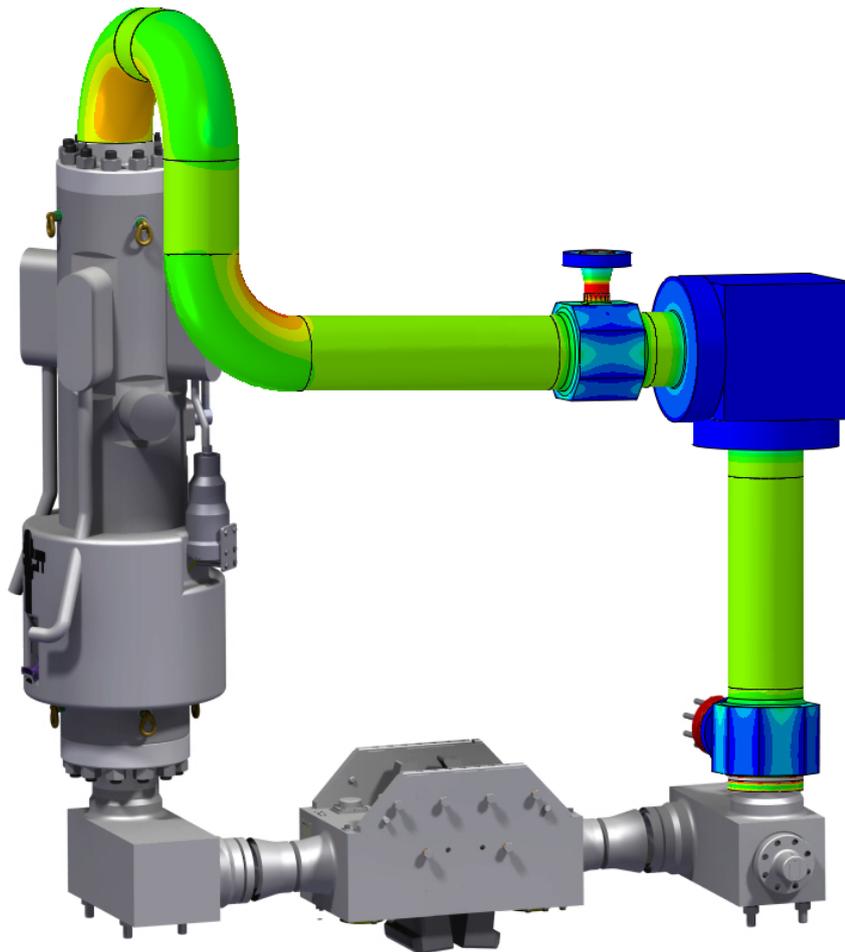


CHALMERS



Automated Flow Loop Simulations

Master's thesis in Applied Mechanics

DAVID ANDERSSON
MARCUS GUSTAFSSON

Department of Applied Mechanics
Division of Material and Computational Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2014
Master's thesis 2014:05

MASTER'S THESIS IN APPLIED MECHANICS

Automated Flow Loop Simulations

DAVID ANDERSSON
MARCUS GUSTAFSSON

Department of Applied Mechanics
Division of Material and Computational Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2014

Automated Flow Loop Simulations

DAVID ANDERSSON
MARCUS GUSTAFSSON

© DAVID ANDERSSON, MARCUS GUSTAFSSON, 2014

Master's thesis 2014:05
ISSN 1652-8557
Department of Applied Mechanics
Division of Material and Computational Mechanics
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: +46 (0)31-772 1000

Cover:
Flow Loop, ©2014 David Andersson, Marcus Gustafsson

Chalmers Reproservice
Göteborg, Sweden 2014

Automated Flow Loop Simulations

Master's thesis in Applied Mechanics

DAVID ANDERSSON

MARCUS GUSTAFSSON

Department of Applied Mechanics

Division of Material and Computational Mechanics

Chalmers University of Technology

ABSTRACT

In offshore subsea engineering, Flow Loops are the common name of pipings and fittings welded together to form a loop that carries production media from one point to another. Flow Loops are made of high strength stainless steels which are susceptible to hydrogen embrittlement. In combination with tensile loads, this may cause failure due to hydrogen induced stress cracking.

Finite element simulations are made when investigating the risk of this type of failure. The analyses must follow recommended practice, issued by Det Norske Veritas, to be approved. Problems with these analyses are that the modelling of Flow Loops is time consuming and rather tedious. Also, the recommended practice has been difficult to interpret with the effect that different engineers perform the analyses in various ways. This project aims to solve these problems by creating a tool that automates the procedure of creating Flow Loop finite element models and from the obtained results evaluates the Flow Loop's resistance against hydrogen induced stress cracking by following the recommended practice.

Two plug-ins for the finite element software Abaqus are developed to automate the simulation of Flow Loops. They are built by scripts written in the object oriented programming language Python. The first plug-in automates the process of creating the finite element model. Different types of parts may be created in this plug-in and these are put together in a model tree which defines the assembly of the Flow Loop. Loads and boundary conditions are also applied to the Flow Loop model using the plug-in. The second plug-in automatically analyses the results from the finite element model to evaluate the risk of hydrogen induced stress cracking, in accordance to the recommended practice. This plug-in automatically documents and visualises the results from the evaluation, showing which parts that are approved and not.

The project has been successful in automating Flow Loop simulations. The resulting plug-ins help the analysing engineer to rapidly build Flow Loop models and evaluate them. Even though there are some possible improvements to make in the verification according to recommended practice in the second plug-in, the automated analyses follow the recommended practice to an equal or greater extent compared to previous manual work.

Keywords: Flow Loop, subsea, oil and gas, offshore, Abaqus, Python, plug-in

PREFACE

This is the report of the Master's Thesis project *Automated Flow Loop Simulations*, performed during the spring semester of 2014. The aim of the project was to automate mechanical verification analyses of so called *Flow Loops*, components that are used in subsea applications for several reasons. The Flow Loops are to be verified with respect to the risk of hydrogen induced stress cracking. The project was performed at the company Aker Solutions AB's office in Gothenburg and is the final part of the five year long Mechanical Engineering program at Chalmers University of Technology.

ACKNOWLEDGEMENTS

We would like to thank all employees at Aker Solution's office in Gothenburg for their help and contribution to a great working environment. Special thanks to our supervisor Fredrik Lundgren, product owner Panagiota Gkallimani and specialist engineer Hongzhou Hu, for their support during the whole project. We would also like to thank our examiner, Professor Magnus Ekh.

Gothenburg, June 2014

David Andersson, Marcus Gustafsson

NOMENCLATURE

3D	Three dimensional
ASME	American Society of Mechanical Engineers
AWB	Annulus Wing Block
BC	Boundary Condition
CAE	Computer Aided Engineering
CPU	Central Processing Unit
DNV	Det Norske Veritas
FCM	Flow Control Module
FE	Finite Element
FEA	Finite Element Analysis
GUI	Graphical User Interface
HIP	Hot Isostatic Pressing
HISC	Hydrogen Induced Stress Cracking
ISO	International Organization for Standardization
LE	Logarithmic Total Strain
MDB	Model Database
MEG	Mono Ethylene Glycol
NPS	Nominal Pipe Size
ODB	Output Database
PWB	Production Wing Block
ROV	Remotely Operated Underwater Vehicle
RP	Recommended Practice
THE	Thermal Strain
WT	Wall Thickness
XMT	Subsea Christmas Tree

CONTENTS

Abstract	i
Preface	iii
Acknowledgements	iii
Nomenclature	v
Contents	vii
1 Introduction	1
1.1 Project background	2
1.2 Objectives	2
1.3 Boundaries	2
2 Flow Loops	3
2.1 Subsea structures	3
2.1.1 Manifold	3
2.1.2 Christmas Tree	3
2.1.3 XMT production Flow Loop	5
2.1.4 FCM production Flow Loop	5
2.1.5 XMT cross over loop	6
2.1.6 Water/gas injection tree Flow Loop	7
2.2 Flow Loop geometry	7
2.2.1 Pipes	7
2.2.2 Elbows	7
2.2.3 Tees	8
2.2.4 Flanges	8
2.2.5 Special blocks	8
2.2.6 Strength reducing factors	9
2.3 Flow Loop assembly	10
2.4 Flow Loop materials	11
2.4.1 Duplex stainless steels	11
3 Hydrogen Induced Stress Cracking in Flow Loops	12
3.1 Corrosion in general	12
3.2 Hydrogen embrittlement	12
3.3 HISC	13
3.4 Recommended practice for design	13
3.4.1 General	13
3.4.2 Design philosophy	13
3.4.3 Loads and conditions	13
3.4.4 Design criteria	14
3.4.5 Material requirements	15
4 Modelling techniques	16
4.1 Material models	16
4.1.1 Implementing material models in Abaqus	17
4.2 Tie constraints	17
4.3 Kinematic couplings	18
4.4 Finite elements	19
4.4.1 C3D8R	19
4.4.2 C3D6	20

5	Automation of Flow Loop HISC analysis	21
5.1	The Flow Loop plug-in	21
5.1.1	Initiation of Flow Loop model	21
5.1.2	Main window - geometry tab	22
5.1.3	Main window - load tab	28
5.1.4	Main window - BC tab	29
5.1.5	Load steps	31
5.1.6	Write Input	31
5.1.7	Output from the plug-in	32
5.2	The HISC Evaluation plug-in	34
5.2.1	Initiation of the HISC Evaluation plug-in	34
5.2.2	Part types included in the analysis	35
5.2.3	Evaluation algorithm	35
5.2.4	Results from the plug-in	37
6	Flow Loop Simulation	40
6.1	Example of a Flow Loop model	40
6.2	HISC analysis of example Flow Loop	41
6.3	Mesh Convergence	42
6.4	Principal strain directions	43
7	Concluding remarks and discussion	45
7.1	The Flow Loop plug-in	45
7.2	The HISC Evaluation plug-in	45
7.3	Future work	47
	References	48

1 Introduction

Today, subsea technologies and products are becoming more important for the oil and gas industries, making it possible to increase their production and access untapped resources. Put simply, the objective of a subsea operation is to bring oil and gas from a reservoir to the sea surface by establishing a piping system in between. Subsea fields enable the extraction of oil from more complex reservoirs located in deeper waters, down to several thousands of meters, compared to the ordinary topside platforms which are attached to the sea bed. A typical subsea oil or gas field with one well is shown in Figure 1.1 with corresponding components typically used.

The oil or gas well is shown to the lower left in Figure 1.1. On top of the wellhead, a so called *Christmas Tree* (XMT) is placed. The XMT contains production tubing and is connected to the *manifold*. The purpose of the manifold is to save on the amounts of *flow lines* needed, which carry the fluids along the sea bed to the *riser*. In many cases, a *riser base* acts as a transition point between a rigid piping system at the sea bed and the flexible riser. The riser is a flexible pipe that carries the production media from the sea bed to a topside facility. The different subsea structures are connected by *spools* and *well jumpers*, which are either rigid or flexible steel pipes. Not all components will be explained in detail, but the ones relevant to this project will be discussed in Chapter 2.

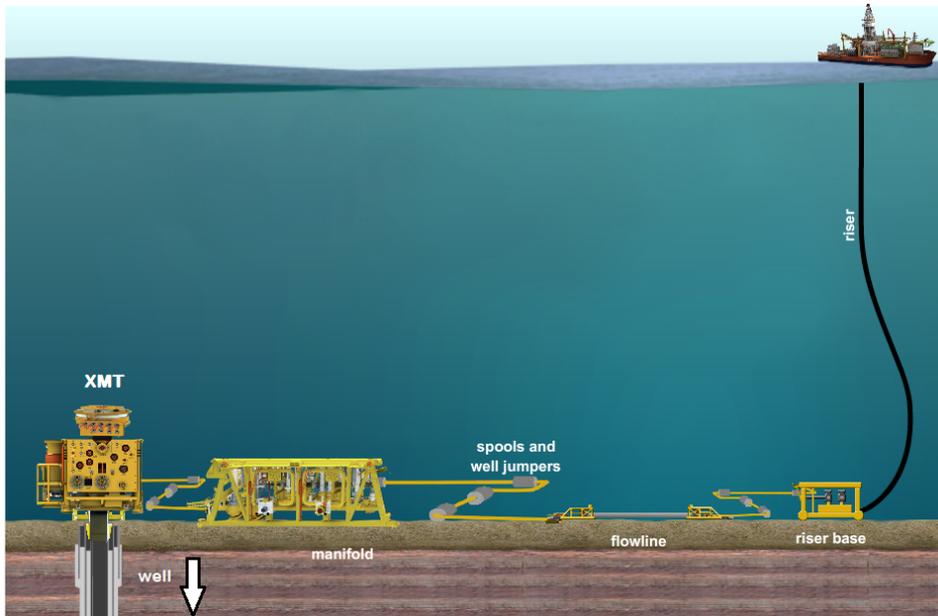


Figure 1.1: *Subsea production field with one well.*

The XMT is a safety barrier between the well and the surrounding environment and enables well interventions, measurements and control of the flow from the well. Important parts of XMTs are Flow Loops which is the common name for pressure containing components consisting of pipes, fittings and flanges. Several different such components exist in a XMT. The Flow Loops as well as the XMT are described in more detail in Chapter 2.

The dominating materials of Flow Loops are duplex stainless steels. Generally in corrosive environments, areas of ferrite and austenite phases in such steels act as anode and cathode, respectively, in electrochemical cells. In subsea applications, cathodic protection of the structures is often used through sacrificial anodes of other materials than steel. This will make both the ferrite and austenite cathode in the corrosion process. Due to the oxidation and reduction processes at the different half-cells, this may give ingress of hydrogen into the material and in turn hydrogen embrittlement.

Hydrogen embrittlement in interaction with tensile loads may cause cracking and in the end, failure of components. This phenomenon is what is referred to as *Hydrogen Induced Stress Cracking* (HISC). In addition to the general fatigue and maximum stress requirements that exist on structures, HISC must be considered when verifying design of subsea components.

1.1 Project background

Aker Solutions is a highly regarded company within the oil and gas industry, known for their capability of delivering custom made subsea solutions that are aligned with advanced, technically demanding client specifications. Since custom made solutions set high demands on their organisation, Aker Solutions continuously strive to improve the execution of projects. They have identified a possibility to improve the verification of Flow Loop design with respect to HISC.

The verification is done by means of Finite Element (FE) analysis and the analyses are reoccurring and time consuming. The problem is that in general, the Flow Loops are different from each other due to the different customer specific solutions. Small differences in Flow Loop design demand new FE models, each with unique pre-processing procedure, for every new XMT. A great amount of time is spent on each analysis even though they fundamentally are very similar. The opportunity of improvement is therefore to automate these analyses to decrease lead times and tedious modelling.

The HISC analyses must follow a so called *Recommended Practice* (RP), DNV-RP-F112, issued by *Det Norske Veritas* (DNV), see [1]. This RP provides guidelines and requirements to design geometries resistant to HISC in their application. Another observed problem is the varying interpretations of the RP among different engineers. The HISC analyses do not always follow a standard procedure in terms of geometry simplifications, loads taken into account, analysis settings, etc., which might cause quality issues in the analyses.

1.2 Objectives

The aim of the project follows directly from the project background above and it is to automate the Flow Loop simulations. Based on input from drawings, a correct 3D Flow Loop geometry ready for FE analysis should be generated. Boundary conditions and loads should be applied to the Flow Loop geometry, stresses and strains calculated and checked against the HISC standard, all as automatically as possible.

Achieving this, the lead times can be lowered significantly. Also, it will be ensured that materials, loads, boundary conditions, output requests and other simulation specific input will be defined correctly and in an identical manner for all projects. All significant input and basis for analysis should be transparent to the engineer running the analysis. Documentation that can be archived together with a simulation report should therefore be produced.

To summarise, the main objective of the project is:

To create a tool that as automatically as possible determines whether a given Flow Loop fulfills the requirements of the recommended practice DNV-RP-F112. The analyses should be well documented.

1.3 Boundaries

Boundaries for the project were formulated to decrease the risk of misunderstandings and uncertainties during the work with the project. The following boundaries were identified:

- The simulations will focus only on Flow Loops and their components.
- The HISC RP will be the design requirement considered in the project.
- The FE software that will be used is Abaqus CAE. The possibility to combine Abaqus with Python scripting will be utilized for the automation process. No other softwares will be considered.
- Loads and boundary conditions applied in the models will follow what has been practice at Aker Solutions in earlier HISC evaluations, these loads will however be evaluated to find their physical relevance.
- No design changes will be proposed, the aim is only to be able to verify existing designs.
- No investigation regarding change of material will be executed.

2 Flow Loops

In Chapter 1, the subsea production system was introduced and roughly explained. In the following sections, some of the components will be explained in more detail. The main focus will be on *Flow Loops* which are the components considered in this project.

