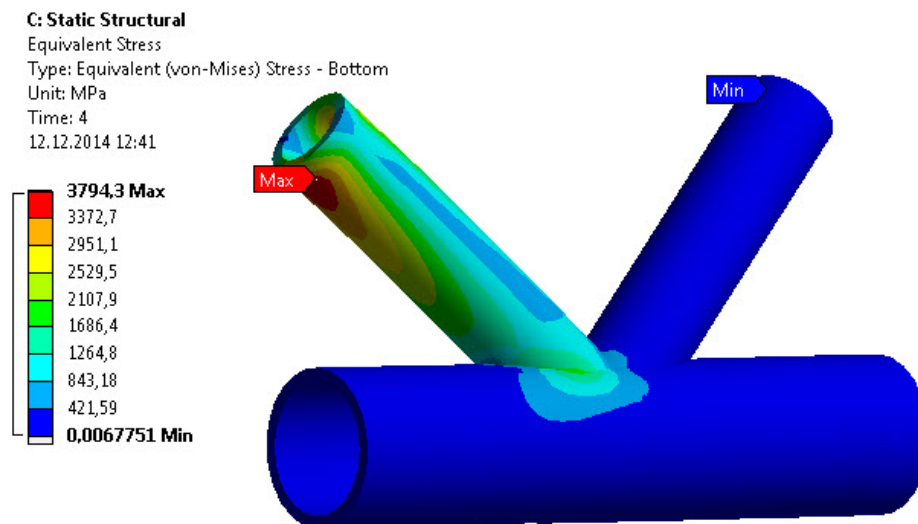


Figure 6.3: *Equivalent Stress generated by ANSYS default result object at time step = 4*

The plots in figures 6.2 and 6.3 represents the same stress, and as they are

almost exactly equal, the stress range can be verified. Be aware that some rounding errors do occur, and is the reason of slight differences.

Looking at stress ranges where start step is not equal to zero demands a little bit more research. By comparing results from a *High Cycle Screening* object with two ANSYS Equivalent stress result objects (where time step for one is equal to the start step of the *High Cycle Screening* object, and the other has time step equal to the end step), we can look at the similarities:

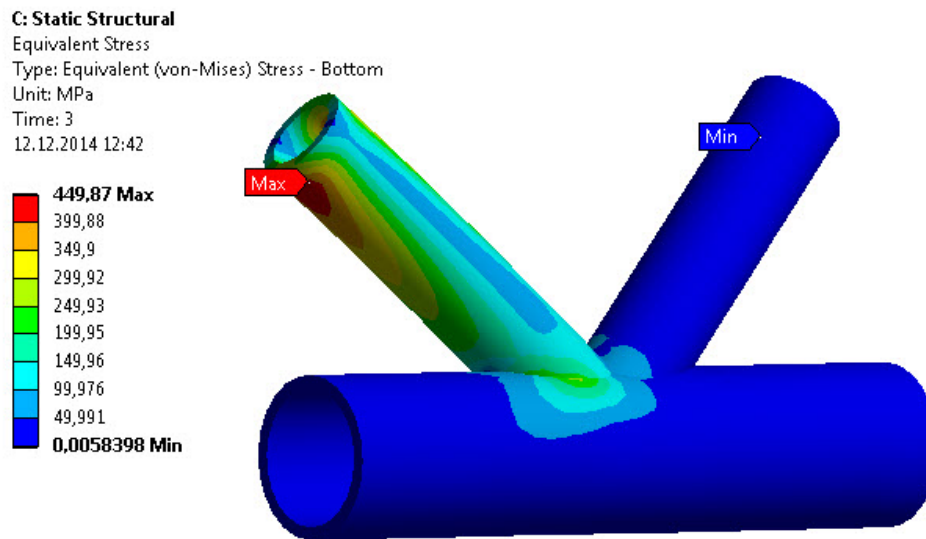


Figure 6.4: *Equivalent Stress generated by ANSYS default result object at time step = 3*

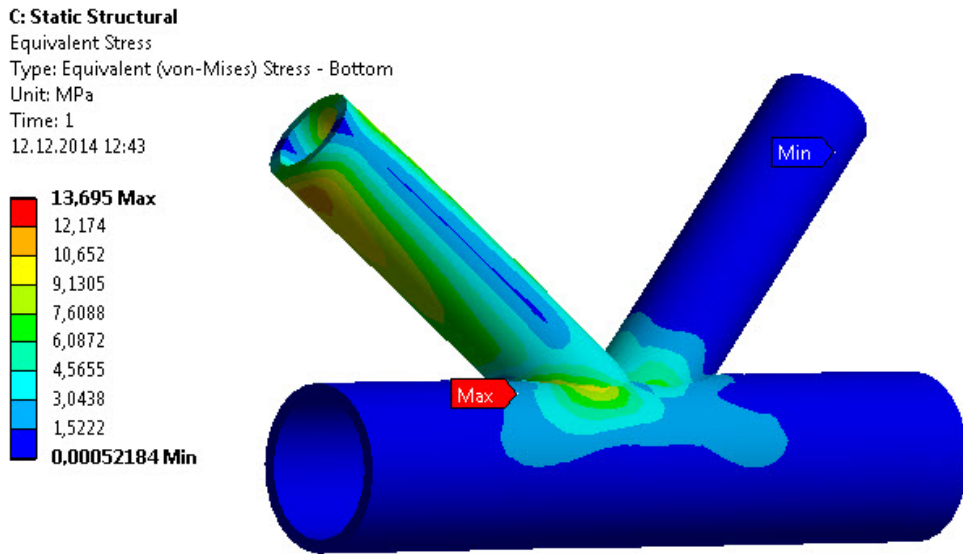


Figure 6.5: *Equivalent Stress generated by ANSYS default result object at time step = 1*

As we can see from figures 6.4 and 6.5 above, we expect a stress range that lies between the two stress values these represent. A *High Cycle Screening* object should give this. Be aware that stress ranges are calculated on component level before equivalent stress is given, and comparison with equivalent stress given by ANSYS default result object can differ, especially if the time steps have different directional applied loading.

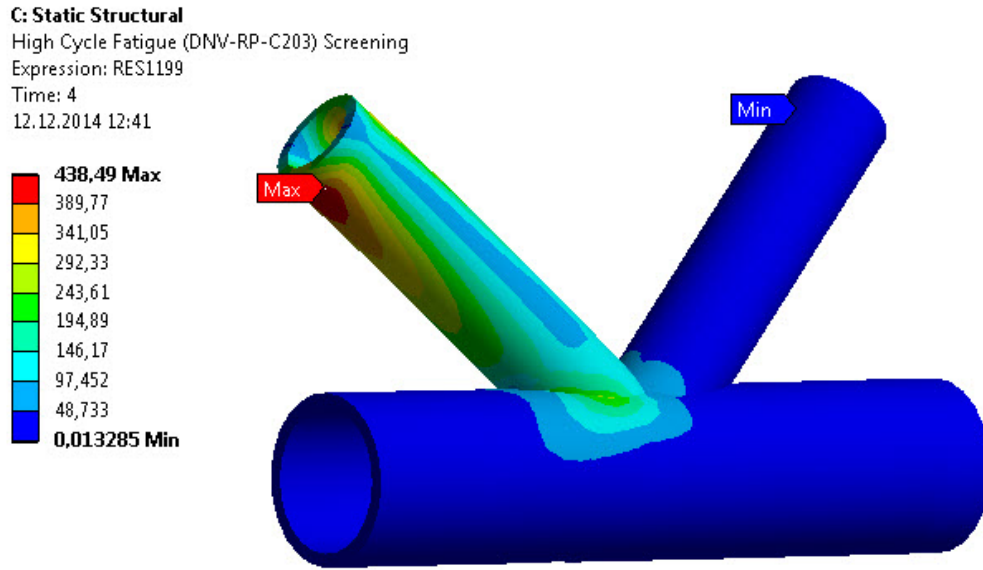


Figure 6.6: *Equivalent Stress Range generated by High Cycle Screening with start step = 1 and end step = 3*

The plot in figure 6.6 above provide reasonable stress ranges, and is therefore concluded as correct. The same procedure for validating absolute maximal principal stress ranges can be conducted, but as this type combines $\Delta\sigma_1$, $\Delta\sigma_1$ and $\Delta\sigma_3$, by taking the highest absolute value, a comparison must be done on all principal stresses, as some areas might have higher absolute $\Delta\sigma_1$ and other have higher absolute $\Delta\sigma_3$ and so on. Calculation of stress ranges is done equally for both stress options on component level, and with validation of equivalent stress, it can be assumed that maximal principal stress is correct as well.

Note that comparison of stress ranges must be done on the same level for shell elements: top, bottom or middle.

6.1.2 Fatigue Life

Now that the stress ranges have been established as correct, fatigue life can be validated. With the stress ranges available, and the S-N Curve known, the validation can be done by selecting a point and insert the stress range in the equation for fatigue, with the assumption that stress ranges used is correct.

Figures 6.7 and 6.8 presents an evaluated *High Cycle Screening* result object with property settings in figure 6.7.

Details of "High Cycle Fatigue (DNV-RP-C203) Screening"	
[-] Scope	
Scoping Method	All Bodies
Shell	Bottom
[-] Definition	
Result View	Logarithm Of Cycles To Failure To Base 10
Stress Option	Equivalent Stress
Load Case Combination	No
Start Step	0
End Step	4
By	Time
<input type="checkbox"/> Display Time	Last
[-] Integration Point Result	
Display Option	Averaged
[-] S-N Class	
S-N Data Equal For All Bodies?	Yes
Environment	Air
S-N Curve	HighStrengthSteel
Thickness Modification Factor = t/t_{ref}	1
Temperature over 100 Celsius?	No
[-] Results	
<input type="checkbox"/> Minimum	0,62349
<input type="checkbox"/> Maximum	25,362
Minimum Occurs On	Surface
Maximum Occurs On	Surface

Figure 6.7: Details view of High Cycle Screening. Chosen output is Logarithm of Cycles to Failure to Base 10