2.1 Subsea structures

A subsea production field may have one or more *wells* where oil and/or gas is extracted. On top of each well, a subsea tree also known as a *Christmas Tree* (XMT) is placed. From the wells and the XMTs, the production media run through piping systems to the *manifold* where the flows from different wells are joined for the *flow line* and the *riser* that carry the production media to the surface.

2.1.1 Manifold

A manifold is a system of branched piping that is used as a base, gathering the production media from several XMTs located on different wells in the field. The manifold provides the possibility to control and distribute the production flow from multiple wells by turning it on and off or just choking the flow. A typical location of a manifold in a subsea field is presented in Figure 1.1. The system of piping is often protected by a template that also acts as structural framework and foundation. These templates are usually very large, Figure 2.1 shows transportation of one which has a weight of 1,800 tonnes.



Figure 2.1: *Manifold template used as gas compression base in the Åsgard field in Norway. The steel frame has a weight of 1,800 tonnes.*

2.1.2 Christmas Tree

A typical XMT is shown in Figure 2.2 where the main parts are pointed out. Its name comes from the assembly of *valves*, *spools* and fittings which in a way resembles an ordinary Christmas tree. A XMT is always placed on top of an oil and gas well, mainly to act as a safety barrier. Other important functions are to allow for different measurements and regulations of the flow by choking it through different valves. A XMT is thus not always used for extracting oil and gas, but sometimes to pump sea water or other media into wells to control the pressure of the production field or drilling further down. The latter functions are called well interventions.

The XMT in Figure 2.2 is a so called horizontal XMT where the production media are lead from the well, through the valves and then out horizontally. The spool is only used for well interventions and is closed with a specially designed lid called Tree Cap during normal use. The *Production-* and *Annulus Wing Block* (PWB and AWB) are used to inject fluids into the production media during usage, one example could be Mono Ethylene

Glycol (MEG) which acts as an inhibitor against gas hydrates. Gas hydrates are crystalline formed structures made of water and gas molecules. They may develop above freezing temperature at certain pressure conditions and form an ice-like plug, clogging the pipes, cf. [2].

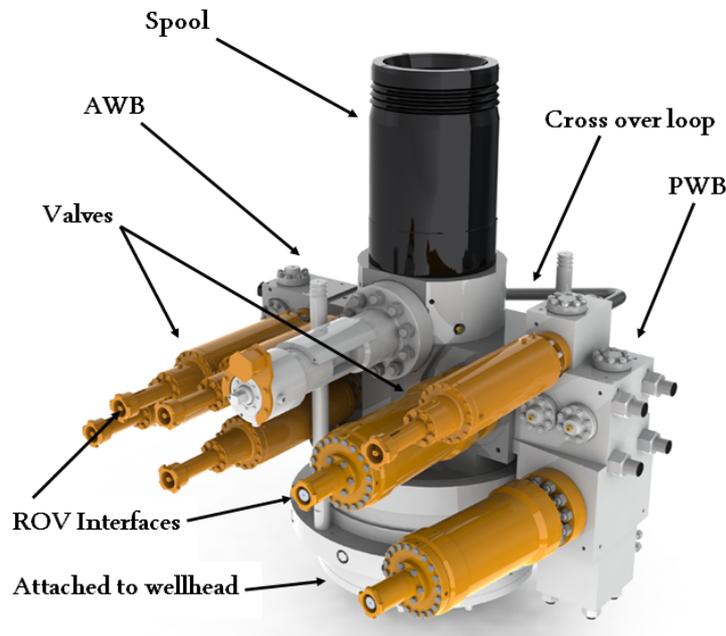


Figure 2.2: *Christmas Tree*.

In subsea environments, the XMT components are mounted in a main steel structure, see Figure 2.3. The structure unites the different parts of the XMT and thus connects the piping and valves to other components such as *sacrificial anodes* used for cathodic protection. The production flow is manually controlled by adjusting the valves with Underwater Robots (ROVs). These are equipped with different tools to be able to adjust the flow via the *ROV interfaces* at the end of the valves, shown in both Figure 2.2 and Figure 2.3. The XMT is shown standing on the seabed in Figure 1.1. A typical tree measures 4×4×4 meters and has a total mass of around 50 tonnes. There exist several different types and configurations of trees, and the requirements are increasing with today's demanding market.

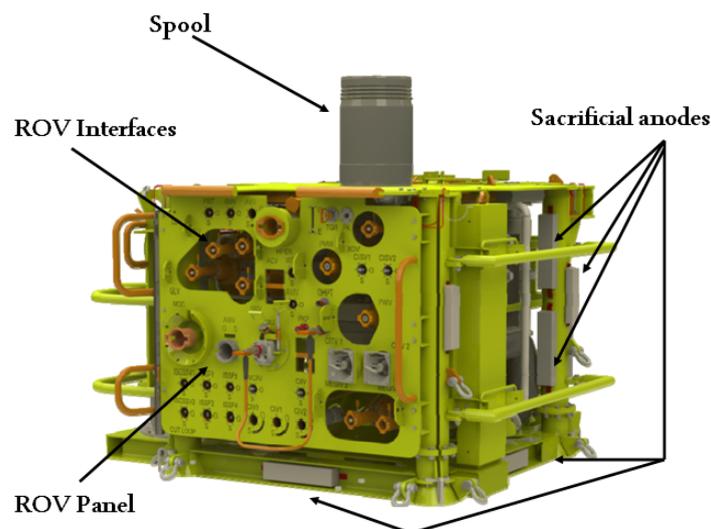


Figure 2.3: *Christmas Tree with surrounding structure*.

Important parts of the XMTs are Flow Loops. In the subsea engineering area, Flow Loop is the common name for pressure containing components consisting of pipes, fittings and flanges with an inner diameter exceeding 1" (25.4 mm) which are welded together. There are four different kinds of Flow Loops within a XMT, named depending on their purpose and corresponding requirements:

1. XMT production Flow Loop
2. FCM production Flow Loop
3. XMT cross over loop
4. Water/gas injection tree Flow Loop

2.1.3 XMT production Flow Loop

The main flow from the well is lead through the XMT to the PWB, see Figure 2.2. The *XMT production Flow Loop* connects the PWB to a so called tie-in hub, connected to pipes carrying the production media to the manifold. This Flow Loop goes via a hub which connects to the *Flow Control Module* (FCM), see Figure 2.4.

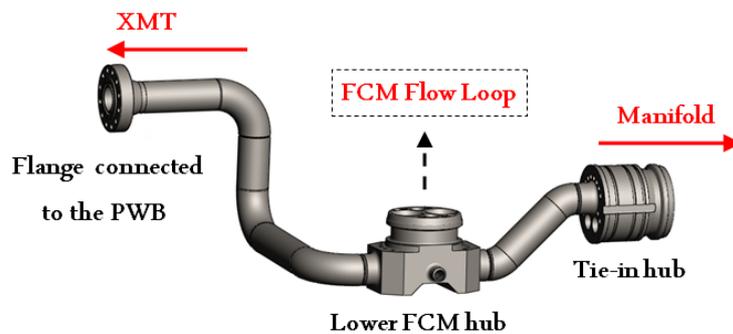


Figure 2.4: *XMT production Flow Loop with the lower hub of the FCM.*

2.1.4 FCM production Flow Loop

The FCM is a module that is installed either on a XMT or a manifold, an example is shown in Figure 2.5. This module enables control of the production media by allowing for different measurements and choking of the main flow.

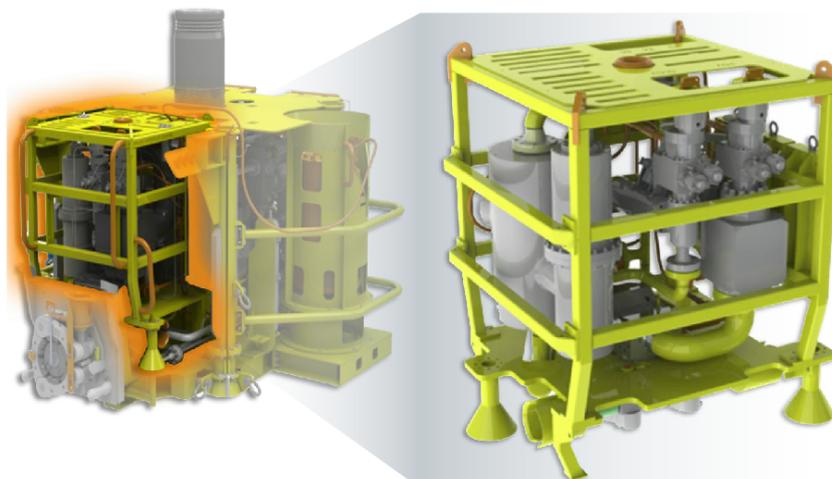


Figure 2.5: *To the left, the FCM is attached to the XMT. To the right, an enlargement of the FCM is shown.*

The piping in the FCM is connected to the upper FCM hub which is attached to the XMT production Flow Loop, see Figure 2.4, via the lower FCM hub. The piping parts connected to the upper FCM hub create a loop, referred to as *FCM production Flow Loop*. The purpose of this is to loop some of the production media from the main pipe past control systems (sensors) and then back to the main flow to enable different measurements. Common measures of interests are temperature, pressure as well as amount of sand and other particles. The latter is measured with a so called Multi Phase Flow Metre.

This means that the actual Flow Loop in the FCM does not consist solely of pipes, but can also include sensors and blocks with different functions and material properties. In Figure 2.6a, an FCM assembly is presented. Also shown is a view of the actual FCM production Flow Loop geometry, Figure 2.6b, and a simplified model of it in Figure 2.6c.

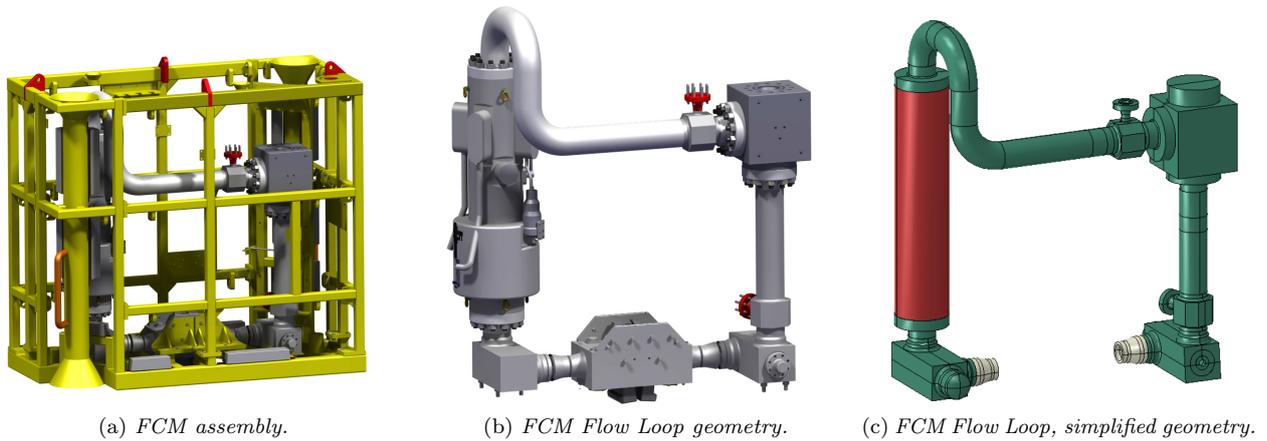


Figure 2.6: *FCM Flow Loop, three different levels.*

2.1.5 XMT cross over loop

Another example of a Flow Loop is the so called cross over loop, shown in Figure 2.2. It consists simply of piping connecting the AWB to the PWB, acting as a safety device used in case of pressure overloads. Sometimes this Flow Loop is cross connected to a third block called the MEG block. Figure 2.7 shows the backside of a XMT and the *XMT cross over loop*.

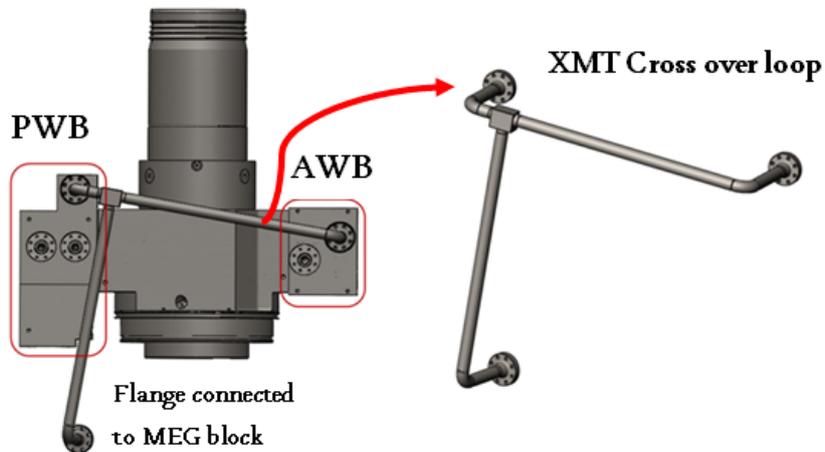


Figure 2.7: *A typical XMT cross over loop, also connected to an MEG block.*

2.1.6 Water/gas injection tree Flow Loop

The last Flow Loop is very similar to the XMT production Flow Loop. The difference is the field of application, a water/gas injection tree is used to inject either water or gas into the well via the production bore, to stabilize the pressure in the wells. The sea water or gas is provided to the XMT from the manifold and the *water/gas injection tree Flow Loop* is simply the Flow Loop connecting the XMT to the tie-in hub for the piping from the manifold. The appearance is just as in Figure 2.4 but without the lower FCM hub and with reversed flow direction.

2.2 Flow Loop geometry

As already mentioned, Flow Loop is the common name for pressure containing components consisting of pipes, fittings and flanges that are welded together, all with an inner diameter exceeding 1" (25.4 mm). The components are divided into two different categories, *standard parts* and *special blocks*. Standard parts are pipes, elbows, tees and flanges. These are manufactured according to different standards where their possible dimensions are tabulated. Special block is in this report used as an generic term for all parts that are not standard. Note, however, that special blocks may be parts fulfilling the same functions as standard parts but they are uniquely designed to withstand higher loads.

2.2.1 Pipes

Pipes are manufactured according to ASME B36.10M-2004, *Welded and Seamless Wrought Steel Pipe*, see [3]. In this standard the size is identified by the Nominal Pipe Size (NPS) measured in inches. Every size available is tabulated in the standard and the input needed to fully define a pipe is:

$$\begin{aligned} D &= \text{outer diameter} \\ t &= \text{nominal thickness} \\ L &= \text{length} \end{aligned}$$

Seamless pipes are manufactured by heating cylindrical steel billets and then pushing them over a piercing device while the billets are rotating. This method is called cross-rolling piercing and is followed by two other rolling operations to get the right wall thickness and by elongation get the wanted outer diameter. After this the pipes are cooled down and then heat treated one last time before quality control and pressure testing is performed on each pipe, cf. [4].

2.2.2 Elbows

Elbows are manufactured according to ASME B16.9-2007, *Factory-Made Wrought Butt welding Fittings*, see [5]. A standard elbow is, just as pipes, defined by the NPS, and this makes their opening interfaces equivalent. Hence, the outer diameter, D , and thickness, t , are taken from the pipe standard. The remaining measures to define a standard elbow are the *center-to-end* distance, B , and angle, θ , which are defined in Figure 2.8.

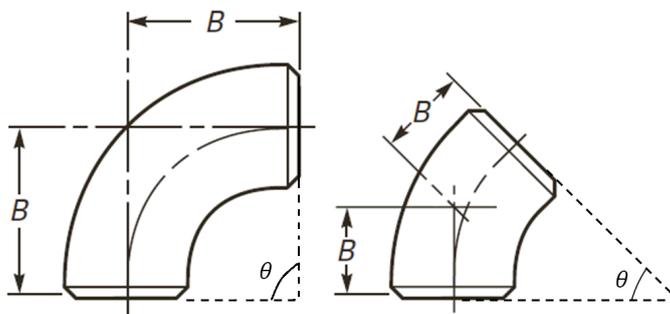


Figure 2.8: Drawing of standard elbows, to the left $\theta = 90^\circ$ and to the right $\theta = 45^\circ$.

To summarise, the input parameters defining a standard elbow are:

- D = outer diameter
- t = nominal thickness
- B = center-to-end distance
- θ = angle

Standard elbows are commonly manufactured by either hot or cold forming. Hot forming starts with a pipe having smaller dimensions than the final elbow. The pipe is heated up and then pushed over a bent mandrel forcing it to expand and take the desired shape. In the case of cold forming, a pipe of standard dimensions is pushed through a die to shape an elbow, cf. [6].

2.2.3 Tees

Tees divide the main flow into two branches and the standard dimensions are covered by the same standard as the elbows, [5]. Tees are also defined by the NPS and two center-to-end measures corresponding to each NPS. These measures are called *run*, C , and *outlet*, M , see Figure 2.9.

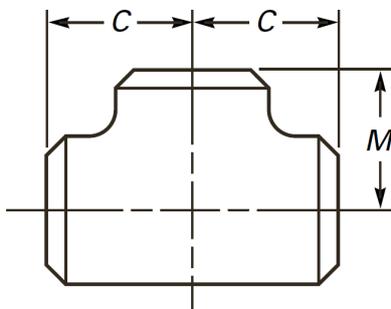


Figure 2.9: Drawing of a standard tee showing the measures M and C .

A common way of manufacturing standard tees is by placing a pipe in a die as in Figure 2.10. By applying high hydraulic pressure, the third branch will be forced out through the opening at the top. The upper end is then cut off and machined to its final design, cf. [6].

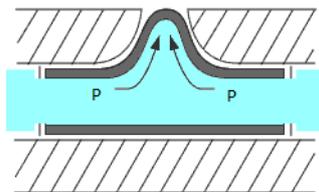


Figure 2.10: Manufacturing of a tee by subjecting a pipe to a hydraulic pressure in a special die.

2.2.4 Flanges

Flanges are often placed at the end of a Flow Loop enabling it to be fixed with bolts. This makes it easy to install, repair and replace different sub-components. Flanges are designed according to the standard ISO 10423, see [7]. An interpretation of the standard is that 11 dimension parameters are required to fully define the design of a flange. A typical design is presented in Figure 2.11a next to a drawing showing these parameters, Figure 2.11b. Flanges are usually manufactured by forging followed by machining to obtain the final shape.

2.2.5 Special blocks

Special block is the generic name for parts not included in any standard. They may have the same functions and contact interfaces as the standard parts introduced above but designed with a much thicker wall thickness,

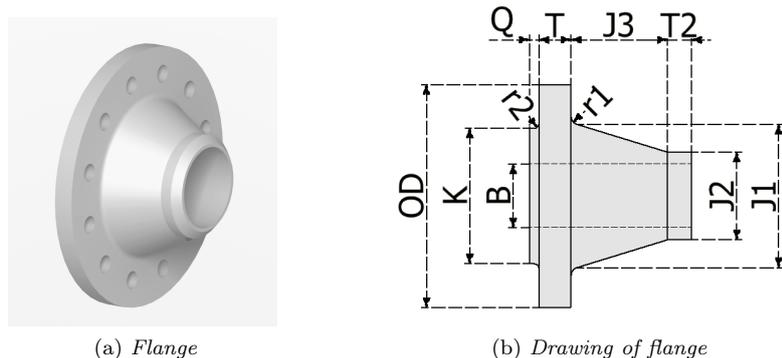


Figure 2.11: A flange with corresponding drawing according to ISO-10423.

often leading to a block shape. Various types of sensors used for monitoring the flow are also added to a Flow Loop as special blocks, Figure 2.12 shows some examples from a typical FCM Flow Loop.

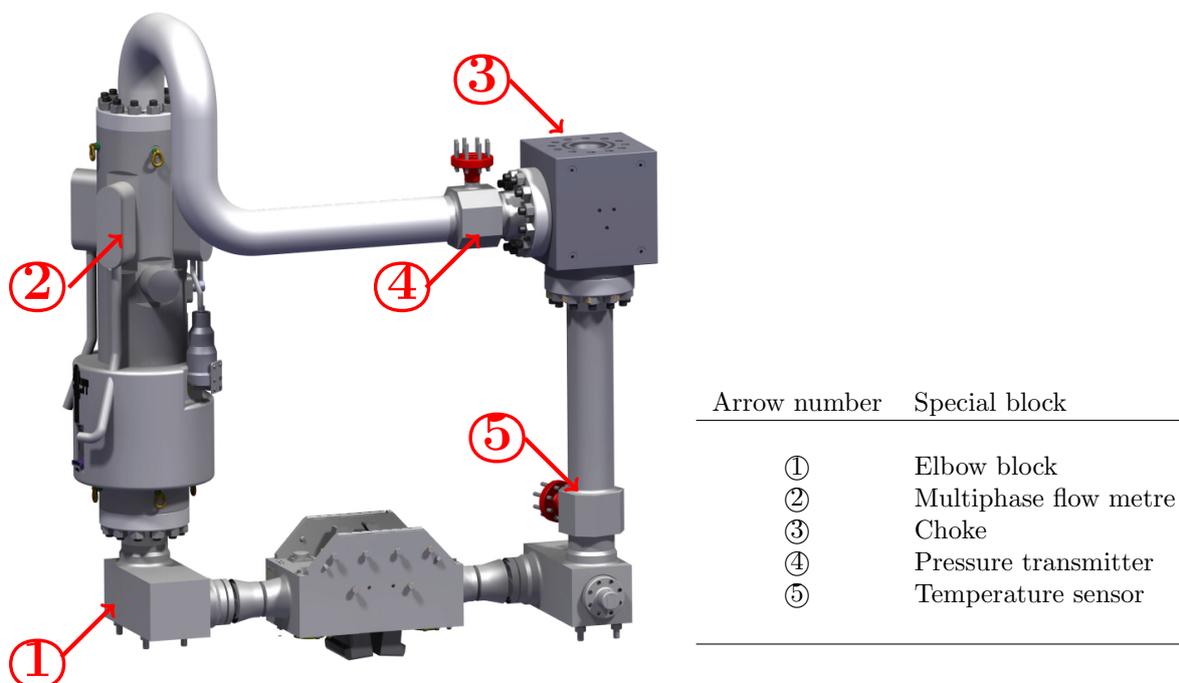


Figure 2.12: Example of special blocks mounted on a typical FCM Flow Loop.

The motivation for using special blocks instead of standard parts is that they can withstand higher loads. The goal is always to use standard parts since they are cheaper with regards to manufacturing and material cost. Standard parts are also less bulky and the amount of space is limited in a XMT. Standard parts must however be replaced by special blocks in case analyses reveal risk of failure. Special blocks are manufactured by forging a block with the right shape and then perform the machining operations needed. A special block, that acts as a tee in a Flow Loop, with a cut view is shown in Figure 2.13.

2.2.6 Strength reducing factors

The production medium flowing through a Flow Loop consists of a mix of oil, gas, sand, sea water and other substances causing a rough internal environment. The Flow Loops are also surrounded by sea water. This will decrease the dimensions of components during usage and thus decrease the lifetime of the construction. The environmental factors are categorised into *erosion* and *corrosion*. They must be taken into consideration when analysing Flow Loops since the aim is to prove that it will last throughout its entire service life.

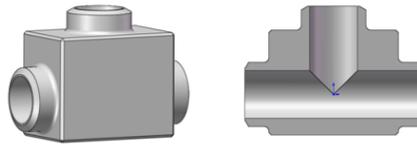


Figure 2.13: A tee special block with a cut view at the centerline.

Erosion

Due to sand and other hard particles, Flow Loops are grinded internally during production. This erosion is considered only to reduce the wall thickness from the inside. The common way of defining an erosion tolerance is by specifying the maximum wall thickness reduction at the end of service life in millimeters.

Corrosion

Corrosion is seldom taken into consideration when analysing Flow Loops since they are made of highly corrosion resistant materials, also helped by cathodic protection. However, when corrosion becomes important it may appear on the inner as well as the outer surface of a Flow Loop. It may also be more severe on one of these two than the other. This is due to the fact that the inside of the Flow Loop is subjected to a different environment than the outside. Inner and outer corrosion estimations at the end of service life are also specified by a maximum reduction of the wall thickness measured in millimeters.

Mill tolerance

Another important factor that must be considered when analysing Flow Loops is that the wall thickness of the separate parts is not always precise. Variations are accounted for by specifying a *mill tolerance* which defines how much the thickness is allowed to vary on the inside. A mill tolerance of 12.5 % is commonly used as a worst case scenario when analysing Flow Loops.

2.3 Flow Loop assembly

After manufacturing of separate parts, the ends are machined to create a welding bevel. During assembly of the Flow Loop the components are fixed together as in Figure 2.14a and the gap is filled with weld material. This type of weld is called girth weld, sometimes also referred to as circumferential weld, cf. [8]. In Figure 2.14b a cross sectional view of the weld is presented. Note that the cross section of the girth weld is similar to a typical butt weld.

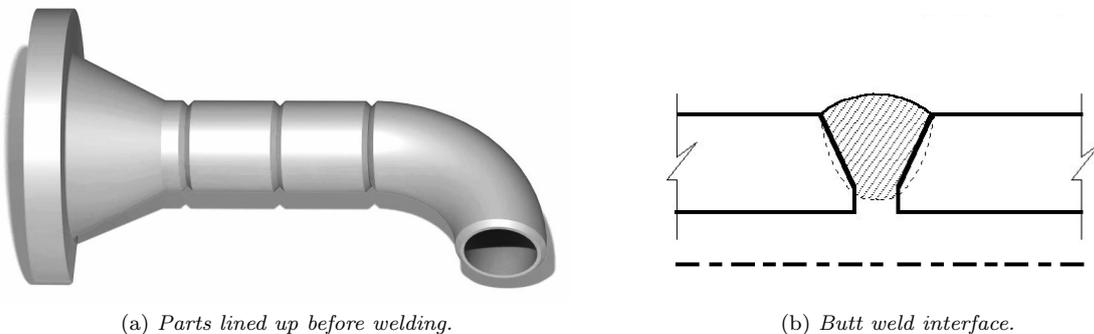


Figure 2.14: Flow loop before parts are welded together and a schematic view of the weld interface.

2.4 Flow Loop materials

Duplex stainless steels are extensively used in subsea equipment. Particularly, the parts that build a Flow Loop are practically always made from the ones listed in table 2.1.

Table 2.1: *Flow Loop materials.*

Material	Yield Strength	Tensile Strength
22% Cr Duplex Stainless Steel	450 MPa	620 MPa
25% Cr Super Duplex Stainless Steel	550 MPa	750 MPa

2.4.1 Duplex stainless steels

Stainless steels exist in different classes based on the phase composition of their microstructure. There are single-phase stainless steels which either consist of ferrite, austenite or martensite. In addition, there are so called duplex stainless steels, described e.g. in [9], that are composed by either both ferrite and austenite or both austenite and martensite. Here, only ferritic-austenitic duplex stainless steels are considered. Without going into development of microstructure and manufacturing, the main reason for using duplex stainless steels is the combination of good corrosion resistance and superior mechanical properties.

Austenitic stainless steels are the most widely used and have good corrosion properties but are sensitive to stress corrosion cracking. The corrosion properties of duplex stainless steels may be equal or even better than those for austenitic ones, particularly when it comes to stress corrosion cracking. At the same time, the yield and the tensile strength are generally higher than both ferritic and austenitic stainless steels. The weldability as well as the formability is as good or better than in other stainless steels. Altogether, these properties motivate the usage of duplex stainless steels in subsea applications. However, failures from hydrogen induced stress cracking (HISC) have occurred.

Duplex stainless steels and HISC

Duplex stainless steels are susceptible to hydrogen embrittlement which means that when there is a hydrogen source and a tensile stress, there is also a risk for HISC. According to Chai [10] the occurrence of HISC is a result of interaction between the hydrogen diffusion and a creep process due to the tensile stress.

In ferritic-austenitic duplex stainless steels, HISC propagation occurs mainly in the ferritic phase while the austenite either stops or slows the crack propagation down depending on the circumstances. Because of this, the austenite spacing or the free ferrite path is important for the HISC properties of the material. The finer the austenite spacing is, the greater resistance the material have to HISC, cf. [9] and [10].

Manufacturing techniques that give materials and components with small austenite spacing are favourable when it comes to HISC properties. Such techniques comprise Hot Isostatic Pressing (HIP), extrusion, rolling and drawing, cf. [1]. In the subsea application, components may be treated with HIP to enhance mechanical properties and seamless rolling is used to produce pipes.

3 Hydrogen Induced Stress Cracking in Flow Loops

As mentioned, Flow Loops are usually made out of duplex stainless steels and are together with the rest of the XMT subjected to cathodic protection by sacrificial anodes. The cathodic protection may generate interstitial hydrogen atoms in the duplex stainless steel which, in turn, may lead to HISC if tensile loads are present.

3.1 Corrosion in general

Corrosion of metals is, as described in [11], an electrochemical process that consists of two half-cell reactions. One is an oxidation reaction that results in degradation of the metal and one is reduction of hydrogen ions in two steps shown in Reactions (3.1) and (3.2). In a corrosive environment, different parts of the material's microstructure will achieve different potentials and thus either become anode or cathode in the electrochemical cell. The oxidation takes place at anode surfaces and the reduction at cathode surfaces.



In the first reduction step, Reaction (3.1), hydrogen is adsorbed at the cathode surface. The adsorbed hydrogen atoms, H_{ads} , stay at the metal surface until they find other H_{ads} to form hydrogen molecules, H_2 , which then leave the surface. This is step two, shown in Reaction (3.2).

3.2 Hydrogen embrittlement

A portion of the adsorbed hydrogen atoms may for certain metals and certain conditions diffuse in through the surface and into the material at cathode areas. Steels are sensitive to this phenomenon, particularly high-strength ones. The interstitial hydrogen atoms may cause greatly reduced ductility and tensile strength of the material. Figure 3.1 shows the reactions 3.1 and 3.2 and a diffused hydrogen atom.

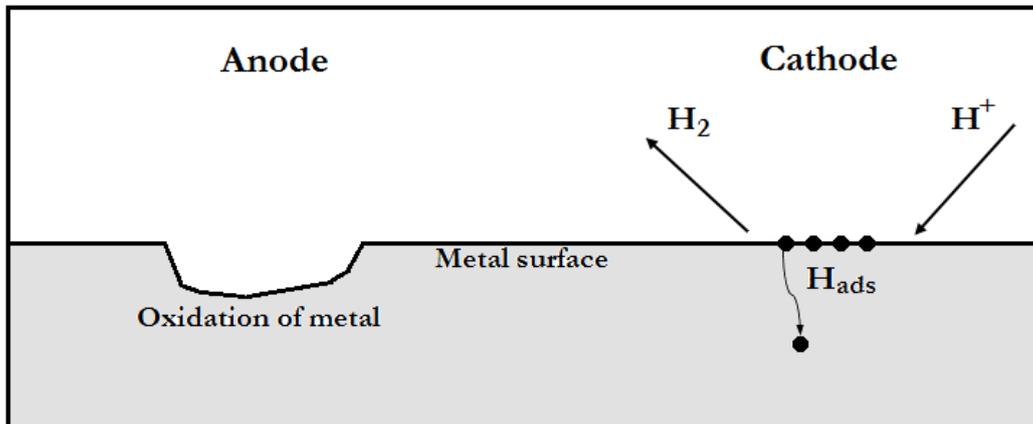


Figure 3.1: *The two half cells in the corrosion process. Below the cathode surface, an interstitial hydrogen atom is shown.*

Hydrogen embrittlement is not entirely understood, but different mechanisms have been proposed. One is that the hydrogen atoms within the microstructure interfere with dislocation motion, cf. [12]. Another is that the diffused hydrogen atoms finally encounter each other and the second step, reaction 3.2, in the reduction takes place within the material. The formed hydrogen gas gives rise to high internal pressures that combine with tensile loads to cause cracks and subsequent failure, see e.g. [11].

Presence of different chemicals increases the risk of hydrogen embrittlement. These substances slow down the second step of the hydrogen reduction at the material surface and hence give the adsorbed hydrogen atoms

more time to diffuse through the surface. One compound that is particularly aggressive is hydrogen sulphide which is found in petroleum fluids, natural gas and oil-well brines, cf. [12].

3.3 HISC

As mentioned, hydrogen embrittlement in combination with tensile loads may cause cracking and in the end, failure of components. This phenomenon is here referred to as HISC. HISC has been observed for both 22Cr and 25Cr duplex stainless steels in subsea applications where these materials are extensively used. The microstructure of the duplex steels in question consists of both ferrite and austenite. Generally, the ferrite acts as anode in corrosive processes and thus gives cathodic protection of the austenite parts of the microstructure. In the case of subsea structures, cathodic protection with sacrificial anodes is used which makes both phases cathode and hence the Flow Loops themselves are acting as cathode in the corrosion process. Hydrogen reduction will thus always take place at the entire Flow loop surface and hydrogen embrittlement is likely to occur. Tensile loads are also present in the working conditions of Flow Loops, which is why there is a risk of failure due to HISC.

3.4 Recommended practice for design

Failure in subsea Flow Loop components is unacceptable for obvious reasons. Consequently, HISC must be avoided and this is ensured by designing in accordance to the Recommended Practice (RP) DNV-RP-F112, see [1]. The following sections correspond to the sections in the RP and summarise the contents.

3.4.1 General

For subsea equipment, duplex stainless steels with cathodic protection have been extensively used. Mostly the usage has been successful, but failures have occurred mainly due to certain combinations of load and hydrogen embrittlement, leading to HISC. DNV-RP-F112 aims to give a best practice for design of components made from duplex stainless steels to avoid HISC and it is based on both small scale laboratory tests as well as full scale tests.