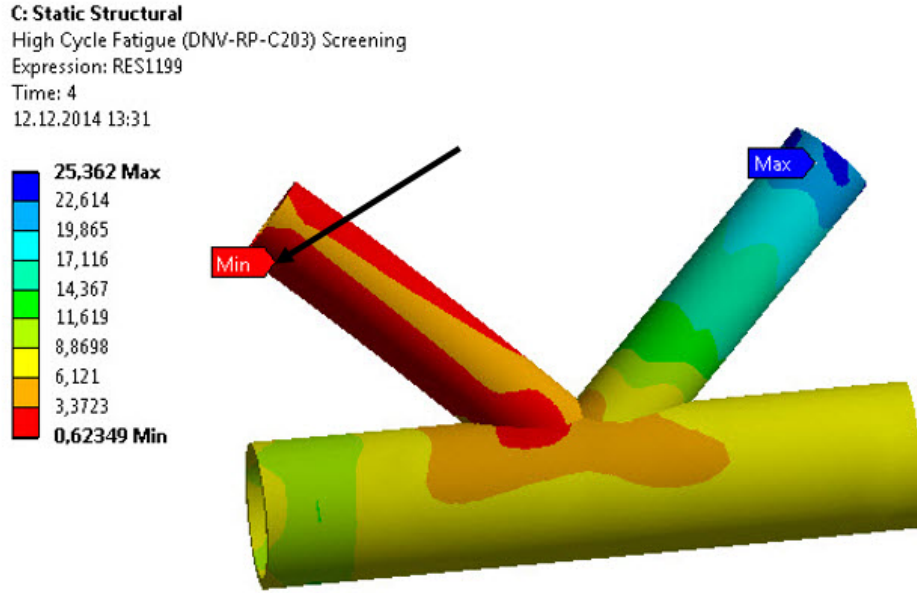


Figure 6.8: *Logarithmic Fatigue Life plotted results for High Cycle Screening result object with property settings as in figure 6.7*

To evaluate the results, the equivalent stress from result view = *Stress Range* (see figure 6.2) with the same properties as shown in figure 6.8 at the node of minimum number of cycles to failure.

$$\log N = \log \bar{a} - m \log \left(\Delta \sigma \left(\frac{t}{t_{ref}} \right)^k \right)$$

with S-N Curve *High Strength Steel* and environment *Air* where:

$\log \bar{a} = 17.446$ (from DNV-RP-C203 S-N Curve ISJjhujhgsfds)

$m = 4.7$ (from DNV-RP-C203 S-N Curve ISJjhujhgsfds)

$k = 0$ (from DNV-RP-C203 S-N Curve ISJjhujhgsfds)

$\Delta \sigma = 3795.8$ (extracted stress range at node with minimum $\log N$ value, see figure 6.2)

$thicknessModificationFactor = 1$ (user input)

which gives

$$\log N = 17.446 - 4.7 * \log(3795.8 * (1)^0) = 0.6233$$

which is the expected value given by the result object, see figure 6.8 marked point. The accumulated damage is then given by selecting Damage as result view and inserting a number of cycles, and output as given in figure 6.9.

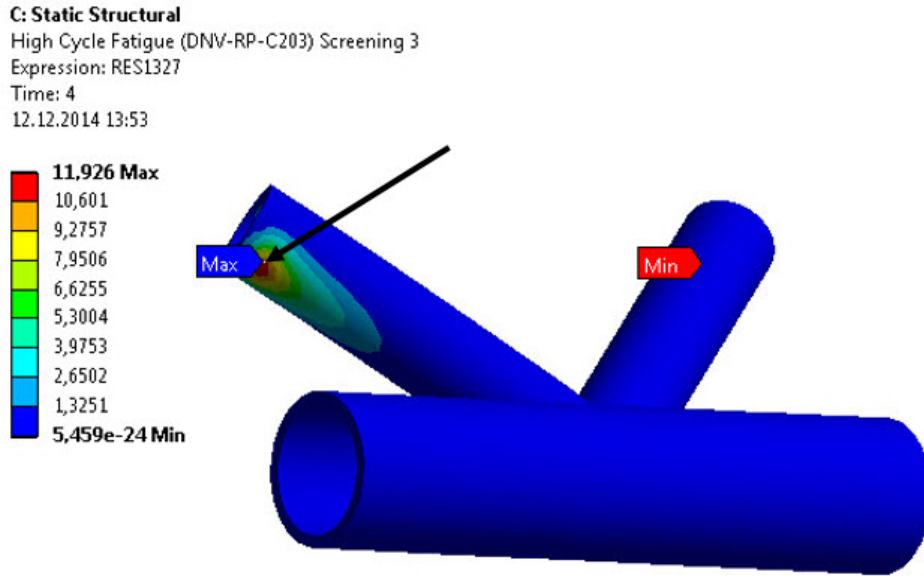


Figure 6.9: *Accumulated Damage plotted results for High Cycle Screening result object with property settings as in figure 6.7. Number of Cycles is set to 50*

$$D = \frac{NumberOfCycles}{NumberOfCyclesToFailure} = \frac{50}{10^{0.6233}} = 11.9$$

The accumulated damage obtained by hand calculation is equal to the one obtained by the result object in figure 6.9 at the point of maximal stress range (figure 6.2) and minimum cycles to failure (figure 6.8). Results are verified.

6.2 Low Cycle Screening

The same model as used in *High Cycle Screening* example is used here. See section 6.1 for information about the model.

6.2.1 Strain Range

Strain range obtaining is equal to stress range obtaining on component level. And strain ranges used is always the absolute maximal principal strain range. We compare:

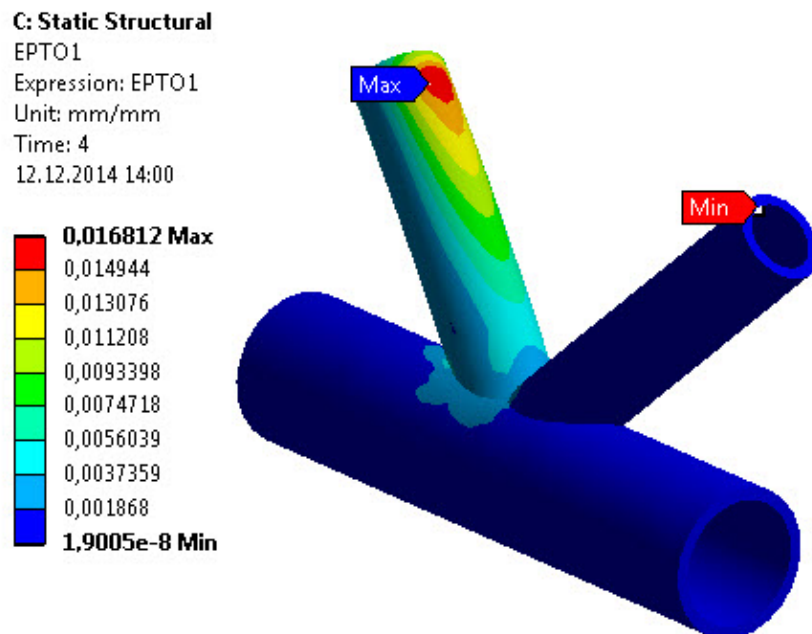


Figure 6.10: *Principal strain, EPT01, plotted in ANSYS by default result object at time step = 4*

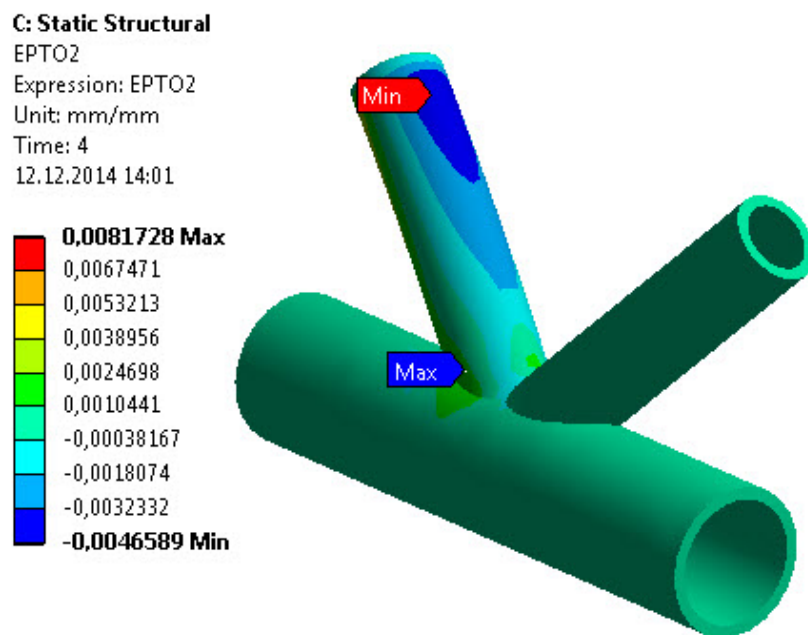


Figure 6.11: *Principal strain, EPTO2, plotted in ANSYS by default result object at time step = 4*

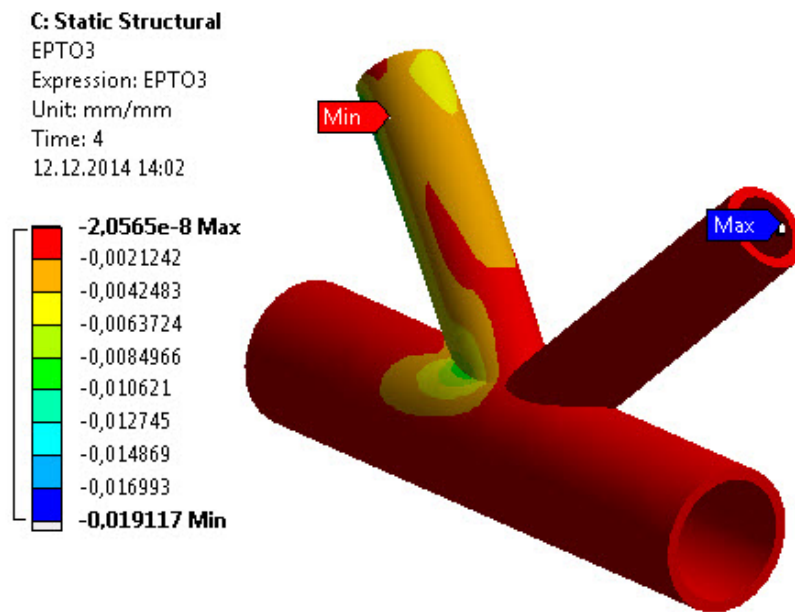


Figure 6.12: *Principal strain, EPTO3, plotted in ANSYS by default result object at time step = 4*

It is expected that the stress range given by the *Low Cycle Screening* object is similar to EPTO1 plot, figure 6.10, with some elements from the other two figures 6.11 and 6.12

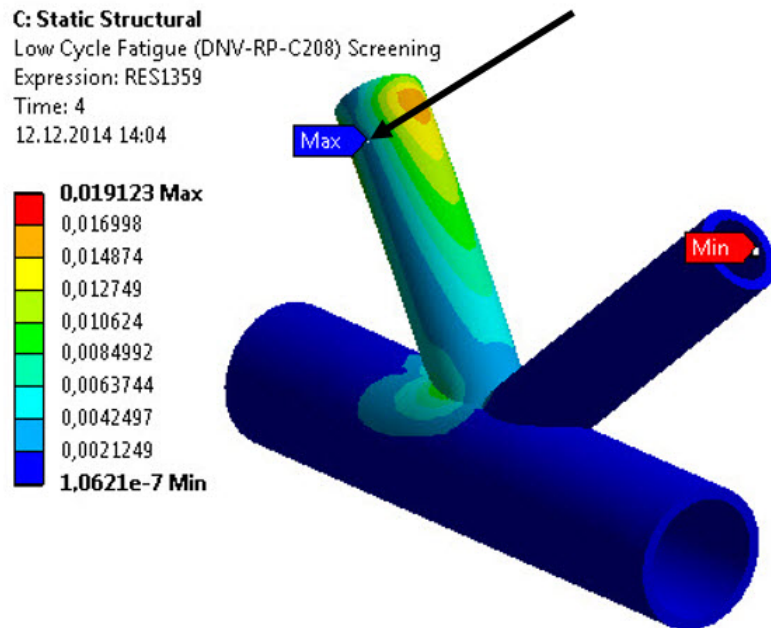


Figure 6.13: *Maximum Principal Strain Range generated by Low Cycle Screening with start step = 0 and end step = 4. Point of maximal value marked*

By probing some areas in figures 6.10, 6.11, 6.12 and 6.13, it becomes clear that the values are extracted correctly. Be aware that some rounding errors do occur, and is the reason of slight differences.