3.4.2 Design philosophy

The RP contains criteria for stress and strain in the material that should be fulfilled. It also gives information about the loads that should be taken into consideration, but it does not give any requirements on the magnitudes of these loads or how to apply them to an FE model. Details about the loads should instead be taken from company requirements.

3.4.3 Loads and conditions

As mentioned, the magnitudes of the loads considered are not discussed, but directions are given on which types of loads that should be taken into account. Occurrence of HISC requires loads applied over certain time intervals. Laboratory tests have shown that these time intervals can be down to a matter of hours. This is why all loads that act on a component for more than a couple of minutes should be included in a HISC analysis. Incidental loads that do not lead to plastic deformation may be neglected. In the following, the load types mentioned in DNV-RP-F112 are described.

Forces due to installation and during operation

The criteria of approval should hold for all pressures that the component will be exposed to while submerged in sea water with active cathodic protection. While assembling and installing subsea systems, forces are always built in since reality is never the ideal design case, these loads should be included in the evaluation. During operation, temperature will change in the components, the seabed will settle and other uncertain loads may occur. These loads are project specific, and differ from case to case, but should always be accounted for.

Residual stresses and strains

Residual stresses may occur during manufacturing, e.g. while welding. Incidental loads leading to plastic deformation may also give rise to residual stresses.

Residual stress and strain, σ_{res} and ε_{res} , from any source must be taken into account. In particular, girth welds contribute and in regions near these welds special criteria shall always be considered. The regions are defined by the distance L_{res} in each direction away from the weld. For a pipe girth weld, L_{res} may be estimated as

$$L_{\text{res}} = 2.5\sqrt{Rt} \quad (3.3)$$

where R is the nominal pipe radius and t is the pipe wall thickness.

When the considered stress or strain direction is parallel to the weld, σ_{res} and ε_{res} do not need to be taken into account. In the case of a girth weld, this means that residual strain does not need to be considered in the hoop direction. If no measurements of the actual magnitude of the residual strain due to weld are known, estimated values may be used. These are given as $\varepsilon_{\text{res}} = 0.15\%$ within the Heat Affected Zone (HAZ) at the weld toe and $\varepsilon_{\text{res}} = 0.25\%$ in the region from HAZ to L_{res} , directed perpendicularly to the girth weld.

Tie-in loads

Whenever a component is attached to a tie-in structure, i.e. an attachment to other components, it may be subject to a fraction of the force acting on this structure. This force should be included in the analysis and understanding of uncertainties related to interface loads like this is important.

Other factors

In addition to loads and stresses, certain design details such as welds, sharp edges and stress raisers are of interest. Corrosion protective coating of components has a risk-reducing effect but normally the strain criteria should be fulfilled just as if no coating is applied. Another thing of importance for HISC is the cathodic protection potential.

3.4.4 Design criteria

DNV-RP-F112 gives two design criteria that are equally applicable. Either one of these criteria satisfied is sufficient for a design to meet the HISC requirements. The criteria only have to be fulfilled when a material surface is exposed to cathodic protection and hydrogen may influence a material volume. Also, only tension may lead to HISC and hence there is no need to fulfil the criteria if the stress state is purely compressive.

The two design criteria given in the DNV-RP-F112 are a linear one and a non-linear one. The non-linear criterion gives limits of maximum principal strain allowable in the material. Strain shall be computed through non-linear FE analysis using a temperature dependent non-linear hardening description of the material. The FE model shall have high enough through-thickness number of elements to capture relevant strain gradients. The strain limits of the non-linear criterion are given in table 3.1.

Table 3.1: *Allowable maximum principal strain.*

<i>Location in the component</i>		<i>Strain criteria</i>	
		Within L_{res} from weld	Outside L_{res} from weld
Outside 5% WT from any surface		Minimum of [0.30%; 0.50% - ε_{res}]	0.30%
Within 5% WT from surface	Fine austenite spacing	1% - ε_{res}	1.00%
	Coarse austenite spacing	0.60% - ε_{res}	0.60%

The strain considered is the sum of elastic and plastic strain. Thermal strains should not be included, but mechanical strains due to stress from mechanical constraints of a thermally loaded component should. The austenite spacing in the microstructure affects the HISC properties of the material and thus the criteria account

for this. There are also different criteria depending on where along the wall thickness (WT) the strain is evaluated.

The linear criterion is not described here since it was decided to use the non-linear criterion for the analyses in this project.

3.4.5 Material requirements

For the mechanical properties of the material, temperature effects must be taken into account through modifying the material's stress-strain curve at elevated temperatures, also known as *de-rating*. Another material property that significantly affects the HISC resistance is the austenite spacing in the microstructure.

DNV-RP-F112 covers 22Cr and 25Cr duplex stainless steels within certain limits. The recommendations are only applicable if the specified minimum yield stress is 450 MPa or less for 22Cr duplex stainless steel and 550 MPa or less for 25Cr duplex stainless steel. The reason for these limits is that higher actual mechanical test data compared to stress parameters from material standards should not be used. Standards and company requirements for materials exist as well as for welding, testing, etc. These are not covered in the RP but are assumed to be followed.

All fabrication techniques that have a decreasing effect on austenite spacing are favourable since fine austenite spacing gives a higher resistance to HISC. Austenite spacing can be measured and is considered to be fine if it is less than 30 micrometres. A number of materials assumed to have fine austenite spacing are listed in the RP. HIP materials, weld material and pipes made by extrusion, seamless rolling or drawing are among these materials.

The material model used for the FE analysis should be defined by elastic-plastic stress-strain curves for non-linear assessment generated according to ASME VIII, see [1]. These curves are based on the elastic modulus, the yield strength, the tensile strength and the type of material, e.g. duplex stainless steel. Temperature dependence is included through de-rating of the stress parameters at different temperatures.

4 Modelling techniques

To verify Flow Loops with respect to requirements for avoiding failure by HISC, FE models are built. The different features of the Flow Loops are described by different models or modelling techniques. This chapter is meant to give some theoretical background regarding these, mostly based on the way they are implemented in the FE program Abaqus CAE.

4.1 Material models

When verifying components through non-linear FE analysis to evaluate HISC criteria, material models with non-linear hardening description should be used according to [1]. Such stress-strain curves may be generated according to the standard ASME VIII, see [13]. The procedure described in this standard generates stress-strain curves based on mechanical parameters and type of material. The mechanical properties in question are elastic modulus, yield strength and tensile strength of the material.

For duplex stainless steels, there are a number of model parameters defining the non linear hardening of the stress-strain curve. These specific parameters are not described in this report since some of that information is confidential. An example of a stress-strain curve generated by this procedure is shown in figure 4.1.

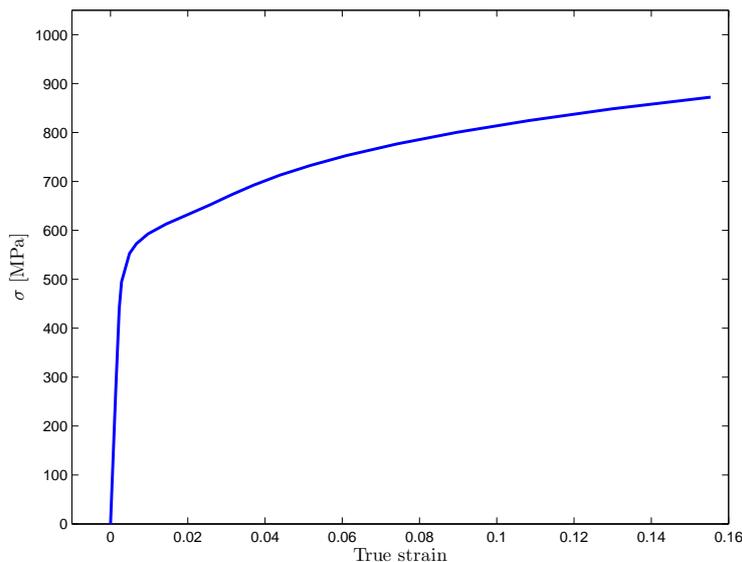


Figure 4.1: *Example of a stress-strain curve generated according to ASME VIII.*

Temperature dependence of the material behaviour should also be included in the models. This is done by de-rating the stress parameters for different temperatures. This means that the yield and the tensile strength are set down to a lower value for a number of temperatures. The higher the temperature is, the higher the de-rating and thus the lower the stress parameters are. The temperature de-rating of duplex stainless steels may be done according to DNV-OS-F101, see [14]. Figure 4.2 illustrates the temperature de-rating curve from [14].

Since thermal loads are considered in HISC analyses, the coefficient of thermal expansion, α , is needed for the material descriptions. Temperature dependent values of the thermal expansion coefficient are taken from ASME II, part D, see [15]. This is also where Poisson's ratio, ν , and temperature dependent values of the elastic modulus, E , are taken. Table 4.1 shows the expansion coefficient, the elastic modulus and Poisson's ratio which are the same for the two duplex steels considered in this report. Note that Poisson's ratio is independent of the temperature.

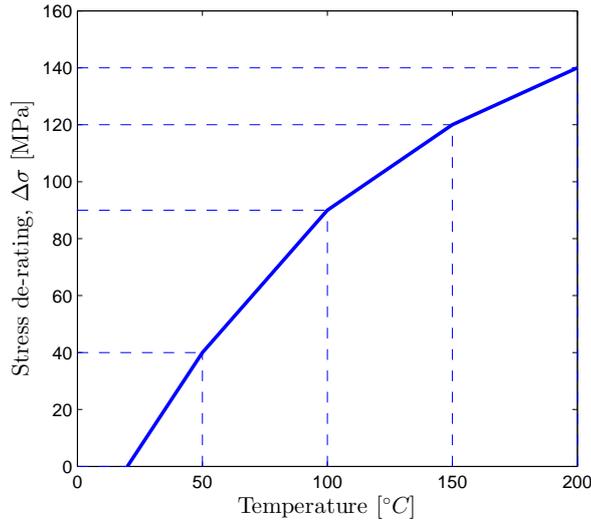


Figure 4.2: *Stress de-rating for duplex stainless steels from DNV-OS-F101.*

Table 4.1: *Thermal expansion coefficient, elastic modulus and Poisson's ratio for the 22Cr and 25Cr duplex stainless steels.*

Temperature [°C]	-75	20	25	50	75	100	125	150
α [$10^{-5}/^{\circ}\text{C}$]	-	1.26	-	1.30	1.33	1.35	1.38	1.40
E [GPa]	209	-	200	-	-	194	-	190
ν	0.31	-	0.31	-	-	0.31	-	0.31

4.1.1 Implementing material models in Abaqus

A temperature dependent, non-linear material model in Abaqus CAE is defined by importing tables describing elastic, plastic and thermal expansion properties. The table with elastic properties contains the elastic modulus and Poisson's ratio for different temperatures. In the table for thermal expansion properties, the thermal expansion coefficient should be specified for different temperatures. These values are given in Table 4.1. The table specifying the plastic material is a combination of several tables, one for each temperature. Each individual table includes values of the true yield stress at different values of true plastic strain at a certain temperature, representing the plastic part of stress-strain curves such as those in Figure 4.3.

To generate these tables of plastic properties, the mentioned procedure from [13] is carried out. For each temperature in Table 4.1, the yield strength and tensile strength of the steels presented in Table 2.1 are de-rated according to [14]. For each of the two duplex stainless steels, a non-linear stress-strain curve can now be generated for each temperature by using the procedure described in [13], which will give values of yield stress versus true plastic strain for each temperature. For 25% Cr Duplex Stainless Steel, some of the resulting true stress versus true strain curves at different temperatures are presented in Figure 4.3 which clearly shows the effect of the de-rating. When solving the FE problem, Abaqus will interpolate between the values in the plastic property tables, using an isotropic hardening model.

4.2 Tie constraints

Two already meshed, separate parts, may be constrained together with a tie constraint. The constraint ties two surfaces together, one is assigned as master and the other as a slave surface. The constraint will remain during the whole simulation, each node on the slave surface will have the same values for all degrees of freedom (e.g. displacements) as the closest point on the master surface, cf. [16]. The slave and the master surface do not need to have matching element type or mesh density.

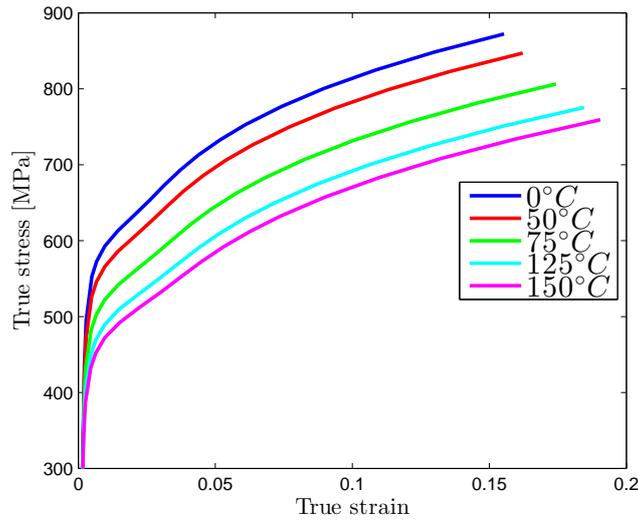


Figure 4.3: *True stress versus true strain for different temperatures, 25% Cr Duplex Stainless Steel.*

4.3 Kinematic couplings

Kinematic coupling allows to define a set of coupling nodes acting as slave nodes to a master node with respect to selected displacement degrees of freedom. These degrees of freedom of the slave nodes are then governed by the translation and rotation of the master node which will cause rigid body motion of the slave nodes, see [17].

The coupling nodes are usually found automatically as all nodes lying on e.g. a selected surface of interest. The master node is commonly defined as a reference point in the Abaqus model. An example of a kinematic coupling between the end of a pipe and a reference point is presented in Figure 4.4.

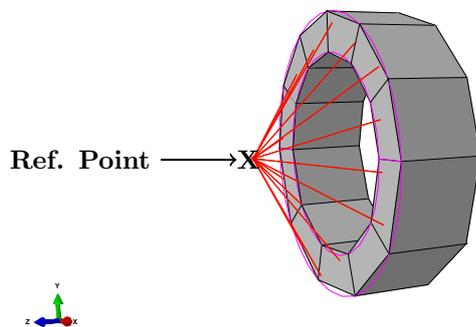


Figure 4.4: *Kinematic coupling restraining all surface nodes by constraining all degrees of freedom in the reference point.*

In Flow Loop simulations, kinematic couplings are often used when applying boundary conditions or point loads at the end of pipes and other fittings. The boundary condition or load is then applied in the master node while typically all the nodes on the cross-sectional surface of an open end are in the slave node set.

4.4 Finite elements

In Abaqus CAE, there are a number of different element types to choose from when an FE model is built. The ones used in the automated Flow Loop simulations in this report are described below.

4.4.1 C3D8R

The element type called C3D8R is a first-order (linear) hexahedral continuum element with 8 nodes. The name stands for *Continuum*, 3D (Solid) with 8 nodes. The "R" stands for reduced integration which means that this element only has one integration point. In comparison, the standard hexahedral element (C3D8) with full integration uses $2 \times 2 \times 2$ integration points. The element node numbering and the location of the integration point of C3D8R are presented in Figure 4.5.

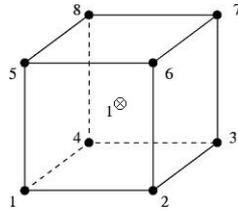


Figure 4.5: The C3D8R element with eight nodes and one integration point at the center.

When using full integration the FE solution tends to be too stiff and the result may be fairly inaccurate when bending is involved, cf. [18], no matter the number of elements through thickness that are used, cf. [19]. This effect does not occur when using reduced integration. Hence, this type of element tends to be softer which could give more accurate results. The drawback is that reduced integration could lead to spurious zero-energy modes. According to [18], the strain energy of a body can be expressed as:

$$U = \frac{1}{2} \mathbf{a}^T \mathbf{K} \mathbf{a} \quad (4.1)$$

where \mathbf{a} is the displacement vector and \mathbf{K} the stiffness matrix. The strain energy U should only be zero in case of rigid body motion, but when using reduced integration this will not hold. Some displacement modes will cause the computed strain in the integration point to be zero, even though the nodal displacements are non zero. These non physical modes are called zero-energy modes since no strain energy has been produced. The shapes of these modes look a bit like a half hourglass and this effect is therefore sometimes referred to as *hourglassing*, e.g. in [20]. The hourglass shape is presented in Figure 4.6.

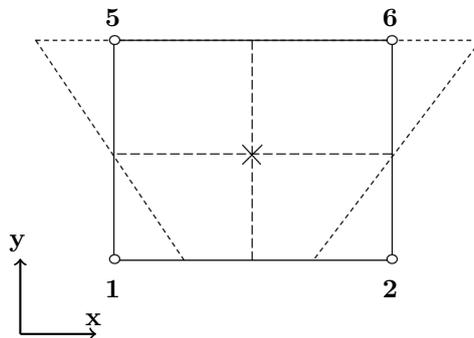


Figure 4.6: A view in one plane of a zero-energy mode, the strain is zero in the integration point.

If hourglassing occurs the FE solution will be inaccurate. This is avoided by using a sufficiently fine mesh resolution which gives small distortion of single elements. For the C3D8R element, a minimum of four elements through thickness is recommended to give a precise prediction of the displacements in bending, cf. [19]. There is also a feature called hourglass control for the C3D8R element in Abaqus. It adds an artificial hourglass stiffness if any zero-energy modes occur, see [21].

Besides the benefits in accuracy of the solution, using elements with reduced integration also reduces the running time and CPU requirements when solving the FE problem.

4.4.2 C3D6

The C3D6 element is also a standard element in Abaqus. It is a first-order (linear) triangular prism continuum element with six nodes, see Figure 4.7.

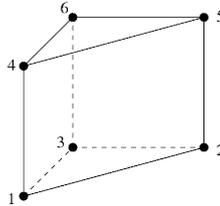


Figure 4.7: *The C3D6 element with six nodes.*

This element uses full integration and will provide accurate results only when using very fine mesh, otherwise the results will be too stiff, cf. [20]. This type of element is recommended to be used far away from areas of interest, due to the poor results. They may, however, be easy to use when meshing difficult geometries and are therefore a good way of completing a difficult mesh.

5 Automation of Flow Loop HISC analysis

The goal of this project is to automate verification of Flow Loop design with respect to HISC. To evaluate the risk of HISC in general, FE models of components may be built with loads and boundary conditions (BCs) representing its working conditions. The output of the FE model is then checked against criteria in DNV-RP-F112, as described in Section 3.4, to determine whether a design is strong enough to last throughout its service life.

For Flow Loops, the FE modelling is highly repetitive and can to a large extent be parameterised. This goes for the geometry as well as for loads and BCs. To collect these parameters from a user, a Graphical User Interface (GUI) may be created as a plug-in for Abaqus CAE. There is also the possibility of building entire FE models in Abaqus by scripting commands. Both the GUI and the scripts are created in the object-oriented programming language Python, for the interested reader further information about Python may be found in [22].

The GUI and the scripts may be linked together. After collecting parameters in the GUI, commands containing these parameters may be sent from the GUI to build different parts of the model in Abaqus CAE. This way, a Flow Loop FE model could be quickly built in a standardised way. The parameters given in the GUI could also be stored to create documentation of a model automatically.

This project has been entirely focused on utilizing the possibility of creating a customised GUI in Abaqus CAE linked to different scripts. Therefore, this chapter is based on the resulting plug-ins that have been built. The analysis procedure goes hand in hand with the functionality of the plug-ins, which is why following these will give a good line of argument. The different modelling methods used will be described as the different parts of the plug-ins are presented.

5.1 The Flow Loop plug-in

The Flow Loop plug-in is accessed from the *Plug-ins* menu in the main menu bar of Abaqus CAE. This plug-in is created to build models of Flow Loops consisting of mainly standard pipes, elbows, tees and flanges, but also other parts, here referred to as special blocks, may be imported to a model. A number of loads and BCs that are commonly applied in FE models of Flow Loops are available in the plug-in and may be time efficiently applied.

5.1.1 Initiation of Flow Loop model

Once you open the plug-in, a first dialog box named *Model Initiation* appears, see Figure 5.1. Here, the user may either choose to create a new Flow Loop model, Figure 5.1a, or to continue working on an existing model created using the Flow Loop plug-in, Figure 5.1b.

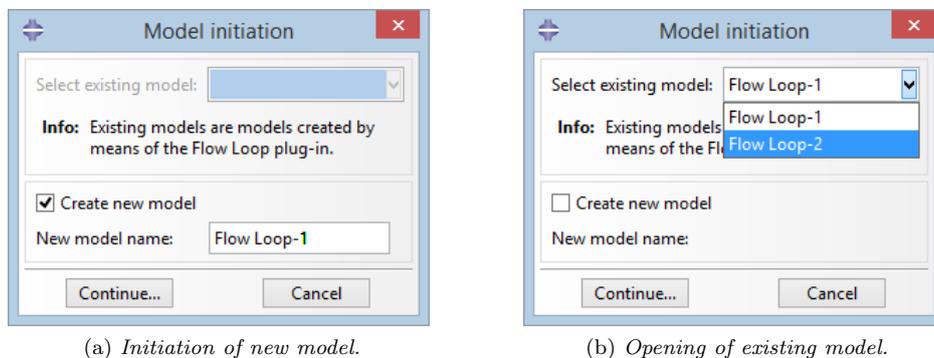


Figure 5.1: *Model initiation window.*

When checking the *Create new model* check button, specifying a new model name and clicking *Continue*, a new model is created in Abaqus CAE. A few things are initially added to the model to transform it into a Flow Loop model, these are:

- **A first branch in the Flow Loop Tree**

- As will be described later in the report, the geometry of a Flow Loop model is built as a tree with one or more branches. The first branch must always exist and is created in the model initiation.

- **A first loading step**

- A Flow Loop model may have several load steps, at least one and at most three. The first loading step, *Step-1*, is created in the model initiation.

- **Default data about the model**

- For the model to work as a Flow Loop model, some data need to be accessible for the plug-in to read. These data specify internal pressure, temperature, etc. in different steps of the analysis and are set as default values initially.

- **Materials**

- Models of the two most commonly used materials in Flow Loops, the duplex and super duplex stainless steels discussed in Section 4.1 are created in the model initiation. This is done by importing tables that define temperature dependent elastic, plastic and thermal properties for the materials.

After initiating a new model or choosing to work with an existing one, the main window of the Flow Loop plug-in will pop up, presented in Figure 5.2. This is where most of the work is done. Starting from the top, the main window consists of a field specifying the current model name. Just beneath the model name there is a tab book with three different tabs, *Geometry*, *Loads* and *BCs*. At the bottom there are two buttons, *Write Input* and *Cancel*.

5.1.2 Main window - geometry tab

The first tab of the Flow Loop plug-in main window is the geometry tab. This tab gives possibility to create parts and to assemble them together. Parts are placed in the *Flow Loop Tree* that is the main part of the geometry tab. The order the parts are placed in determines what the assembly is going to be like.

The first part in the first branch of the Flow Loop Tree will be inserted to the assembly, followed by the rest of the parts in the same sequence as they are arranged in the tree. Wherever a branch divides into two new branches in the tree, the Flow Loop assembly will also split into two directions. Parts that will give rise to new branches are tees and three way special blocks. Examples of how the Flow Loop Tree works will be shown further back in the report.

As Figure 5.2 shows, there are two buttons in the lower *Actions* part of the geometry tab, *Create new part* and *Assemble parts*. Upon clicking the first of these two buttons, the window in Figure 5.3 will appear.

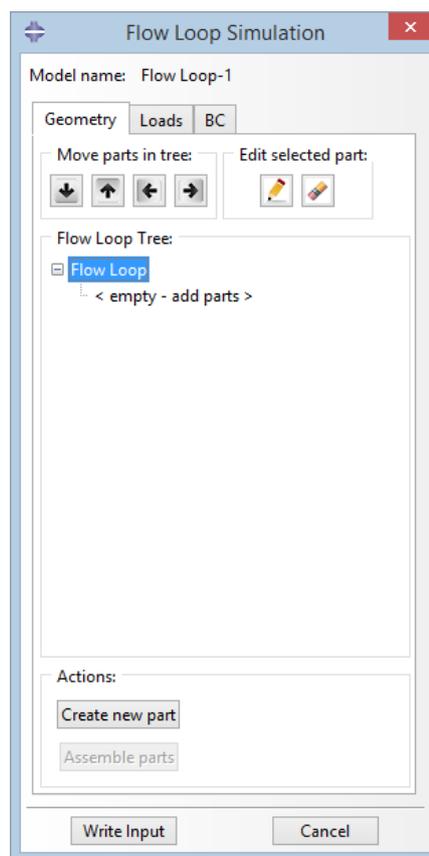


Figure 5.2: *The main window of the Flow Loop plug-in; the geometry tab.*

In this window, the user picks type of part to create. The standard parts of Flow Loops; pipes, elbows, flanges and tees can be created and other parts created outside the Flow Loop plug-in, special blocks, can be imported.

Creating standard parts - pipes, elbows and tees

When the user wants to create a pipe, an elbow or a tee, the corresponding button in the create part window, see Figure 5.3, is clicked. Depending on which part that is selected the part windows shown in Figure 5.4a, 5.4b , 5.4c will appear for a pipe, elbow and tee respectively. For these parts, the part name together with a number of parameters specifying the dimensions of the part are asked for. The nominal pipe size (NPS) and wall thickness can be selected in drop lists. The values in these drop lists are taken from the different tabulated standard values described in Section 2.2 and are given in inches. The plug-in is working with dimensions measured in millimeters, so standard dimensions are converted into millimeters. The user have the possibility to specify custom values if standard dimensions are not used.

In addition to the input parameters nominal outer diameter, D , and the nominal thickness, t , some other parameters are needed to fully define these standard parts. In all windows shown in Figure 5.4a–5.4c there is a drawing of the part to visualise the measures. For a pipe the length, L , must be specified and is always measured in millimeters since it is not a standardised dimension. The parameters specific for an elbow are the angle, θ , and the center-to-end measure, B . An elbow will make a Flow Loop turn and the angle and the radius of this turn will be determined by these parameters. For tees, there are two center-to-end values, run, C , and outlet, M . These parameters determine the length of the different ends of the tee.

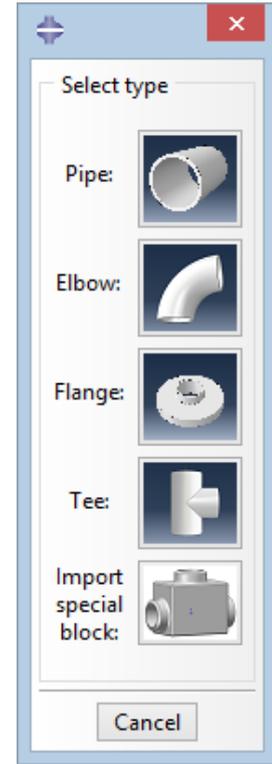


Figure 5.3: The create new part window of the Flow Loop plug-in.

For all parts the material is selected in the *Material* drop list. The user selects one of the two materials that was imported in the model initiation, *22% Cr Duplex Stainless Steel* or *25% Cr Super Duplex Stainless Steel*. Other materials could also be selected by defining new ones manually in Abaqus CAE, these will then appear automatically in the Material drop list.

Additionally, reduction parameters need to be specified. These parameters are *erosion*, *mill tolerance*, *corrosion on outer surface* and *corrosion on inner surface*. The erosion is coming from the flow of production medium inside of the Flow Loop and the specified value is thus added to the inner radius as it represents removed material from the inside. The inner corrosion value is used in the same way. In a similar fashion, the outer corrosion parameter is deducted from the outer diameter as outer corrosion removes material from the outside surface of a Flow Loop. The mill tolerance is a manufacturing tolerance given on the nominal thickness of a piping part and thus the thickness is reduced with respect to this tolerance before considering any other parameters. Altogether, the reduced outer and inner diameter, D_{red} and d_{red} , of the standard piping parts are computed as:

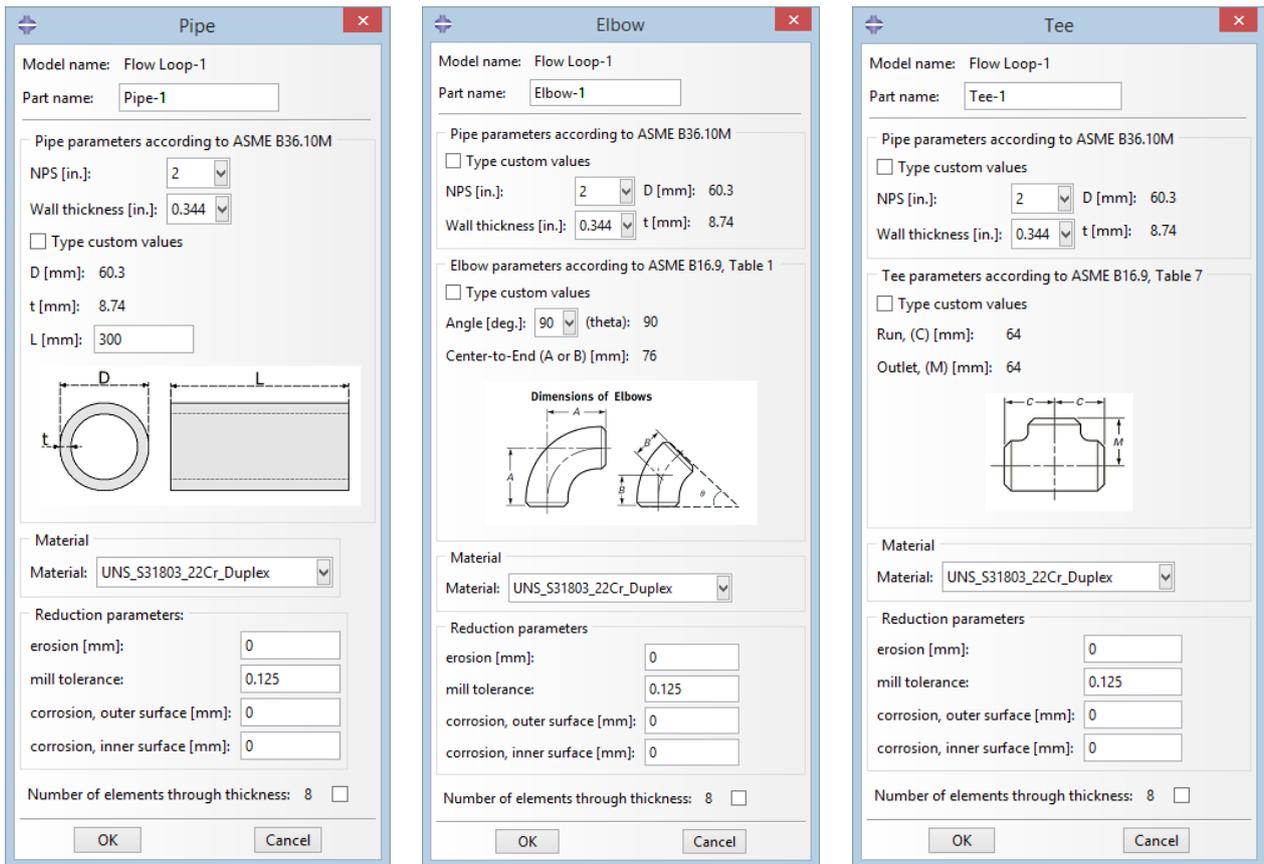
$$D_{red} = D - 2c_o$$

$$d_{red} = D - 2t(1 - m) + 2e + 2c_i$$

where:

D = nominal outer diameter	t = nominal thickness	c_o = outer corrosion
c_i = inner corrosion	e = erosion	m = mill tolerance

The parameters D_{red} and d_{red} are calculated in the background when the part is created and define the actual dimensions of the part in the simulation model.



(a) The pipe window.

(b) The elbow window.

(c) The tee window.

Figure 5.4: Standard part windows of the Flow Loop plug-in.

The last parameter common for pipes, elbows and tees, is a mesh parameter possible to modify. This parameter represents the number of elements through the thickness of the part and is set to 8 as default, a number recommended by experienced engineers at Aker Solution. It is possible to increase the parameter but not to decrease it, this restriction is made to always ensure a high quality of the mesh. An investigation about the mesh quality and convergence check is done and presented in Section 6.3.

When creating standard parts with the Flow Loop plug-in, these parts become automatically meshed. For pipes, elbows and tees, the C3D8R element is exclusively used. As mentioned, what governs the meshing of standard piping components is the number of through-thickness elements. The mesh sizes in other directions are based on the size through the thickness. Consequently aspect ratios and corner angles of the elements are, depending on the part type, the same or very similar no matter the dimensions of the part. The mesh quality is thus always guaranteed and within the limits of what Abaqus requires not to give warnings, see [23] for a table of limits on aspect ratio and corner angle.

An example of a pipe created in the Flow Loop plug-in is shown in Figure 5.5, it has the default value of eight elements through the thickness. A slice of the cross section is also presented in Figure 5.5b. Note that the element rows at the surface do not have the same thickness as the interior element rows. The mesh is created this way due to the different HISC criteria within 5% of the thickness from any surface of a part compared to the criterion for internal regions of a part. The element rows at the surfaces cover the regions of the surface HISC criteria and thus the strain values in elements of different regions can be evaluated against different criteria. These surface and internal regions are created in all standard piping parts.