Note that comparison of strain ranges must be done on the same level for shell elements: top, bottom or middle.

6.2.2 Fatigue Life

Now that the strain ranges have been established as correct, fatigue life can be validated. With the strain ranges available, and the ϵ -N Curve known, the validation can be done by selecting the point minimum cycles to failure and insert the stress range in the equation for fatigue at the same point (see figure 6.13), with the assumption that strain ranges used are correct. Figures 6.14 and 6.15 presents an evaluated *Low Cycle Screening* result object with property settings in figure 6.14.

Details of "Low Cycle Fatigue (DNV-RP-C208) Screening"	
Scope	
Scoping Method	All Bodies
Shell	Bottom
Definition	
Result View	Logarithm Of Cycles To Failure To Base 10
Load Case Combination	No
Start Step	0
End Step	Last
By	Time
<input type="checkbox"/> Display Time	Last
Integration Point Result	
Display Option	Averaged
ϵ-N Class	
ϵ -N Data Equal For All Bodies?	Yes
Environment	Air
Temperature	22 [C]
Advanced	
Newton Raphson Initial Guess	1000
Results	
<input type="checkbox"/> Minimum	2,0302
<input type="checkbox"/> Maximum	10,
Minimum Occurs On	Surface
Maximum Occurs On	Surface

Figure 6.14: *Details view of Low Cycle Screening. Chosen output is Logarithm of Cycles to Failure to Base 10*

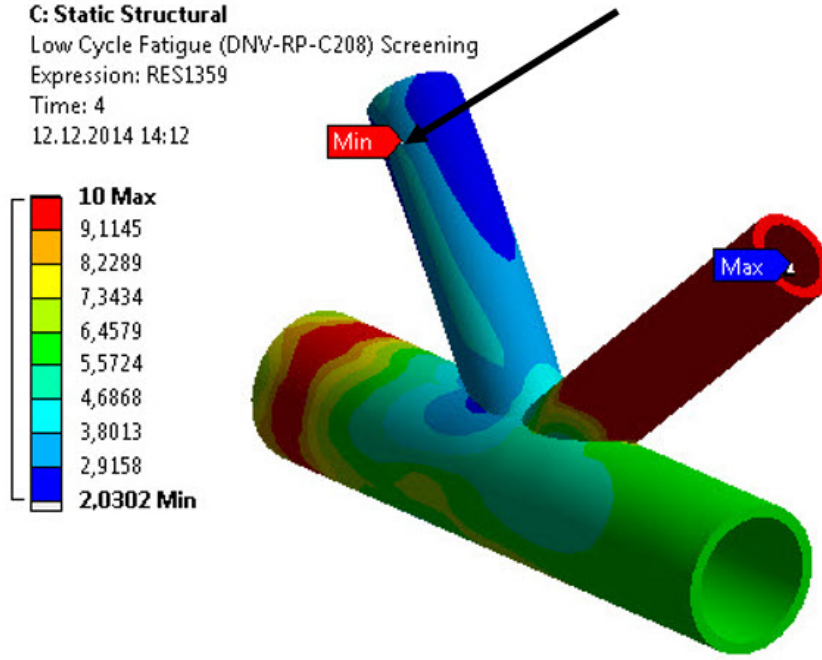


Figure 6.15: *Logarithmic Fatigue Life plotted results for Low Cycle Screening result object with property settings as in figure 6.14*

To validate the result, the stress range from result view *Strain Range* with the same properties as shown in figure 6.13 at the node with of minimum number of cycles to failure is extracted and inserted in the equation

$$\frac{\Delta\epsilon_l}{2} = \frac{\sigma'_f}{E}(2N)^{-0.1} + \epsilon'_f(2N)^{-0.43} \quad (6.1)$$

with environment *Air* we have

$$\sigma'_f = 175 \text{ [MPa]}$$

$$\epsilon'_f = 0.091 \text{ [-]}$$

$$E = 200000 \text{ [MPa]}$$

by inserting the calculated number of cycles:

$$\frac{\Delta\epsilon_l}{2} = \frac{175}{200000} * (2 * 10^{2.0302})^{-0.1} + 0.091 * (2 * 10^{2.0302})^{-0.43} = 9.5 * 10^{-3}$$

and

$$\Delta\epsilon_l = 2 * 9.5 * 10^{-3} = 0.019$$

which is the expected strain value with the number of cycles inserted. Accumulated damage is given by selecting Damage as result view and inserting number of cycles.

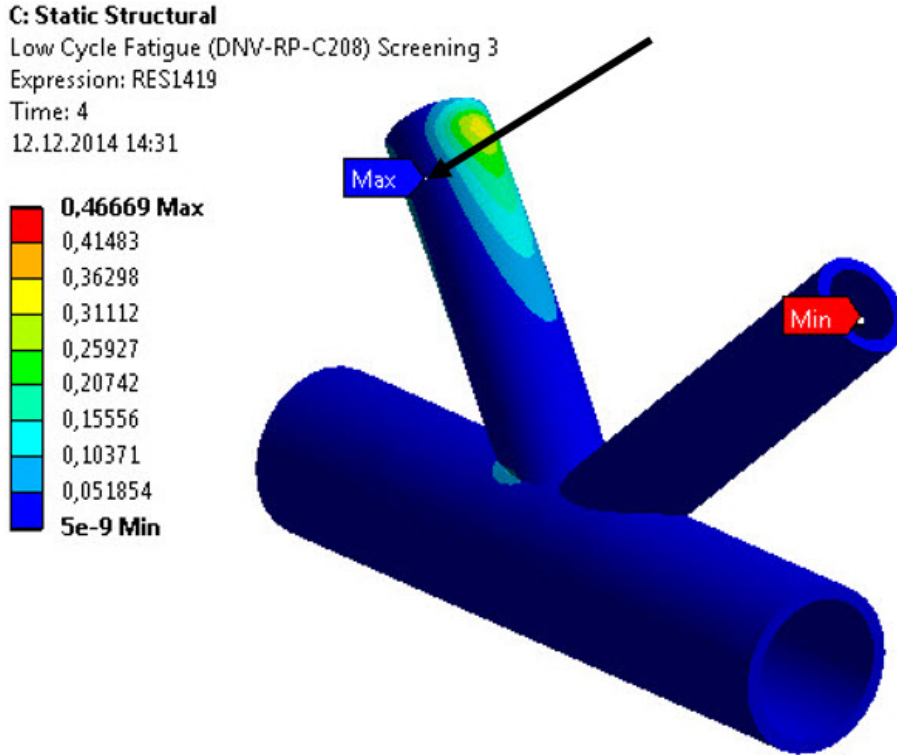


Figure 6.16: *Accumulated Damage plotted results for Low Cycle Screening result object with property settings as in figure 6.14. Number of Cycles is set to 50*

$$D = \frac{NumberOfCycles}{NumberofCyclestoFailure} = \frac{50}{10^{2.0302}} = 0.46$$

The accumulated damage obtained by hand calculations is equal to the one obtained by the result object in figure 6.16. Results are verified.

6.3 High Cycle Plane Plated Weld

This example shows the steps required to use the High Cycle Plane Plated Weld result object in an analysis, and verifies the result.

The geometry consist of a simple L welded plate without the weld itself included in the shell-element model as shown in figure 6.17

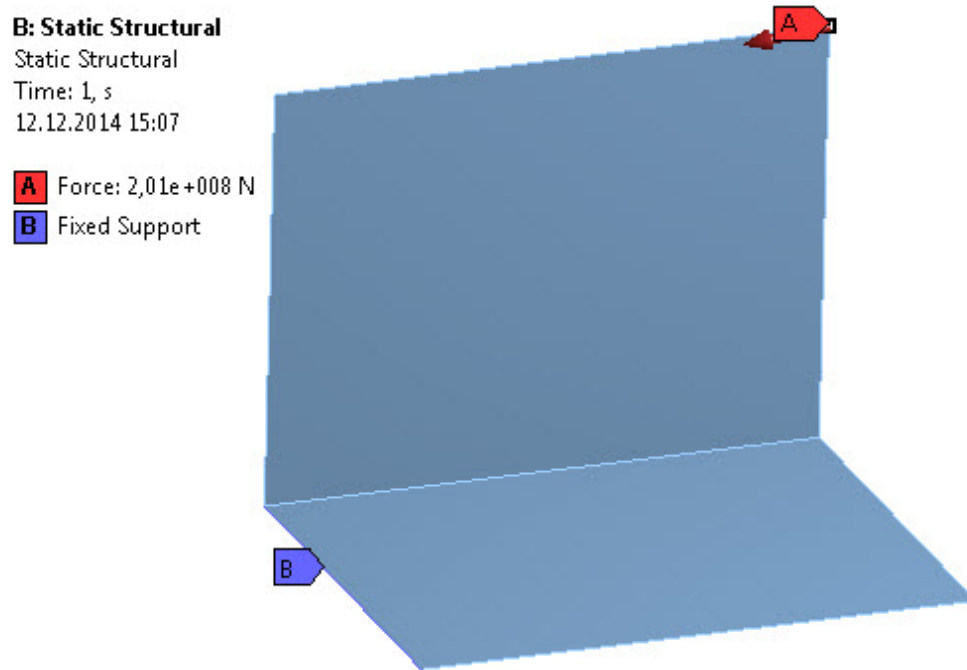


Figure 6.17: *Loads and Boundary Conditions for the simple simple L-shell example model*

Material Properties:

Material Property	Structural Steel Material
Elastic Modulus [GPa]	200
Poisson's ration [-]	0.3
Density [kg/M ³]	7850

And applied loading:

Load	Applied in timestep
Force on upper right corner	$F_x, F_y, F_z = 0, -2e7, 2e8$
Fixed Support on left edge	Applied

Note that this model only have one time step and that the model only consist of one body. For the *Plane Plated Weld* result objects, this means that the only input that will give results is when geometry is scoped to edge.