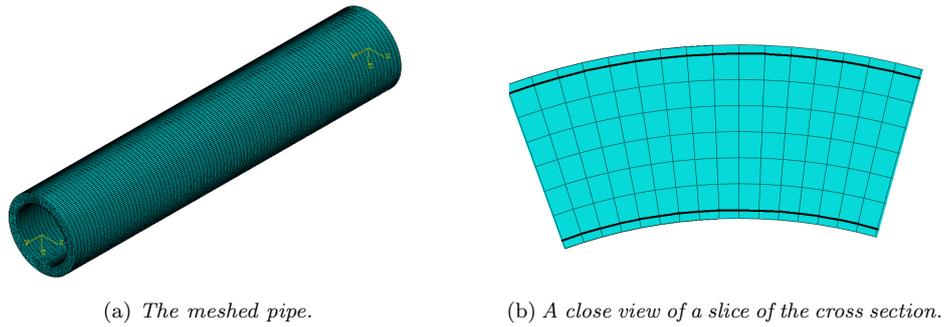


Figure 5.5: *Example of a pipe created in the Flow Loop plug-in.*

In addition to the geometry, there are several features that are automatically created for each part when it is created by the script. Features are coordinate systems, reference points, sets containing geometry of interest, etc. and are visible for the user in Abaqus. These are necessary for other functionalities of the plug-in to work. To be able to assemble the parts, coordinate systems and different geometry sets containing the interaction surfaces are created. Other features that are automatically defined are sets containing all internal pressure faces where the internal pressure is applied and sets containing all geometry used to apply the thermal loads. Figure 5.6 shows an example of two elbows with different angles, created with the plug-in. Due to this angle they will make the Flow Loop assembly to turn in a new direction. Because of the turn, it matters how the elbow is directed, rotation-wise. Compared to a pipe, extra features to adjust this rotation are created with an elbow.

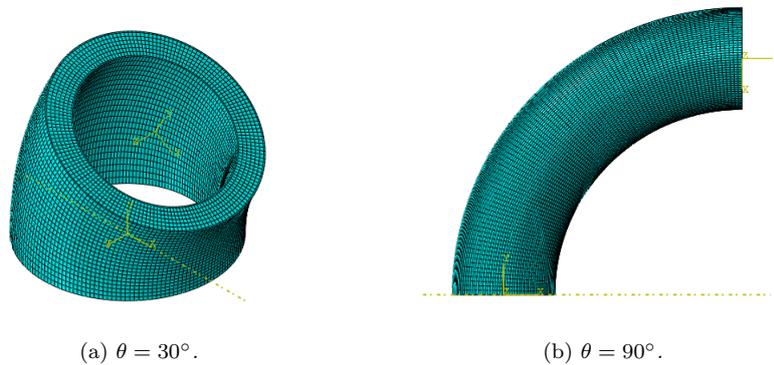


Figure 5.6: *Example of different elbows created in the Flow Loop plug-in.*

Data is stored in the background about each part, e.g. telling the plug-in where to place the part in the Flow Loop Tree. When just created, all parts except tees and three way special blocks are placed at the end of the first branch of the tree. Tees and three way special blocks are placed at the leftmost end of the tree, see the *Assembly* section further down. There is an important difference between these and the other piping parts in the Flow Loop plug-in. They divide a Flow Loop branch into two new branches and this demands more than the two way pipes and elbows. First of all, there have to be additional features to be able to assemble new parts to two different ends. Secondly, two new branches of the Flow Loop Tree have to be created and initiated.

Creating standard parts - flanges

Flange is another type of part that may be created with the Flow Loop plug-in. To create a flange, a number of dimension parameters need to be specified in the flange window presented in Figure 5.7. These are in accordance with the ISO10423 standard, see Section 2.2. Part name, material and reduction parameters are specified. For flanges, the reduction parameters are primarily there to make it easy for the user to make the flange match the part attached to it that could for example be a pipe. In reality, all of the reduction parameters

do not apply to a flange. A last parameter is the *flip* parameter. Through this parameter, the user may change the direction of the flange in the assembly.

Just as for the pipes, elbows and tees, flanges are meshed when created. To mesh a flange automatically is quite cumbersome and the quality of the flange mesh might not be as good as for the other parts. The flanges are not subject to the Flow Loop HISC analysis, they are individually evaluated. The reason for including flanges in the Flow Loop models is just to give reliable results in the pipes, elbows and tees. This is why the mesh quality is not as important and also the reason why bolt holes are not included in the flange geometry. What is interesting, to obtain good results in other parts of the model, is the global stiffness of the flange. Therefore the bolt holes' contribution to the stiffness is assumed to be negligible. The dominating element type is C3D8R, but also C3D6 is used in radii since this element type is easier to use when meshing irregular geometries. The mesh of a flange can not be modified by the user as for the piping parts, it is however parameterised in the meshing script.

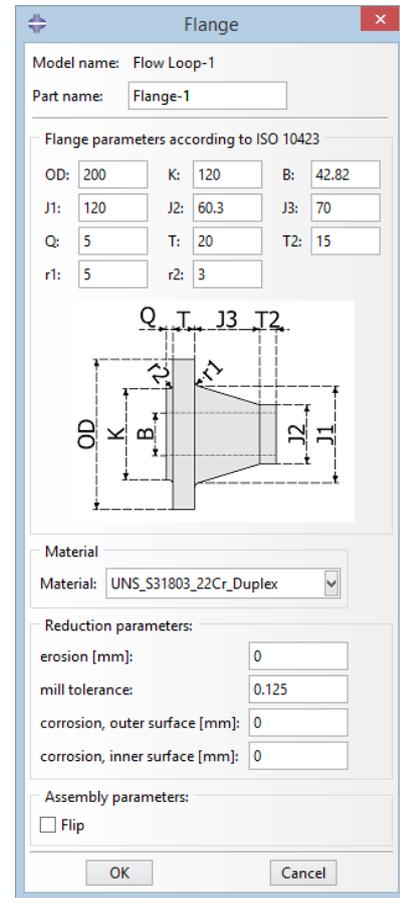
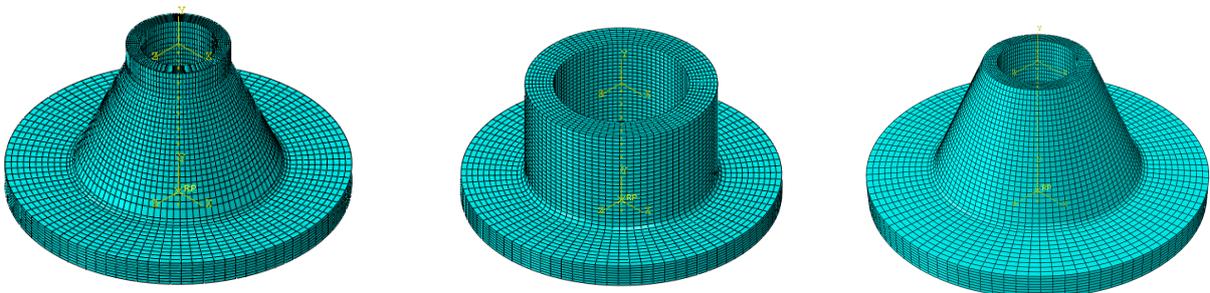


Figure 5.7: The flange window of the Flow Loop plug-in.

Also in the case of flanges, features for enabling assembly and other functionality are created. Three examples of flanges created in the Flow Loop plug-in, with different input parameters, are presented in Figure 5.8.



(a) Predefined dimension parameters from the flange window in Figure 5.7.

(b) Parameters $J2 = J1$.

(c) Parameter $T2 = 0$.

Figure 5.8: Examples of flanges created in the Flow Loop plug-in with different dimension parameters.

Importing special blocks

A Flow Loop may consist of other parts than just the standard ones previously described. Since an arbitrary part can not be parameterised, the possibility to import externally created parts was added to the plug-in. Clicking the import special block button in the create part window in Figure 5.3 will yield the window in Figure 5.9.

The part is created by the user as an ordinary part in Abaqus CAE, without using the Flow Loop plug-in. It must also be meshed manually and assigned a material. When this is done, the part may be imported to the Flow Loop plug-in via the special block window in Figure 5.9. First, the user must select the externally created part in a drop list containing all part names in the simulation model that are not created with the plug-in. Then the associated part type should be selected, one way, two way or three way. Depending on selected type the window will update and ask the user to pick a number of surfaces and points in the part, the number depends on the selected part type. Figure 5.9 shows the surfaces and points to select that correspond to a three way special block. Note that there is also a figure explaining where to pick these entities on the part. From these points and surfaces the plug-in automatically creates the features needed to transform the part into a Flow Loop part. Clicking *OK* in the special block window after specifying the right parameters will make the imported part appear in the Flow Loop Tree. Depending on whether the part is one, two or three way, it will act accordingly when it appears in the tree.

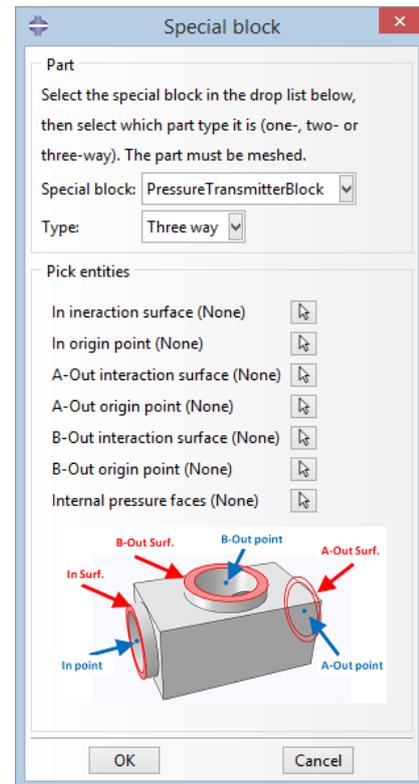


Figure 5.9: *The import special block window of the Flow Loop plug-in.*

Assembly

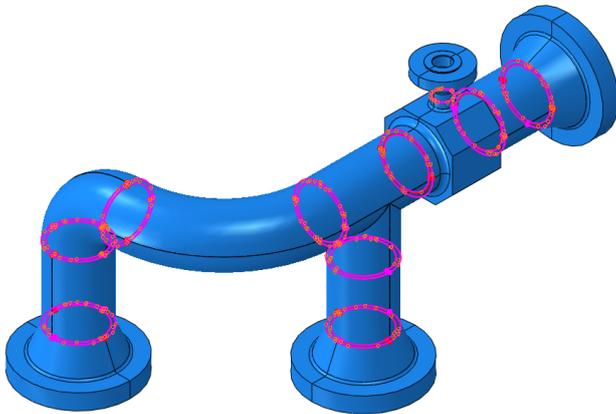
As soon as more than one part has been created in or imported to a Flow Loop model, the Flow Loop Tree may be assembled. It is assembled in the same order as the parts are placed in the Flow Loop Tree. As explained above, all created parts are visible in the tree. The separate parts of the Flow Loop may be created in an arbitrary order and then be moved around in the different branches of the tree by using the move buttons above the tree in the geometry tab, see e.g. Figure 5.2. If the user wants to change the parameters for any standard part that already has been created, it may be done by selecting the part in the tree and clicking the edit button next to the move buttons. If clicking the delete button instead, the part will be deleted from the Flow Loop model. Apart from assembling the parts of the tree, pressing the *Assemble parts* button will do a number of additional things to the model. These are:

- **To insert the parts of the Flow Loop Tree and move them to the right positions**
 - Each part has in-going features and out-going features for at least one out-going end. These features consist of among other things; interaction surfaces and coordinate systems. Based on these features and the order of the Flow Loop Tree, the parts are put together to the complete Flow Loop geometry.
- **To keep track of regions for internal pressure and temperature load**
 - Each part will bring in its own regions to be loaded by mechanical pressure and elevated temperature. These are gathered in global regions to be used later on.
- **To add tie constraints between the assembled parts**
 - Since the parts of a Flow Loop are welded together, the interfaces between them are modelled as tie constraints.

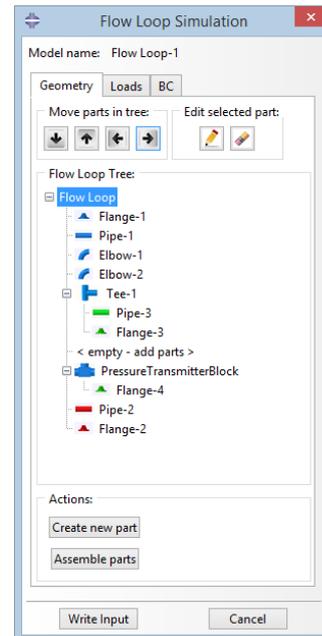
- **To activate the rotation function of the Flow Loop plug-in**

- Elbows, tees and all special blocks need to be rotatable to enable the user to create any desired Flow Loop geometry. Every rotatable part will generate its own rotation window where an angle may be set.

An example of an assembled Flow Loop Tree is shown in Figure 5.10. Note the highlighted tie constraints between the connected parts. The plug-in is made as a powerful tool when it comes to assemble large, complex Flow Loops with many branches. One such Flow Loop could take hours to manually assemble with defined tie constraints, while it is done instantly with the Flow Loop plug-in. Another example is attached in Appendix A, showing a complex Flow Loop with corresponding geometry tab.



(a) Assembled Flow Loop with the tie constraints marked in red.



(b) The main window corresponding to the Flow Loop geometry in (a).

Figure 5.10: Example of an assembled Flow Loop Tree.

5.1.3 Main window - load tab

In the Flow Loop HISC analysis, there are two loads that are always included. These are:

- **Internal pressure**

- All the pipings in a Christmas Tree will or may be subjected to internal pressure emanating from the well.
- Pressure variations within the Flow Loop are not included in the modelling, the internal pressure is applied uniformly.
- In the Flow Loop plug-in, the internal pressure is applied in at least one, at most three load steps. The pressure may be of different magnitude in each step.

- **Temperature**

- Due to pressure fluctuations and elevated temperature of the production media, the temperature of a Flow Loop will be raised substantially.

- The temperature load is applied as a uniform temperature field. The HISC RP [1] says to include temperature gradients but these are not included by the Flow Loop plug-in. This is motivated by that the Flow Loops usually are insulated and thus the much colder sea water does not set the temperature of the outer Flow Loop surface.
- In the Flow Loop plug-in, the temperature is applied in at least two, at most four steps. The first step is an initial step and the temperature value here is the initial, stress free temperature. The rest of the steps are the load steps and the temperature may here be modified from the initial value to different levels.

These two loads are treated in the load tab of the Flow Loop plug-in main window, shown in Figure 5.11. In the first section of the tab, *initial step*, the initial stress free temperature is specified and in the remaining part of the tab, *load steps*, the values of the internal pressure and the temperature are set in the different load steps. Note that the first parameter in the load steps section is *number of load steps*. This parameter may be set to the values 1, 2 or 3, yielding either one, two or three step tabs for specifying the load values.

The reason for wanting to work with different load steps is to be able to try different combinations of loads and also to be able to apply the loads in the sequence best representing the real case. This might be of interest since plastic effects are included and thus the loading order affects the result in the end.

The load values may be set by either picking a value in a drop down menu or by typing a custom value in a text field. As described in Section 3.4.2, details about the loads to apply should follow company requirements. The values in the drop down menus are taken from the standard ISO10423, see [7]. Here, design pressures and temperature ratings are given, used by Aker Solutions to classify Flow Loops.

5.1.4 Main window - BC tab

The final tab of the main window is the BC tab. This tab enables the user to apply a few types of BCs and loads specific for Flow Loop models in a highly time efficient way. The BCs and loads possible to apply are:

- **Flange BC**

- Flanges are often attached to, relatively, quite rigid connections and are therefore often subjected to a displacement BC in Flow Loop models with rotation set to zero.
- The Flange BC consists of a kinematic coupling of one of the interaction surfaces of the flange to a reference point at the center of this surface's inner edge and a displacement BC applied in this reference point.
- The displacement could be set to zero or be specified in three directions. Non-zero displacement values represent misalignment that could have occurred e.g. while mounting the Flow loop in a Christmas Tree. The displacement directions are defined by a local coordinate system with origin the aforesaid reference point and with its z-axis in the outward direction of the flange.
- It is possible to choose in which of the load steps this non-zero displacement is to be applied.

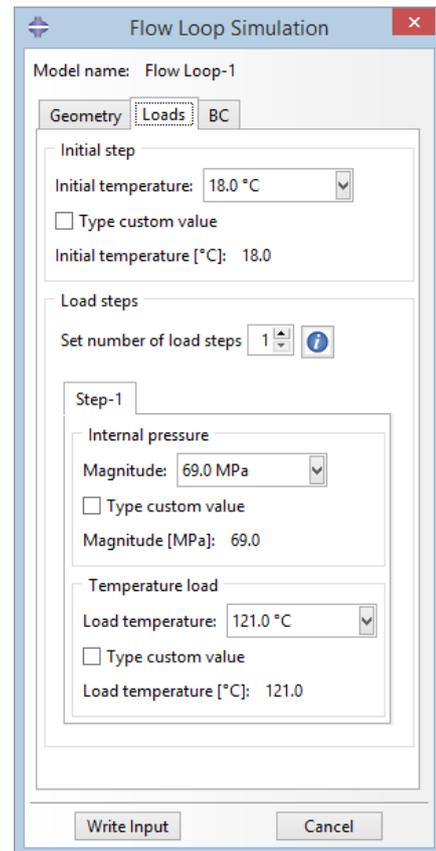


Figure 5.11: The main window of the Flow Loop plug-in; the load tab.

- When creating a flange in the Flow Loop plug-in, it comes already prepared with special features to make it easy to apply the Flange BC.

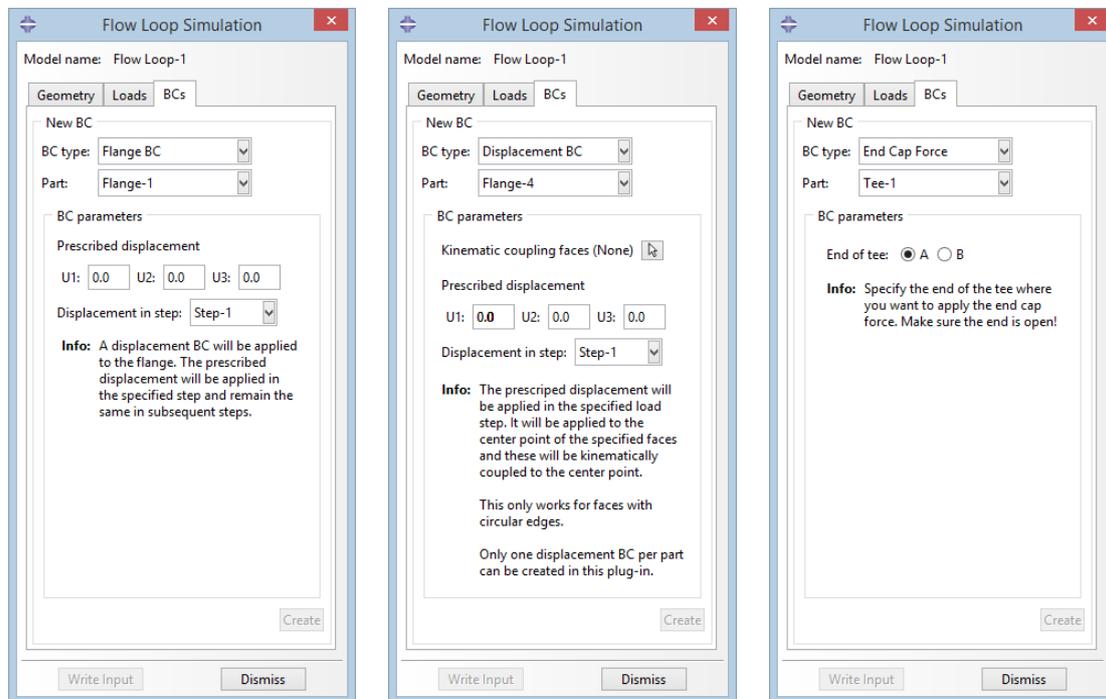
- **End Cap Force**

- A Flow Loop model geometry often contains open ends where the flow in reality continues into another part. These other parts, not included in the model, will in reality transfer force from the pressure of the fluid to the parts that are included in the model. This force will correspond to the internal pressure multiplied by the cross-sectional area of the open end and is calculated automatically for parts with circular openings.
- The End Cap Force consists of a kinematic coupling of the surface surrounding the open end of a part to a reference point at the center of this surface’s inner edge and a concentrated force applied in this reference point.
- End Cap Forces may be applied to standard parts as well as special blocks. Depending on the part type, the open end’s surrounding surface might need to be picked by the user and then needs to have circular edges for the location of the reference point and the surface area to be computed. The magnitude of the End Cap Force will always follow the magnitude of the internal pressure in the different load steps.

- **Displacement BC**

- Sometimes other parts than flanges are subjected to displacement BCs, e.g. special blocks.
- The Displacement BC is almost identical to the Flange BC, with the difference that the interaction surface is replaced by a user defined surface in the kinematic coupling. The user picks faces to form this surface and the location of the reference point where the BC is applied is automatically computed.
- Any part could be subjected to a Displacement BC. The kinematic coupling surface always needs to be picked and must have circular edges.

The different interfaces of the functionalities of the BC tab are presented in Figure 5.12.



(a) *Flange BC.*

(b) *Displacement BC.*

(c) *End Cap Force.*

Figure 5.12: *The boundary conditions tab.*

5.1.5 Load steps

To illustrate the roll of the load steps, an example is given in Figure 5.13 below with initial step, a first step and a second load step at 0, 1 and 2 on the horizontal axis, respectively. The scenario is that a Flow Loop is first mounted on a XMT in the factory. The fit is not perfect, so a flange needs to be forced in place. This is modelled as a Flange BC with non-zero displacement which is applied in the first load step, see the blue dashed-dotted line. The mounting is done in room temperature and subsequently the structure is lowered down to the sea bed and its working environment. During its life it will have to endure different pressure and temperature conditions and thus, in the second load step, the temperature is elevated from an initial value to a load value and an internal pressure applied. An End Cap Force follows the pressure directly. The temperature is represented by a black, dashed line while a red line shows the pressure and the End Cap Force.

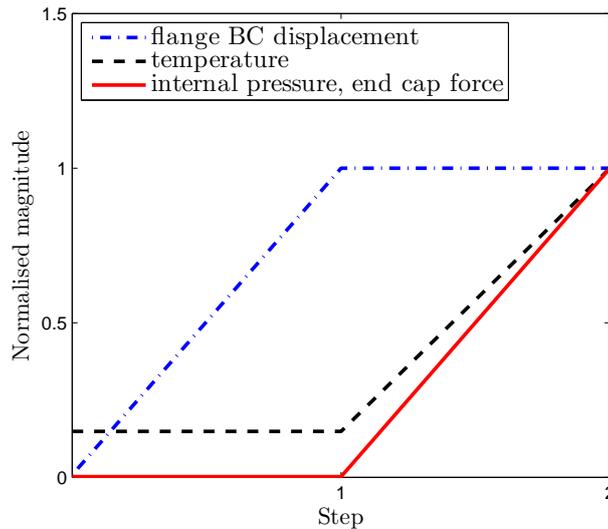


Figure 5.13: *Example of a step sequence with initial step at 0 on the x-axis and two load steps at 1 and 2.*

5.1.6 Write Input

When the parts of the Flow Loop model are assembled and loads and boundary conditions are defined, all steps necessary to write the input file are done. The input file is a text file containing all information of the model. Upon pressing the *Write input* button of the main window, the Flow Loop plug-in will ask for a directory path. Once this is specified, the following things will happen to the current Flow Loop model:

- **Internal pressure is applied**

- A uniform pressure will be applied to an internal pressure surface created by the assembly function. This surface contains all internal faces of the Flow Loop. The magnitude of the pressure in the different load steps will be gathered from the load tab of the main window.

- **Thermal load is applied**

- A temperature field will be created and applied to the entire Flow Loop geometry. The initial temperature and the temperatures in the different load steps will be gathered from the load tab of the main window.

- **Output is requested**

- A field output is created to write the variables needed to perform the HISC evaluation to the output file when the input file is run in the solver.

- **An input file is written**

- An input file is written and saved in the specified directory. The entire Flow Loop model created in Abaqus CAE, with its so called *Model Database* (MDB), is also saved to the same folder. The input file may then be run in the Abaqus implicit solver to obtain the desired results.

- **A model report is written**

- The different parts of the model are well documented in a text file, also saved in the specified directory.

5.1.7 Output from the plug-in

The input file that is created when clicking the *Write input* button is solved with Abaqus's implicit solver. The results are then stored in a file called *Output Database file* (ODB file). This file may be opened in Abaqus to view the results, but it is also possible to open it in the background with Python scripts to evaluate the results.

Field outputs

The requested field outputs are the logarithmic total strain, ϵ_{LE} , and strain originating from the applied temperature field, ϵ_{THE} . They are called field output in Abaqus since they consist of values from each integration point in every element of the FE model. All strain components are included.

Model report

The model report mentioned in Section 5.1.6 is a text file containing information about the Flow Loop created by the plug-in. Its main function is to make the background processes made by the plug-in visible to the user and the contents are:

- **Simulation model information**

- The first page of the report informs the reader of which MDB that is described and the date it was created.

- **Parameters of the Flow Loop parts**

- All parts created or imported with the Flow Loop plug-in are listed with corresponding dimension and reduction parameters specified by the user. The calculated reduced outer and inner diameter, D_{red} and d_{red} , are also presented.

- **Interactions between parts**

- The tie constraints automatically created by the assembly functions are written out in a list. It contains information about the constraint's name, parts in interaction, which surfaces on the parts that act as master and slave surfaces and which tolerance that is used.

- **Kinematic couplings**

- All kinematic couplings in the simulation model are written out by name. Additionally, the name of the surface and reference point that are coupled are given including the coupled degrees of freedoms.

- **Mesh**

- For each part that is created with the plug-in, important information about the mesh is presented. The information consists of; number of elements, element types, average and worst aspect ratio as well as smallest and largest corner angle. The aspect ratios and corner angles are measures of how distorted the elements are. Distorted elements should be avoided, see Section 4.4.1. For the C3D8R element type, since it is a hexahedral element, the corner angels should ideally be close to 90 degrees.

- **Loads**

- The loads defined in the load tab are listed in the report. The initial temperature and load temperature corresponding to the first load step, *Step-1*, are always listed. If the user chose to define more load steps, the temperatures corresponding to these steps are also printed. The same goes for the internal pressure.

- **Boundary conditions**

- Displacement boundary conditions and flange boundary conditions are both listed in the report. The displacements u_1 , u_2 and u_3 are specified for both the initial step and each load step that is defined.
- End Cap Forces are also listed. The magnitude of the computed force acting on the reference point and computed area are specified so that the engineer reading the report knows what force magnitudes that are used.

An example of a Model report is attached in Appendix B, it was automatically generated when the Flow Loop simulation model described in Section 6.1 was created.

5.2 The HISC Evaluation plug-in

The Flow Loop plug-in automated the procedure of creating a Flow Loop FE model since this have been found time consuming. To evaluate a Flow Loop's resistance against HISC according to DNV RP-F112, see [1], has also been found time consuming to do strictly according to the RP. The common way of evaluating results against the criteria has therefore been by manually finding maximum strain values for each part of the Flow Loop and making sure that these are lower than the most stringent limit in the RP. This procedure ensures that all regions of a Flow Loop fulfill the criteria, but is still a simplified, over conservative method with no consideration of regions close to welds and surfaces. Including these regions with corresponding strain criteria when investigating the risk of HISC will give a more accurate evaluation with a greater chance of getting Flow Loops passed the criteria.

This section describes how the results obtained from the Flow Loop plug-in, described in Section 5.1.7, are automatically analysed to determine whether the different parts of a Flow Loop fulfill the non-linear criteria in [1]. This is made by Python scripts reading the ODB file, scripts which are linked to a second plug-in called the *HISC Evaluation* plug-in.

5.2.1 Initiation of the HISC Evaluation plug-in

The *HISC Evaluation plug-in* may be accessed from the main window in Abaqus. The *HISC Evaluation* window, presented in Figure 5.14a, is the first step where the user is asked to specify a pathname corresponding to the file containing the MDB of the Flow Loop to analyse. This file name is of the format *.cae. This file contains all information specific to the Flow Loop, i.e. all input parameters specified by the user when using the Flow Loop plug-in. A Flow Loop model created outside the Flow Loop plug-in can therefore not be run in this application.

When an MDB is selected and the user clicks the *Continue* button, the window is replaced by another, see Figure 5.14b. In this new window, the user is asked to specify which Flow Loop model to evaluate since an MDB might include several models, see Section 5.1.1. All models in the MDB are shown in the *Select model* drop list. When a model has been selected, the *Select step* droplist is automatically updated with all load steps corresponding to the actual model. When the Flow Loop model and load step to be evaluated are selected, the program knows where to find the wanted results, results that are stored in the *output database file*. The pathname to this file should also be given as input. The HISC evaluation is then started by clicking *Run Analysis*.

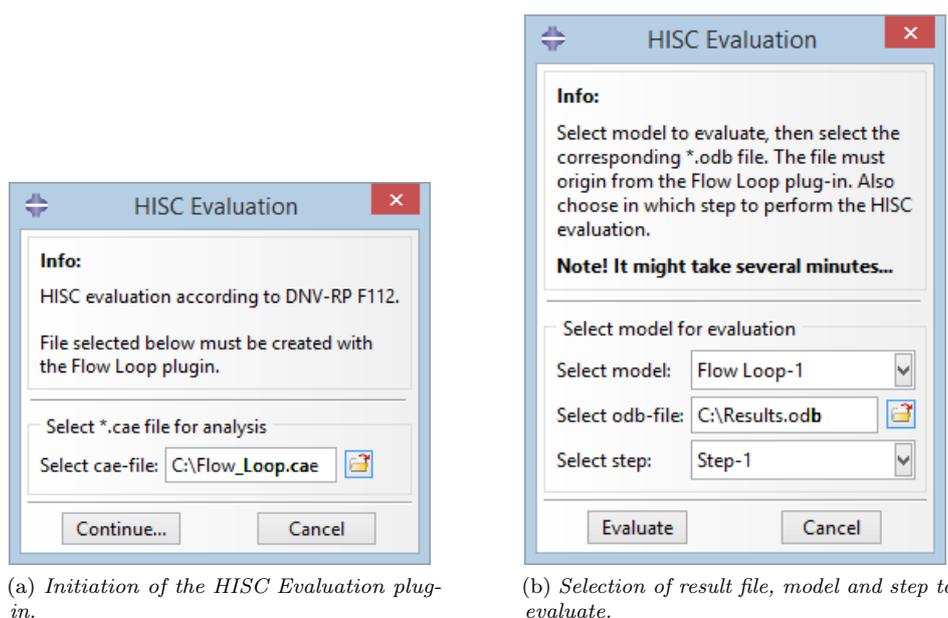


Figure 5.14: Windows of the HISC Evaluation plug-in.

5.2.2 Part types included in the analysis

Since no geometric modelling of the welds is made in the Flow Loop models, they are included by defining different regions near the weld interface with associated strain criteria, presented in Table 3.1. A geometrical view of these partitions is shown in Figure 5.15, presenting a lengthwise cross section of a pipe.

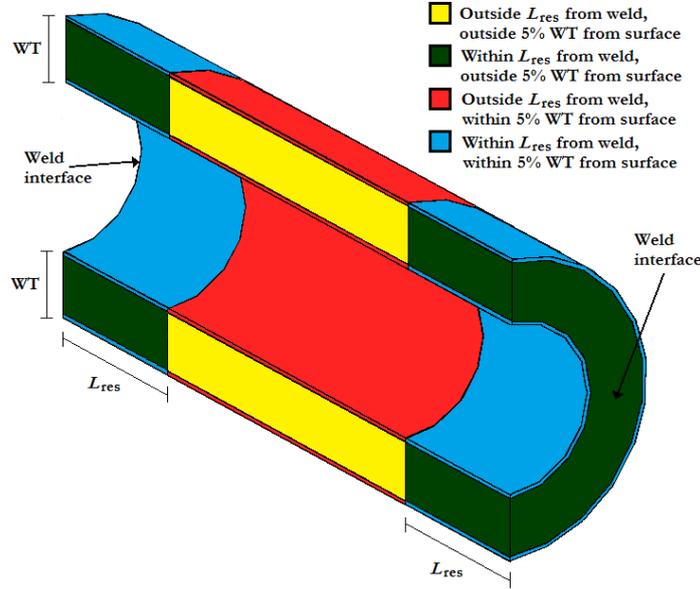


Figure 5.15: *Regions of a pipe with different strain criteria.*

The distance L_{res} depends on the wall thickness, WT , and radius, R , of piping parts, see Equation 3.3. In the analysis script, the location of any point in an analysed part can thus be determined to be in one of these four regions. Special blocks however, can not be evaluated automatically according to [1]. Since their geometry is not parameterised and the location of all weld interfaces are not given as input, the partitions can not be found by the code. Flanges are individually rated for certain pressure and temperature ratings and to evaluate them is thus not part of the project scope. Also, due to their irregular geometry, it would be very difficult to automatically identify the regions of criteria. Another thing is that boundary conditions often are applied to flanges, which results in non-physical stress concentrations. Hence, the standard parts pipes, elbows and tees are evaluated automatically by the script, while special blocks and flanges remain unevaluated.