6.3.1 Extrapolated Stress Range

Validating the extrapolated stress range, with the assumption that stress ranges calculated is correct based on section 6.1.1, can be done manually if knowledge about the geometry is sufficient. For the L-Shell, with thickness = 1000 [mm], we can extrapolate values manually by probing the stress range values at the desired locations:

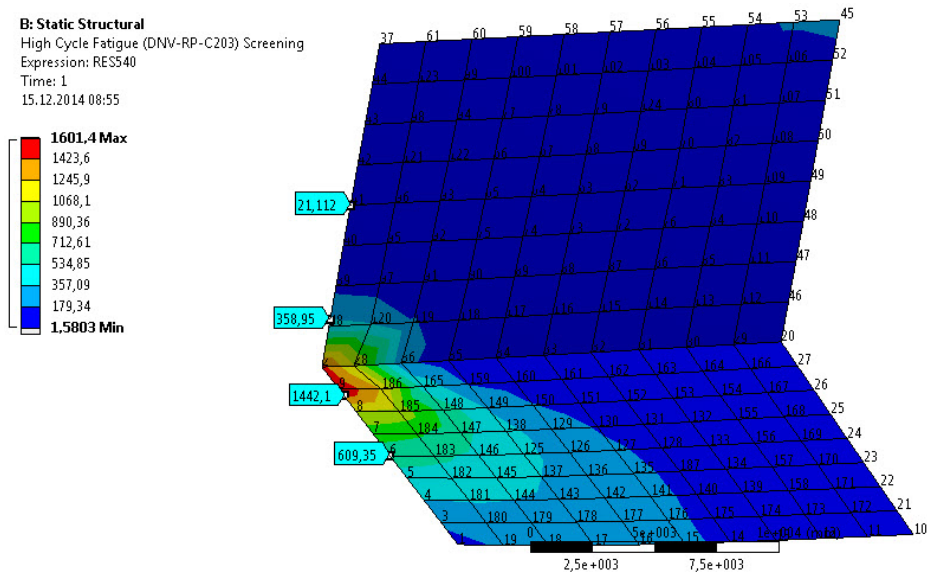


Figure 6.18: *Probed maximal principal strains generated from Low Cycle Screening result object for extrapolation of values to hot spot at end of edge, $Id = 2$*

B: Static Structural

High Cycle Fatigue (DNV-RP-C203) Screening 2

Expression: RES572

Time: 1

15.12.2014 09:02

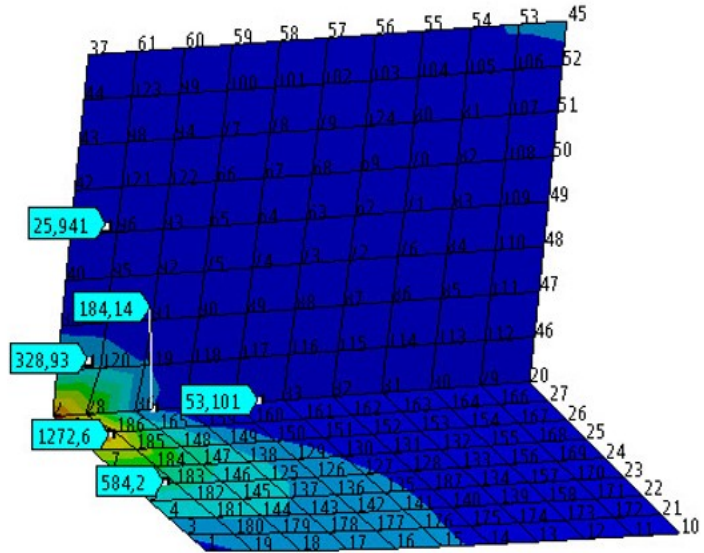
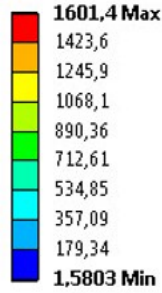


Figure 6.19: Probed maximal principal strains generated from Low Cycle Screening result object for extrapolation of values to hot spot in edge, Id = 28

Hot Spot Id	Extrapolation Direction	$\Delta\sigma_a$ [MPa]	$\Delta\sigma_b$ [MPa]	$\Delta\sigma_{hs}$ [MPa]
2	Up (in positive y-direction)	358.95	21.112	527.869
2	Down (in positive x-direction)	1442.1	609.35	1858.475
28	Up (in positive y-direction)	328.93	25.941	480.42
28	Down (in positive x-direction)	1272.6	584.2	1616.8
28	Right (in negative z-direction)	184.14	53.101	249.66

Resulting in the extrapolated values

$$\Delta\sigma_{HS2} = 1858.475 MPa$$

$$\Delta\sigma_{HS28} = 1616.8 MPa$$

Giving fatigue life with S-N Curve High Strength Steel:

$$\text{Log}(N)_{HS2} = 17.446 - 4.7 \log(1858.475) = 2.08$$

$$\text{Log}(N)_{HS28} = 17.446 - 4.7\log(1616.8) = 2.23$$

Though these values found by hand calculation is not exactly the same as the values obtained by the result object, they correspond in difference of location. This correspondence and the small difference in result points to human error in calculation, which can be traced to the manual extrapolation. As it is not possible to extract values at coordinate points in ANSYS Mechanical, the extrapolation points found manually are not exact, and the value therefore not entirely correct. With this in mind, the results are concluded as verified.

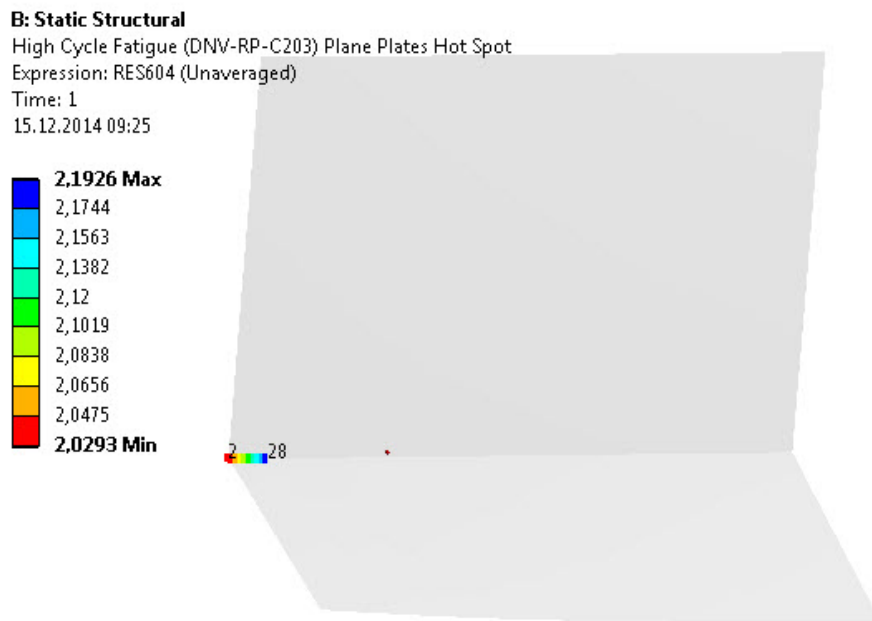


Figure 6.20: *Logarithmic Fatigue Life plotted on part edge with environment Air and S-N Curve High Strength Steel, generated by High Cycle Plane Plated Weld*

As this result object and the *High Cycle Screening* result objects calculates fatigue life equally with stress ranges known, based on section 6.2.1, the fatigue life can be assumed correct.

6.4 Low Cycle Plane Plated Weld

As *Low Cycle Plane Plated Weld* reuse procedures validated in the previous sections it can be assumed:

- The Strain Ranges calculated are correct, see section 6.2.1
- The extrapolates Strain Ranges are correct, see section 6.3.2
- Fatigue Life calculations are correct, see section 6.2.2

and thus that this result object performs correct fatigue analysis.

6.5 High/Low Cycle Tubular Joint

As these result objects are not completed, verification of these will not be performed.

Chapter 7

Conclusion

Four of the six result objects in the *DNV Fatigue Toolbar* has been tested for numerous shell-models, and the extensive testing has resulted in a solution that behaves as expected, both in terms of user interference and accuracy of results returned.

The two *Screening* objects has been added to a larger toolkit created by EDR Medeso 2014, *Oil & Gas Toolkit*. The *Oil & Gas Toolkit* has been subjected to extensive testing, both internal and external as the Toolkit has matured. As such, the two screening objects has been tested by several instances, and corrected as testers found bugs, issues and enhancements.

The two *Plane Plated Weld* result objects is based on the fatigue life calculation and setup of the screening objects, and has been implemented parallel to the *Oil & Gas Toolkit* test-rounds, and even though these result objects did not participate in the test-rounds, the issues found almost always applied in *Plane Plated Weld* result objects as well, and were fixed for all four result objects simultaneously.

The two *Tubular Joint* result objects were implemented after the extensive test-round were complete and all issues discovered in the test period were resolved. Unfortunately, the time available at this point, was not sufficient to complete the implementation for these two result objects.

7.1 Further Work

For future work, further development of the *Tubular Joint* result objects is needed. Tubular joints introduce exceptional handling of geometry, and is therefore difficult to automatize. The proposed setup that is the result objects for the time being is not intuitive for users, has the limitation of not being able to plot results on model in addition to depend on users ability to classify crown and saddle points as well as chord and brace.

A more suitable solution, though more complex, would include:

- Geometry Scope set equal to the Plane Plate Weld solution were weld lines are either given as input or recognized by the extension
- Chord and Brace is classified by the extension without user interference
- Saddle and Crown points are classified by the extension without user interference

Changing the geometry scope to non-mesh entities will give the opportunity to plot results on model and limit user input errors by locating and classifying hot spots without the user interference.

In addition, many further combinations of improved solution can be recommended:

- Combine High Cycle and Low Cycle Fatigue calculations, where $Damage = Damage_{highCycle} + Damage_{lowCycle}$
- Combine the 3 main categories so that only one evaluation is enough and will give accurate results in all parts of the models geometry: hot spot method is conducted on all nodes defined as hot spots (all nodes lying on an edge that is connected to two or more bodies) and screening method is done otherwise.

Bibliography

- [1] S. Suresh, *Fatigue of Materials*. Cambrigde University Press, 2nd edition, 1998.
- [2] Bjørn Haugen and Tore Holmås *Fatigue Assessment of Offshore Wind Turbines - Some Analysis Examples*. IPM, NTNU 2011.
- [3] Tore Holmås *FAT31 Theory*. Virtual Prototyping, 2009.
- [4] *Fatigue Design of Offshore Steel Structures, Recommended Practice DNVGL-RP-C203* Technical Report, Det Norske Veritas og Germanischer Lloyd, June 2014
- [5] *Determination of Structural Capacity by Non-linear FE analysis Methods, Recommended Practice DNV-RP-C208* Technical Report, Det Norske Veritas, June 2013.
- [6] R. D. Cook, D. S. Malkus, M. E. Plesha, R. J. Witt, *Concepts and Applications of Finite Element Analysis*. University of Wisconsin-Madison, 4th edition, 2002.
- [7] D. R. Finley, *Point-In-Polygon Algorithms - Determining Whether a Point is Inside a Complex Polygon*. Cambrigde University Press, 2nd edition, 2007.
- [8] T. H. Cormen, C. E. Leiserson, R- L- rivest. C. Stein, *Introduction to Algorithms*. Massachusetts Institute of Technology, 3rd edition, 2009.
- [9] ANSYS, Inc., *ANSYS Documentation, Mechanical Applications, Mechanical User's Guide*. User Manual, SAS IP, Inc, 2013.
- [10] *ANSYS Workbench Application Customizatopn Toolkit Developer's Guide*. Developers Guide ANSYS Inc., November 2013.
- [11] *ANSYS Workbench Application Customizatopn Toolkit Reference Guide*. Developers Guide ANSYS Inc., November 2013.

- [12] http://www.dnv.no/mer_om_dnv/profile/om_oss/index.asp. DNV GL, December 2014.
- [13] [http://en.wikipedia.org/wiki/Fatigue_\(material\)](http://en.wikipedia.org/wiki/Fatigue_(material)) Wikipedia.org, December 2014.

Appendices

Toolbar Usage

DNVV Fatigue Toolbar is denoted as an extension and a binary version is build named *DNVFatigueToolbar.wbex*

Installing the toolbar

To install an ANSYS ACT extension in binary version, in ANSYS version 15, open ANSYS Workbench 15. On the project page, navigate to *Extensions* -> *Install Extension...*

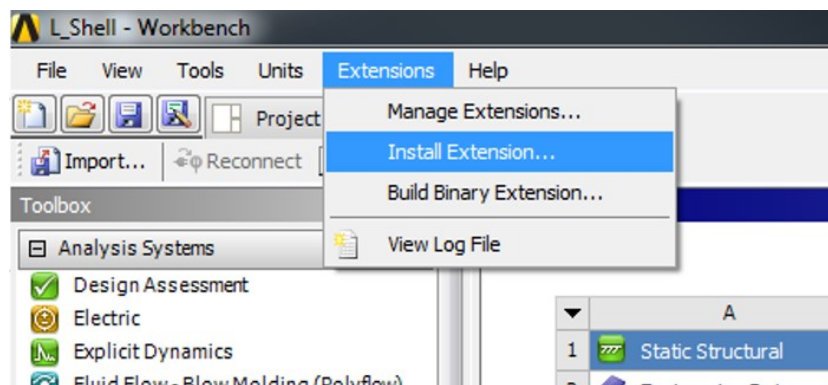


Figure 1: *Installing an extension in ANSYS Workbench*

And selecting the binary extension file (.wbex file) in the browsing window. When the extension is successfully installed, a pop up window will appear:

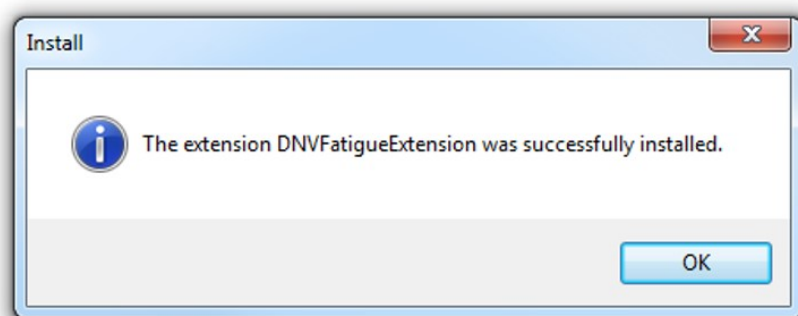


Figure 2: *Successfully installed extension in ANSYS Workbench*

The extension can now be used in any project by activating it in *Extensions* -> *Manage Extensions* -> select the binary extension file (name of the .wbex file). Note that version number is noted in the rightmost row.

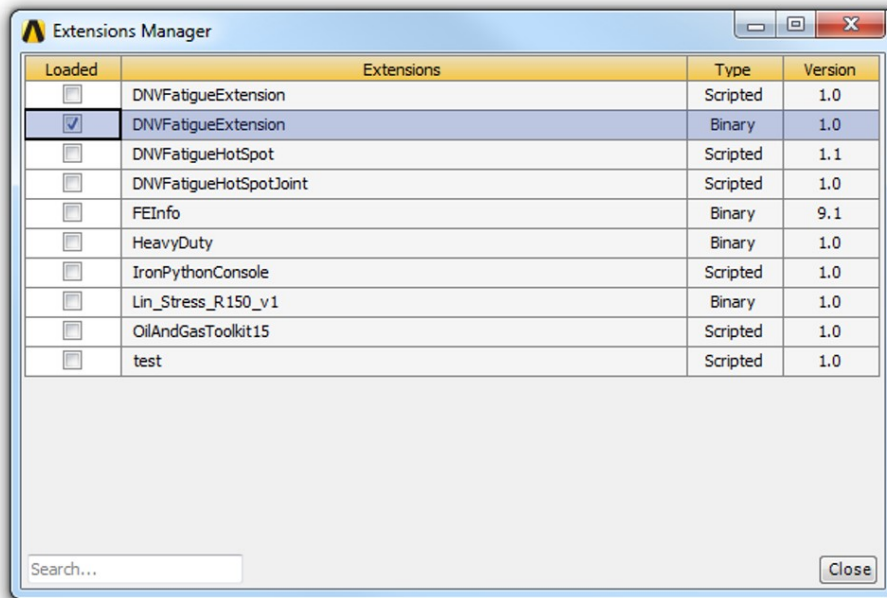


Figure 3: *Activate extension to use it in a project*

By selection the extension, as shown in figure 3, the extension will be available in ANSYS Mechanical for the project the next time it is opened. The extension will be active for the project as long as it is saved, and can be removed by deselecting it again in *Extension Manager*

Installing the toolbar, scripted version

To use a scripted version of an extension, make sure that ANSYS Workbench known where the extension script is stored by *Tools* -> *Options* -> *Extensions* and copy the address of the folder where the extensions xml file is stored.

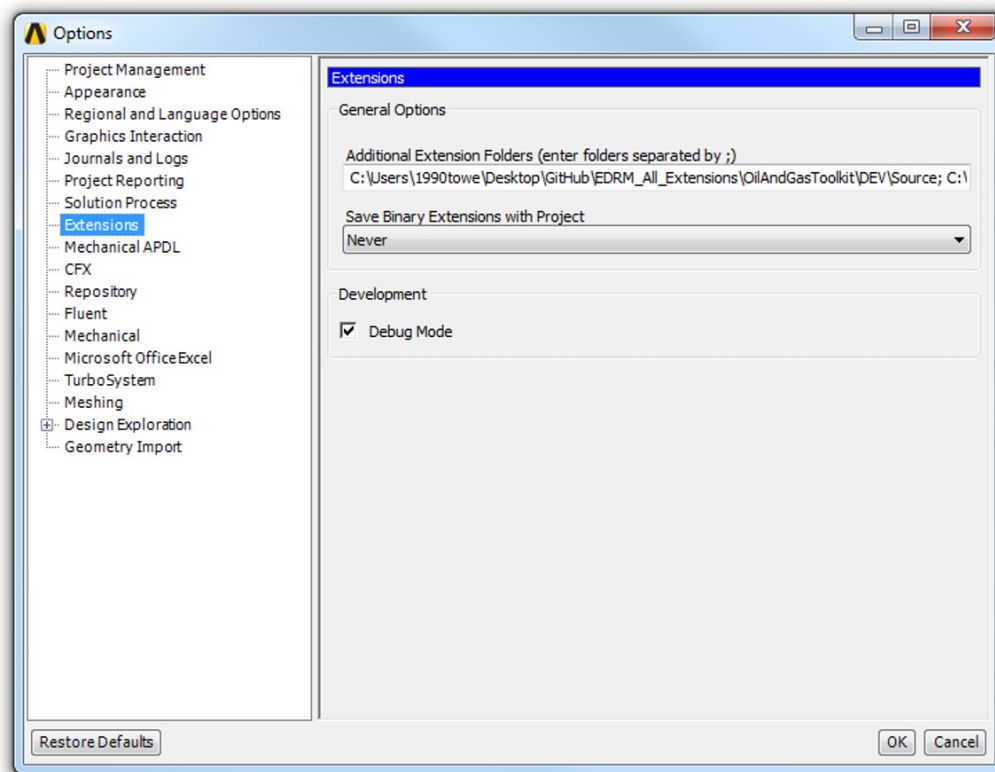


Figure 4: *Manually insert the address where the scripted version of an extension is stored*

Make sure *Debug Mode* is checked and that you have a Developer Mode license. After successfully inserting the location of the folder where the scripted version is stored, activate the scripted extension the same way as for binary extensions. Note that scripted extensions are noted as Type *Scripted* in the extension manager while binary extensions are noted as Type *Binary*.

Now that the extension is activated, open ANSYS Mechanical and the extension GUI is visible. Note the additional toolbar *ACT Development* with two click-able buttons *Update* and *View Extension Log File* from figure 5.

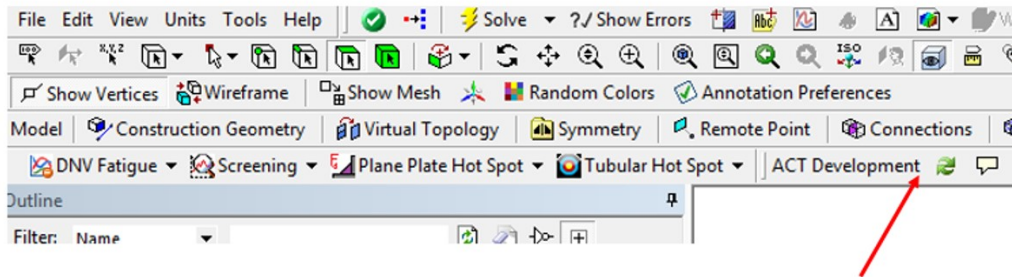


Figure 5: *ACT Development Extension. Update and view Extension Log File*

When source code to an active scripted extension is updated, clicking the *Update* button will incorporate these changes to apply the next time a result object from the scripted version is evaluated. The extension log file will include any error messages the source code generates, and makes it possible to print self-made messages from including information directly from the source code by the command `ExtAPI.Log.WriteMessage("write this to extension log file")`.

```
meshUnit = result.Analysis.MeshData.Unit
ExtAPI.Log.WriteMessage("ready for HC Screening " + str(meshUnit))
```

Figure 6: *Example of string written in Extension Log File when High Cycle Screening is evaluated. Output will be "ready for HC Screening m" when the mesh unit is meters*

Uninstalling the toolbar

To uninstall an extension, use *Extensions -> Extension Manager* select the desired extension, right mouse click and select *Uninstall*. Note that only binary extensions can be uninstalled.

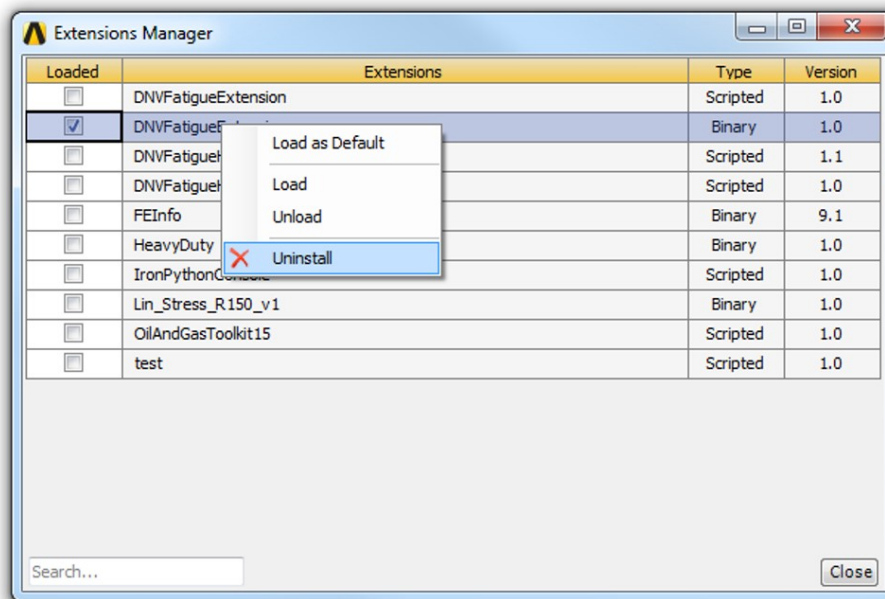


Figure 7: *Uninstall an extension in ANSYS Workbench*

Property Input Guidance

Table 1: Details View of the *High Cycle Screening* result object

Scope	
Geometry	Result geometry scope. Select vertices, edges, faces or bodies
Shell	Specify whether <i>Top</i> , <i>Bottom</i> or <i>Middle</i> result should be displayed for shell bodies
Definition	
Result View	Specify result type to display. <i>Stress Range</i> cannot be set for load case combination
Stress Option	Specify which stress type to use in evaluation
Load Case Combination	Specify to evaluate upon a single load case or a combination of load cases*
Start Step	Specify the start of a load case**
End Step	Specify the end of a load case**
Number of Cycles	Used to calculate damage. Positive integers are valid
Load Case Table	Used to define load cases. Same properties as above
By	Not used. Keep as default***
Display Time	Not used. Keep as default***
Integration Point Result	
Display Option	Specify how to display the result, <i>Averaged</i> or <i>Unaveraged</i> across elements
S-N Class	
S-N Data equal For All Bodies	Specify whether the same S-N Data should apply for all bodies or differ
Environment	Specify which environment the structure is exposed to
S-N Curve	Specify which S-N Curve to apply in fatigue calculations
Thickness Modification Factor	Thickness ratio, values >1.0 are not valid
S-N Table	Table for unique S-N Data for each body. Properties as above
Temperature Over 100 Celsius	For high temperatures, a temperature dependent S-N Curve will be used
Temperature	Insert temperature over 100 Celsius if previous property is set to <i>Yes</i>

Table 2: Details View of the *Low Cycle Screening* result object

Scope	
Geometry	Result geometry scope. Select vertices, edges, faces or bodies
Shell	Specify whether <i>Top</i> , <i>Bottom</i> or <i>Middle</i> result should be displayed for shell bodies
Definition	
Result View	Specify result type to display. <i>Stress Range</i> cannot be set for load case combination
Load Case Combination	Specify to evaluate upon a single load case or a combination of load cases*
Start Step	Specify the start of a load case**
End Step	Specify the end of a load case**
Number of Cycles	Used to calculate damage. Positive integers are valid
Load Case Table	Used to define load cases. Same properties as above
By	Not used. Keep as default***
Display Time	Not used. Keep as default***
Integration Point Result	
Display Option	Specify how to display the result, <i>Averaged</i> or <i>Unaveraged</i> across elements
ϵ-N Class	
ϵ -N Data equal For All Bodies	Specify whether the same ϵ -N Data should apply for all bodies or differ
Environment	Specify which environment the structure is exposed to
Temperature	Specify temperature, used to find or interpolate Young's Module
ϵ -N Table	Table for unique ϵ -N Data for each body. Properties as above
Advanced	
Newton Raphson Initial Guess	Initial guess as guidance for iteration method

Table 3: Details View of the *High Cycle Plane Plated Weld* result object

Scope	
Geometry	Result geometry scope. Select edges, faces or bodies
Shell	Specify whether <i>Top</i> , <i>Bottom</i> or <i>Middle</i> result should be displayed for shell bodies
Definition	
Result View	Specify result type to display. <i>Stress Range</i> cannot be set for load case combination
Stress Option	Specify which stress type to use in evaluation
Load Case Combination	Specify to evaluate upon a single load case or a combination of load cases*
Start Step	Specify the start of a load case**
End Step	Specify the end of a load case**
Number of Cycles	Used to calculate damage. Positive integers are valid
Load Case Table	Used to define load cases. Same properties as above
By	Not used. Keep as default***
Display Time	Not used. Keep as default***
S-N Class	
S-N Data equal For All Bodies	Specify whether the same S-N Data should apply for all bodies or differ
Environment	Specify which environment the structure is exposed to
S-N Curve	Specify which S-N Curve to apply in fatigue calculations
Plate Thickness	Used to find extrapolation points
Reference Thickness	Values giving $\frac{t}{t_{ref}} > 1.0$ are valid
S-N Table	Table for unique S-N Data for each body. Properties as above
Temperature Over 100 Celsius	For high temperatures, a temperature dependent S-N Curve will be used
Temperature	Insert temperature over 100 Celsius if previous property is set to <i>Yes</i>

Table 4: Details View of the *Low Cycle Plane Plated Weld* result object

Scope	
Geometry	Result geometry scope. Select edges, faces or bodies
Shell	Specify whether <i>Top</i> , <i>Bottom</i> or <i>Middle</i> result should be displayed for shell bodies
Definition	
Result View	Specify result type to display. <i>Stress Range</i> cannot be set for load case combination
Load Case Combination	Specify to evaluate upon a single load case or a combination of load cases*
Start Step	Specify the start of a load case**
End Step	Specify the end of a load case**
Number of Cycles	Used to calculate damage. Positive integers are valid
Load Case Table	Used to define load cases. Same properties as above
By	Not used. Keep as default***
Display Time	Not used. Keep as default***
ϵ-N Class	
ϵ -N Data equal For All Bodies	Specify whether the same ϵ -N Data should apply for all bodies or differ
Environment	Specify which environment the structure is exposed to
Plate Thickness	Used to find extrapolation points
Temperature	Specify temperature, used to find or interpolate Young's Module
ϵ -N Table	Table for unique ϵ -N Data for each body. Properties as above
Advanced	
Newton Raphson Initial Guess	Initial guess as guidance for iteration method

Table 5: Details View of the *High Cycle Tubular Joint* result object

Scope	
Geometry	Result geometry scope. Select mesh nodes
Shell	Specify whether <i>Top</i> , <i>Bottom</i> or <i>Middle</i> result should be displayed for shell bodies
Hot Spot Classification	
Is Geometry Selected	Geometry Scope must be selected****
Classification Table	See table 7
Definition	
Result View	Specify result type to display. <i>Stress Range</i> cannot be set for load case combination
Stress Option	Specify which stress type to use in evaluation
Load Case Combination	Specify to evaluate upon a single load case or a combination of load cases*
Start Step	Specify the start of a load case**
End Step	Specify the end of a load case**
Number of Cycles	Used to calculate damage. Positive integers are valid
Load Case Table	Used to define load cases. Same properties as above
By	Not used. Keep as default***
Display Time	Not used. Keep as default***
Integration Point Result	
Display Option	Specify how to display the result, <i>Averaged</i> or <i>Unaveraged</i> across elements
S-N Class	
S-N Data equal For All Bodies	Specify whether the same S-N Data should apply for all bodies or differ
Environment	Specify which environment the structure is exposed to
S-N Curve	Specify which S-N Curve to apply in fatigue calculations
Plate Thickness	Used to find extrapolation points
Reference Thickness	Values giving $\frac{t}{t_{ref}} > 1.0$ are valid
S-N Table	Table for unique S-N Data for each body. Properties as above
Temperature Over 100 Celsius	For high temperatures, a temperature dependent S-N Curve will be used
Temperature	Insert temperature over 100 Celsius if previous property is set to <i>Yes</i>

Table 6: Details View of the *Low Cycle Tubular Joint* result object

Scope	
Geometry	Result geometry scope. Select mesh nodes
Shell	Specify whether <i>Top</i> , <i>Bottom</i> or <i>Middle</i> result should be displayed for shell bodies
Hot Spot Classification	
Is Geometry Selected	Geometry Scope must be selected****
Classification Table	See table 7
Definition	
Result View	Specify result type to display. <i>Stress Range</i> cannot be set for load case combination
Load Case Combination	Specify to evaluate upon a single load case or a combination of load cases*
Start Step	Specify the start of a load case**
End Step	Specify the end of a load case**
Number of Cycles	Used to calculate damage. Positive integers are valid
Load Case Table	Used to define load cases. Same properties as above
By	Not used. Keep as default***
Display Time	Not used. Keep as default***
Integration Point Result	
Display Option	Specify how to display the result, <i>Averaged</i> or <i>Unaveraged</i> across elements
ϵ-N Class	
ϵ -N Data equal For All Bodies	Specify whether the same ϵ -N Data should apply for all bodies or differ
Environment	Specify which environment the structure is exposed to
Plate Thickness	Used to find extrapolation points
Temperature	Specify temperature, used to find or interpolate Young's Module
ϵ -N Table	Table for unique ϵ -N Data for each body. Properties as above
Advanced	
Newton Raphson Initial Guess	Initial guess as guidance for iteration method

Table 7: Hot Spot Classification Table****

Hot Spot Classification	Table to classify each hot spot given through geometry scope
Hot Spot Id Column in table	Read only cells to inform user about hot spot properties
Connected Edge Id Column in table	Read only cells to inform user about hot spot properties
Connected Face Id Column in table	Read only cells to inform user about hot spot properties
Connected Face Id Column in table	Read only cells to inform user about hot spot properties
Chord Id Column in table	Select which body represents the chord of the tubular joint
Brace Id Column in table	Select which body represents the brace of the tubular joint
Hot Spot Type Column in table	Select hot spot type crown or saddle point

**All solved analysis that share the model cell in Workbench are valid*

***Step 0 corresponds to step 0 and step Last corresponds to the last time step available. Note that the steps end time is used*

****Default property in a result details view, cannot be removed*

*****Geometry scope must be selected before using the Hot Spot Classification Table*

Class Diagrams

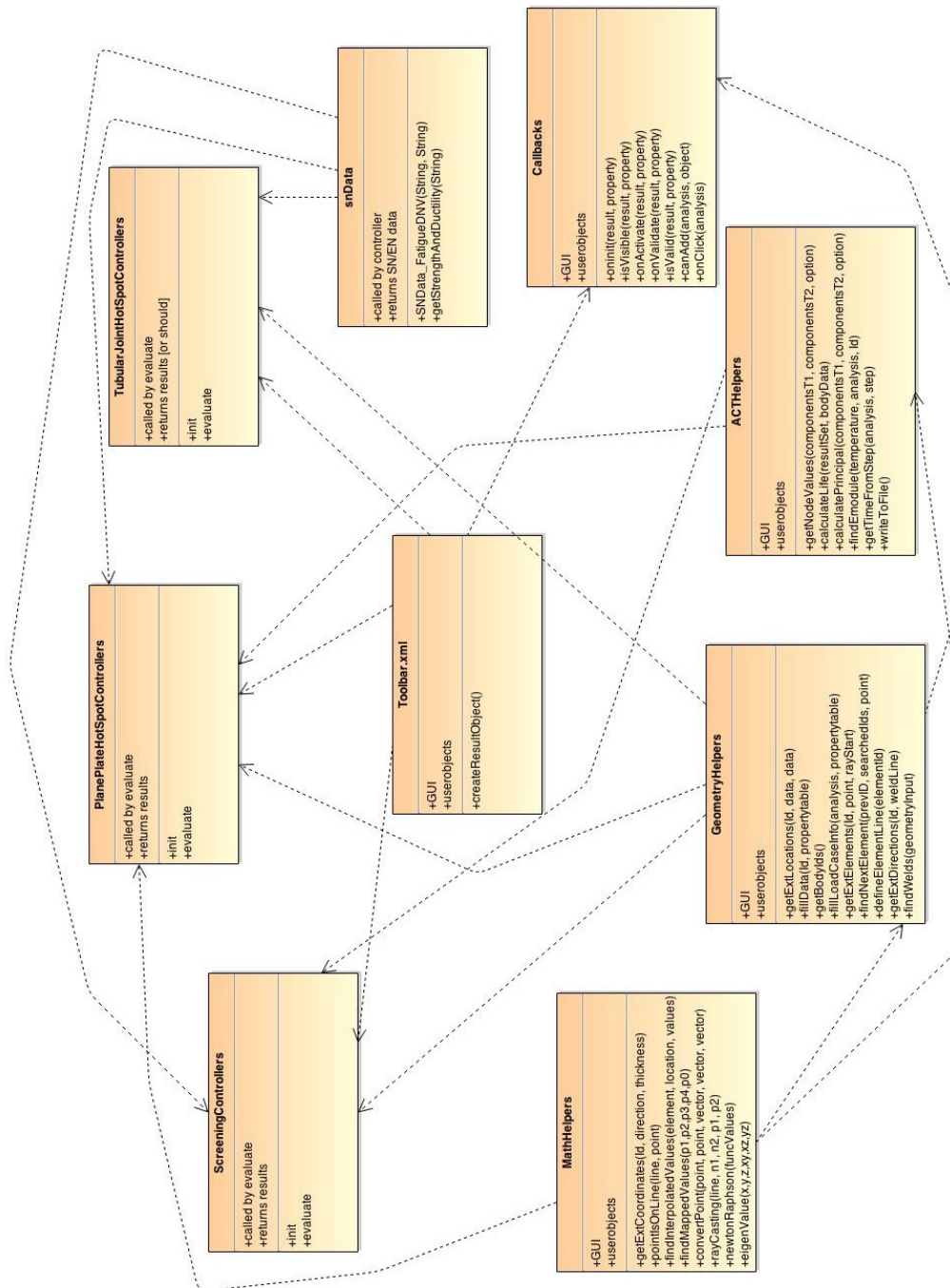


Figure 8: *System Overview*

Sequence Diagrams

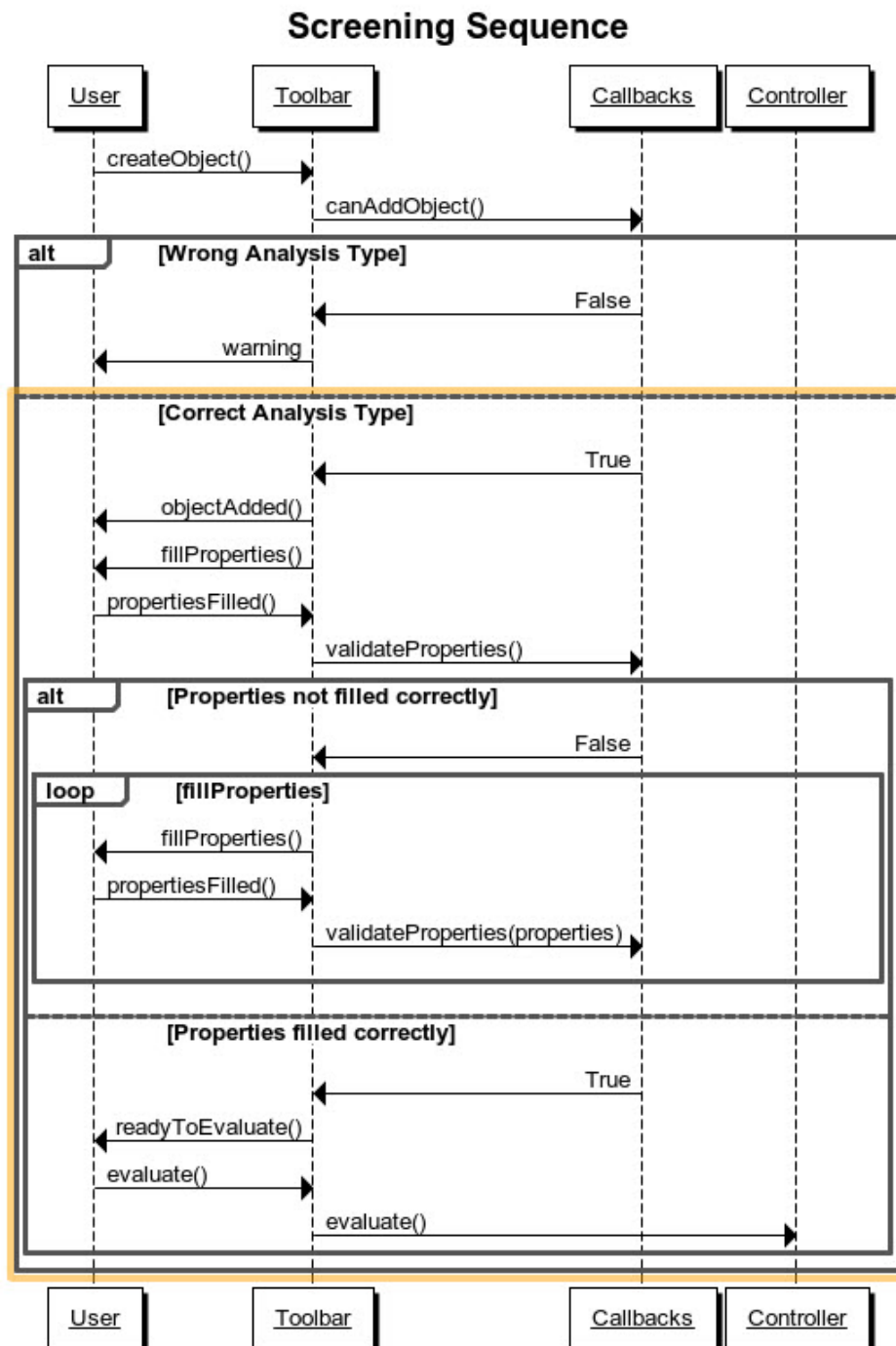


Figure 9: *Sequence diagram 1*

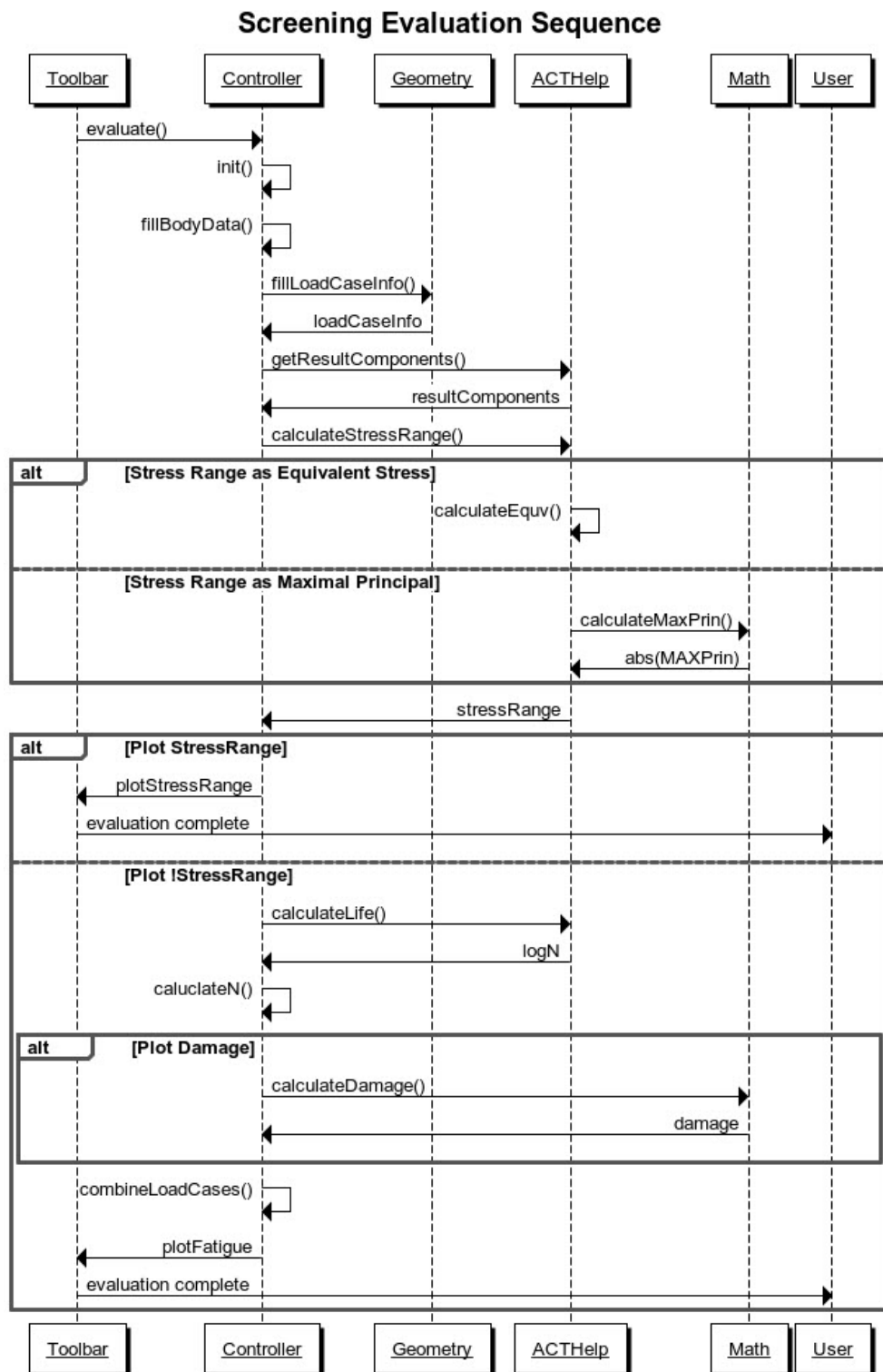


Figure 10: *Sequence diagram 2*

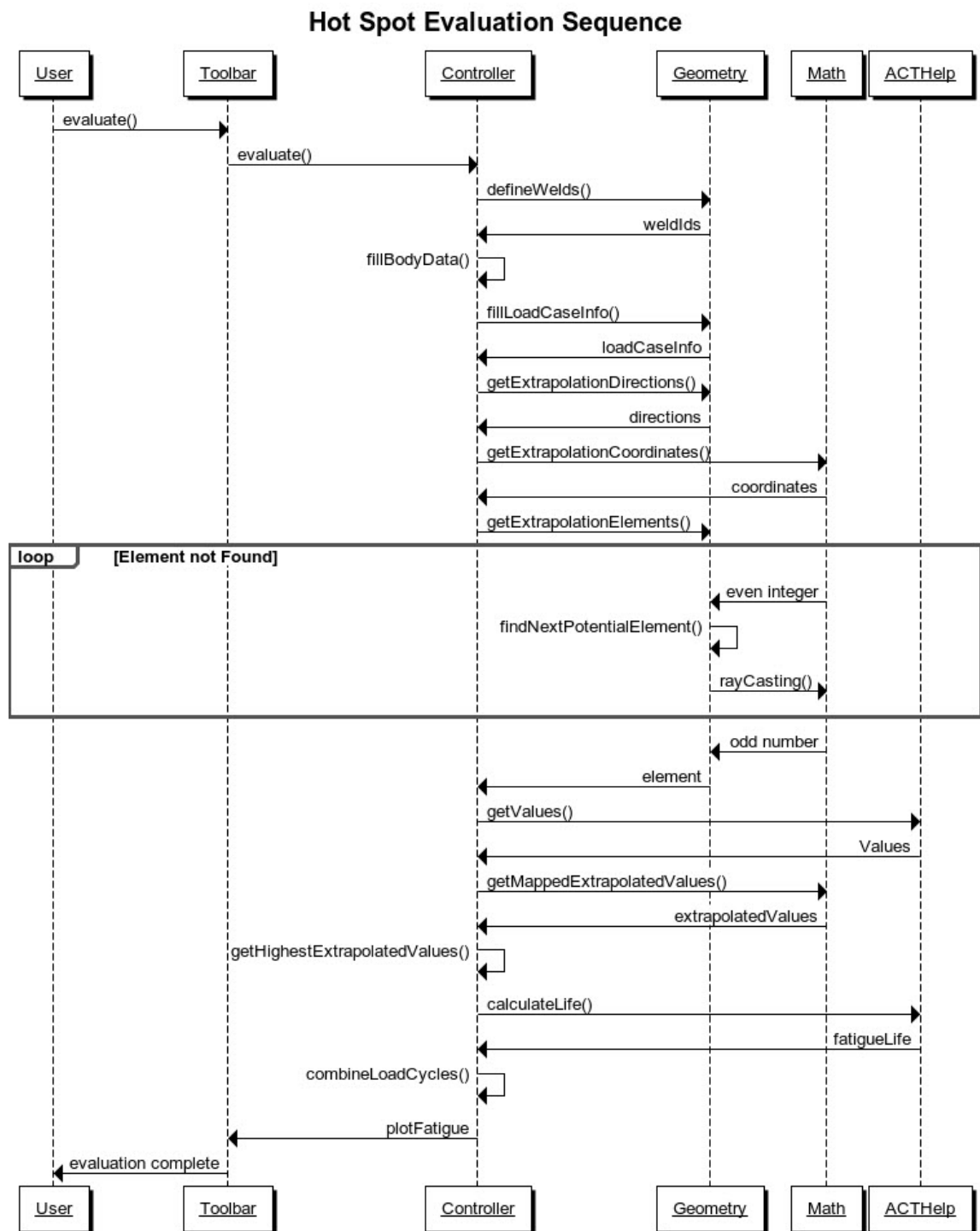


Figure 11: *Sequence diagram 3*

Software Specifications

Purpose

Function	Status	Explanation
Toolbar Buttons	Completed	Provide user with a simple and recognizable GUI in ANSYS Mechanical
Toolbar Input Sections	Completed	Provide user with a simple and recognizable GUI in ANSYS Mechanical for fatigue evaluation property input
Collecting Stress/Strain Ranges	Completed	Gather stress/strain ranges from ANSYS results to be used in fatigue evaluation
Calculate Fatigue High Cycle	Completed	Using stress ranges to calculate high cycle fatigue
Calculate Fatigue Low Cycle	Completed	Using strain ranges to calculate low cycle fatigue
Extrapolate for plane plated welds	Completed	Extrapolate values in hot spots and use these stress/strain ranges to calculate fatigue life
Extrapolate for simple, tubular joints	Incomplete	Extrapolate values in hot spots along chord and brace, and use these stress/strain ranges to calculate fatigue life
Combine Load Cases	Complete	Allowing user to evaluate fatigue on several load cases with only one result object
Displaying Fatigue Results	Complete	Plotting evaluated fatigue life and accumulated damage on input geometry in ANSYS Mechanical GUI

System Overview

Sub Process	User Interference	Task Performed	Limitations
Initiate result object	User button press	Result object added in solution outline	Only Static Structural and Transient Structural Analysis
Input property change	User change property in result object details view	Callbacks for validation and dependency	Limitations according to property restrains
Screening Evaluate	User select evaluate result object, type screening	fatigue is evaluated on geometry input and plotted on model	see Chapter 4
Plane Plated Weld Evaluate	User select evaluate result object, type plane plated weld	Welds and hot spots are defined from geometry and Fatigue is evaluated by extrapolation before plotted on model	see Chapter 4
Tubular Joint	User select evaluate result object, type tubular joint	Geometry input is classified and groundwork for future development is established	Incomplete application, result are not given back to user

Input

All input from user is given through details view. See tables 1, 2, 3, 4, 5 and 6 in Appendix *Property Input Guidance* for valid input and guidance.

Output

Output is given by fatigue life/accumulated damage or stress/strain ranges plotted on the model.

As the two *Tubular Joint* result objects are not completed, results will not be given.

Other output is error and warning messages given to user (see Chapter 4), and a error log file for when Newton-Raphson does not converge, which will be stored in the projects working directory if non-convergence should occur.

User Manual

The *DNV Fatigue Toolbar* extension is created with user friendliness in mind. The user available setup is designed to replicate default ANSYS result object setups, so that regular ANSYS users find the interface intuitive. To use the toolbar, follow the steps beneath:

1. For first time use: install the toolbar (see appendix install)
2. For first time use in a ANSYS project: activate the extension (see appendix install)
3. Open Mechanical
4. Select desired result object through categories *Screening*, *Plane Plated Weld* or *Tubular Joint* and insert in a static structural or transient structural analysis by either selecting the analysis solution before selecting the result object through the toolbar interface or by right-mouse-clicking the solution branch and insert directly
5. Select desired properties in the result objects details view
6. Solve result object by right-mouse-clicking evaluate or solve analysis
7. Watch for error or warning messages. Be aware that evaluation will not be complete if an error or warning message pop-up window is open
8. Review results on model

NTNU		Kartlegging av risikofylt aktivitet		Utarbeidet av	Nummer	Dato
				HMS-avd.	HMSRV2601	22.03.2011
HMS				Godkjent av		Erstatter
				Rektor		01.12.2006

Enhet: IPM **Dato: 13-08-2014**

Deltakere ved kartleggingen (m/ funksjon): Tone Wermundsen(student), Bjørn Haugen (veileder ved NTNU), Frøde Halvorsen (medveileder ved EDR Medeso)

(Ansv. veileder, student, evt. medveiledere, evt. andre m. kompetanse)

Kort beskrivelse av hovedaktivitet/hovedprosess: Masteroppgave student 714954. Efficient evaluation of fatigue and damage in welded joints from finite element analysis

Er oppgaven rent teoretisk? (JA/NEI): JA «JA» betyr at veileder innestår for at oppgaven ikke inneholder noen aktiviteter som krever risikovurdering. Dersom «JA»: Beskriv kort aktiviteteten i kartleggingskjemaet under. Risikovurdering trenger ikke å fylles ut.

Signaturer: Ansvarlig veileder: *Bjørn Haugen* Student: *Tone*

ID nr.	Aktivitet/prosess	Ansvarlig	Eksisterende dokumentasjon	Eksisterende sikringstiltak	Lov, forskrift o.l.	Kommentar
1	Kontoraktivitet		NTNUs HMS-Håndbok		Arbeidsmiljøloven	
1a	Skrivepultarbeid			Brannalarm, rømningsveier	Arbeidsmiljøloven HMS-instrukser	
1b	Dataarbeid			Backup		
2	Reise			Fortau, setebelte	Veitrafikkloven	