5.2.3 Evaluation algorithm

From Table 3.1, six criteria are identified. Pipes, elbows and tees are all assumed to have fine austenite spacing due to their manufacturing methods and this gives four criteria left. The HISC evaluation is based on an algorithm described below, for every pipe, elbow or tee, it identifies what criterion to use and evaluates the strain value of interest in each integration point.

Step 1: Calculate the strain of interest

As mentioned in Section 3.4.4 of the summary of [1], the total strain that arises from mechanical loads and constraints of applied thermal loads should be considered when evaluating the HISC criteria. This strain of interest is computed by subtracting the strain induced by the applied thermal field, ϵ_{THE} , from the logarithmic total strain, ϵ_{LE} . An ODB file originating from the Flow Loop plug-in contains these strain fields and a new field is calculated:

$$\epsilon_{LE-THE} = \epsilon_{LE} - \epsilon_{THE} \quad (5.1)$$

This new strain field contains all strain components in each integration point.

Step 2: Find maximum principal strain in each integration point

The maximum principal strain, $\epsilon_{\max,LE-THE}$, is calculated from the strain tensor $\underline{\epsilon}_{LE-THE}$. Since C3D8R only have one integration point and is the only element type used for pipes, elbows and tees, there is only one value of $\epsilon_{\max,LE-THE}$ corresponding to each element of the part.

During the automatic meshing of standard parts in the Flow Loop plug-in, two element sets are created as described in Section 5.1.2. One element set contains all elements within 5 % WT from the surface and one contains all elements outside 5 % WT from the surface. It is possible to go through the integration point values of $\epsilon_{\max,LE-THE}$ one element set at the time and this is how the script knows whether the integration point element is within or outside 5 % WT from the surface. The strain values in the surface set are called $\epsilon_{\max,surf}$ while the ones in the internal set are called $\epsilon_{\max,inner}$. These will be evaluated against corresponding criteria.

Step 3: Find location of each element in the part

Each strain value in both $\epsilon_{\max,surf}$ and $\epsilon_{\max,inner}$ have two possible criteria in Table 5.1 to be evaluated against, depending on the location along the part, within or outside L_{res} from a weld. To determine this location the coordinates of each element node corresponding to the strain value is checked in the script. Every element in each of the two element sets are checked, if any element node is located within L_{res} from the weld the corresponding strain value will be evaluated in that region. Hence, for a strain value to be checked against the criteria associated with strains located outside L_{res} from the weld, all element nodes corresponding to that strain value must be located outside L_{res} . This way of determine the location is based on giving a conservative evaluation since the criteria within L_{res} from the weld are more stringent.

The exact procedure will be explained for an elbow with the geometry presented in Figure 5.16. The figure also shows a node, located in an arbitrary position. The center-to-end distance B and angle θ are known, the radius R measured from the imagined center point to the center line is computed as

$$R = \frac{B}{\tan^{-1}(\theta/2)}. \quad (5.2)$$

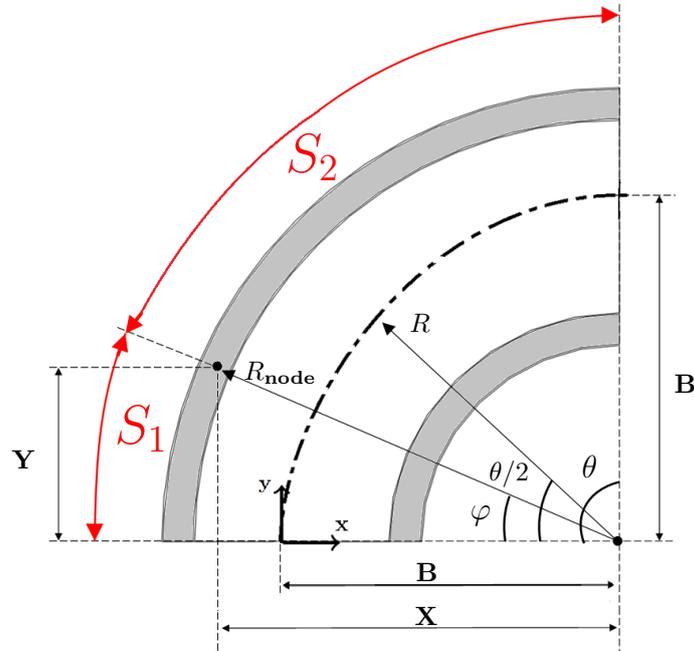


Figure 5.16: Cross section of an example elbow, showing the arc lengths S_1 and S_2 to an arbitrary node.

The position of the arbitrary node is determined by its coordinates in the local $x - y$ system with origin at the

center line, see Figure 5.16. The distances X and Y is therefore determined by

$$X = R - x \quad (5.3)$$

and

$$Y = y. \quad (5.4)$$

Now the radius, R_{node} can be computed as

$$R_{node} = \sqrt{X^2 + Y^2} \quad (5.5)$$

and the angle φ as

$$\varphi = \tan^{-1}(Y/X). \quad (5.6)$$

The arc lengths S_1 and S_2 are measured from the welds to the arbitrary node in the elbow and are now computed as

$$S_1 = R_{node} \cdot \varphi \quad (5.7)$$

and

$$S_2 = R_{node} \cdot (\theta - \varphi). \quad (5.8)$$

The location along the part, within or outside L_{res} from the weld, is then determined by checking these arc lengths:

if $S_1 \leq L_{res}$ **or** $S_2 \leq L_{res} \rightarrow$ Within L_{res} from weld

if $S_1 > L_{res}$ **and** $S_2 > L_{res} \rightarrow$ Outside L_{res} from weld.

In the script, this procedure is made for every element node. If any element node lies within L_{res} from the weld, the strain value corresponding to that element is assumed to lie within that region.

Step 4: Evaluate each strain value against the correct criterion

In the last step, where the criterion of either $\epsilon_{max,surf}$ or $\epsilon_{max,inner}$ has been determined, the value is verified against this criterion, see Table 5.1. All integration points are checked and if any element with corresponding maximum principal strain value does not fulfill the associated criterion, the whole part will fail the HISC evaluation.

Table 5.1: Allowable maximum principal strain.

Strain	Strain criteria	
	Within L_{res} from weld	Outside L_{res} from weld
$\epsilon_{max,inner}$	Minimum of [0.30%; 0.50% - ϵ_{res}]	0.30%
$\epsilon_{max,surf}$	1% - ϵ_{res}	1.00%

The value of the residual strain was set to $\epsilon_{res} = 0.25\%$. The weld regions were not further investigated and therefore the more conservative of the two estimated residual strain values mentioned in Section 3.4.3 of the summary of [1] was picked. This value was used everywhere within L_{res} from the welds. Discussion about this is found in Chapter 7.

5.2.4 Results from the plug-in

The results from the HISC Evaluation plug-in are divided into two separate files. The first one is a copy of the original ODB file with new strain fields of interest added. It may be opened in Abaqus Viewer, a post-processing program, to view the results. The plug-in creates one new output field called LE-THE, it is the same as $\epsilon_{max,LE-THE}$, showing the maximum principal strains. Another new field is called *Not approved elements*, and contains the maximum principal strain values in $\epsilon_{max,LE-THE}$ for all elements that failed the

HISC criteria according to [1]. All other elements will have no value at all. This facilitates for the engineer viewing the results since it gives direct feedback on failed areas in a very distinct way.

A typical result from a Flow Loop simulation is presented in Figure 5.17a, here showing the field $\epsilon_{max,LE-THE}$ when subjected to an internal pressure with magnitude 69 MPa. The same Flow Loop was evaluated with the HISC Evaluation plug-in and the resulting field, *Not approved elements*, is shown in Figure 5.17b. All elements without a strain value, the ones with approved strain values, are plotted in white.

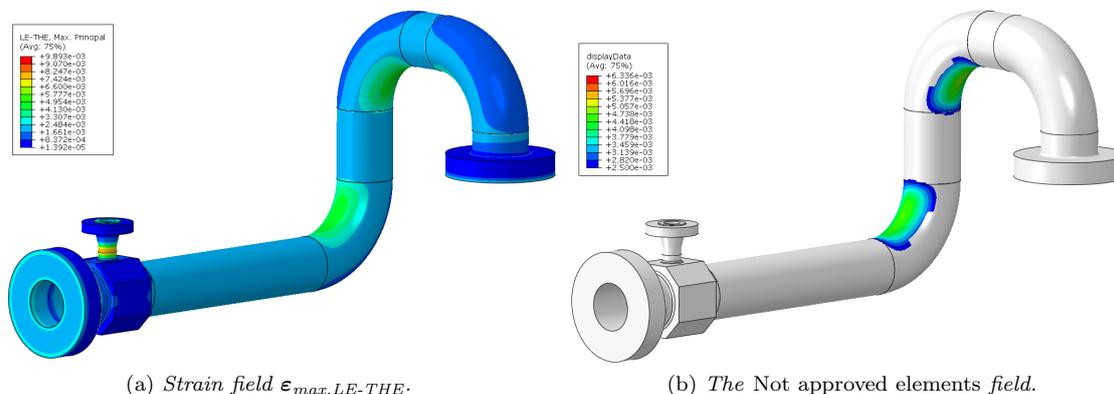


Figure 5.17: *Maximum principal strains in a Flow Loop subjected to internal pressure, one only showing the values in elements failing the HISC criteria.*

A closer view of the area with high strains of one elbow is shown in Figure 5.18a and 5.18b. The influence of the more stringent criteria within L_{res} from the welds is pointed out in Figure 5.18b. Elements with higher strain values are approved, and thus white, outside the weld regions.

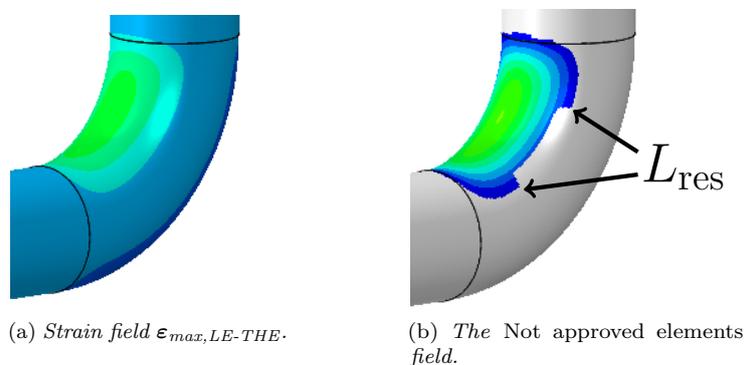


Figure 5.18: *A closer view on one elbow with an area of high maximum principal strains*

The second result file from the plug-in is a report of the HISC Evaluation written as a text file. It provides the user will useful information such as:

- **Analysis information**

- The first pages of the report inform the reader which MDB and Flow Loop model that is analysed and the date the analysis was performed. It includes an explanation about what is done when evaluating the Flow Loop parts against HISC, which criteria that are used and how the element position with corresponding strain value is determined.

- **Approved parts**

- Before going into analysis details, all parts that are approved in the HISC evaluation will be listed with their specified part name. An approved part has no elements with corresponding maximum principal strain values that fails its criteria.

- **Failed parts**

- All parts that failed the HISC evaluation will be listed with their specified part name.

- **HISC evaluation part by part**

- Here the analysis details of every part in the Flow Loop are described. Parts that are excluded in the analysis, flanges and special blocks, are written out by name with a notation that no HISC evaluation is made on these to avoid the engineer reading the report believing that these part are approved. For the parts that are evaluated, the following information is written; L_{res} in millimeter, total number of failed elements, number of failed elements within 5 % WT from surface, number of failed elements outside 5 % WT from surface.

- **Failed elements**

- For each part that fails the HISC evaluation, all element labels are listed. This information could be very useful, especially if only a few elements fail, since elements may be found quickly through this label in Abaqus Viewer.

An example of a HISC Evaluation report is attached in Appendix C, created when analysing the Flow Loop described in Section 6.1.

6 Flow Loop Simulation

In this chapter, a simple example of a Flow Loop model created with the Flow Loop plug-in is described. The results of the model are presented and also results from a HISC evaluation that was conducted using the HISC Evaluation plug-in, described in Section 5.2. A mesh convergence study is carried out to verify that the number of elements through the piping part thickness is sufficient.

6.1 Example of a Flow Loop model

The example Flow Loop consists of two flanges, two elbows and one pipe, as shown in Figure 6.1a. The nominal pipe size and the thickness are 60.3 mm and 8.74 mm, respectively. In Figure 6.1b the geometry tab of the Flow Loop plug-in corresponding to the example Flow Loop is presented.

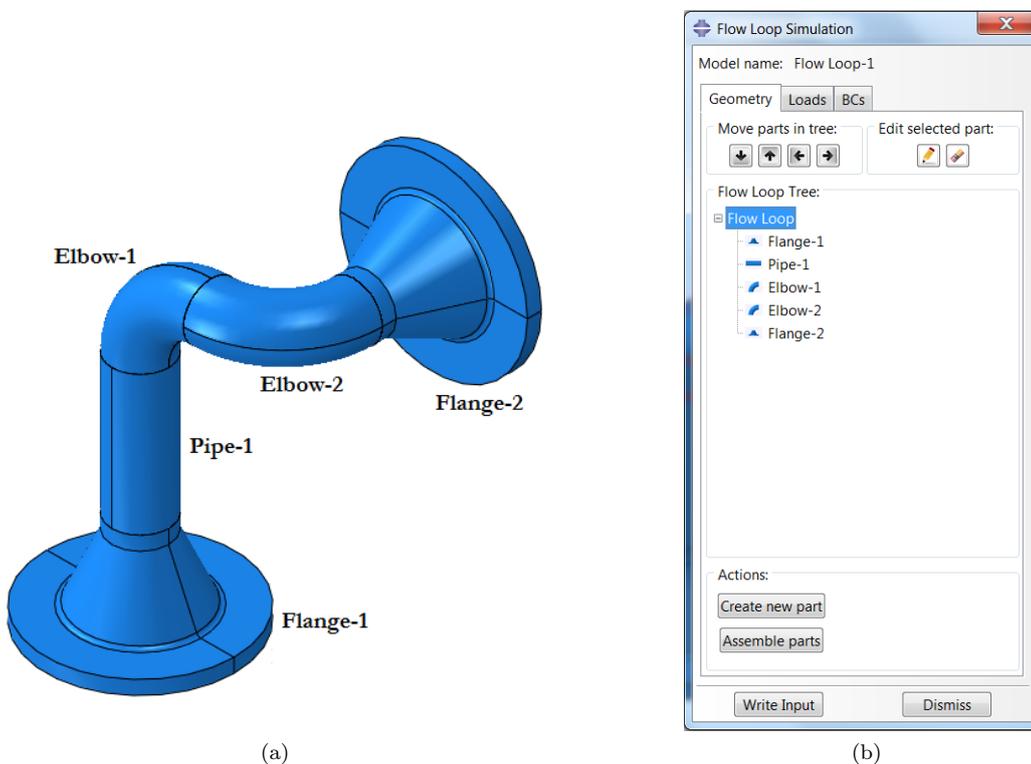


Figure 6.1: The example Flow Loop (a) with its Flow Loop tree (b).

The loads and BCs of the example model are all created with the Flow Loop plug-in. In Table 6.1, these are presented with their corresponding values. The displacement vector of a flange BC is represented by \mathbf{u} . In this model, two load steps in excess of the initial step were created. For details about the loads and BCs, see Section 5.1 about the Flow Loop plug-in. In Appendix B, a full description of the model is presented. This description originates from the Model report file that is automatically generated by the Flow Loop plug-in.

Table 6.1: Loads and boundary conditions of the example Flow Loop.

Type	Initial step	Step-1	Step-2
Flange BC, Flange-1	$\mathbf{u} = \mathbf{0}$	$\mathbf{u} = \mathbf{0}$	$\mathbf{u} = \mathbf{0}$
Flange BC, Flange-2	$\mathbf{u} = \mathbf{0}$	$\mathbf{u} = [-1 \ -1 \ 0] \text{ mm}$	$\mathbf{u} = [-1 \ -1 \ 0] \text{ mm}$
Internal pressure	0	0	85 MPa
Temperature	18°C	18°C	121°C

6.2 HISC analysis of example Flow Loop

As described in Section 5.2.3, the logarithmic strain subtracted by the applied thermal strain is considered when evaluating HISC criteria. A contour plot of the maximum principal value of this strain, $\epsilon_{\max,LE-THE}$, from the solution of the example Flow Loop model is shown in Figure 6.2.

The results from the example Flow Loop model was run in the HISC Evaluation plug-in and the design turned out to fail the HISC criteria. Part by part, Pipe-1 was approved whereas both Elbow-1 and Elbow-2 failed. Flanges are not included in the HISC evaluation.

The highest, not approved strain value was found in the failed part Elbow-2. As described in Section 5.2.2, the parts to be evaluated against HISC criteria are subdivided into regions within 5 % wall thickness from the material surface and an internal region outside these surface regions. This is due to different criteria of approval between the different regions. No elements failed in the surface regions, only in the internal region.

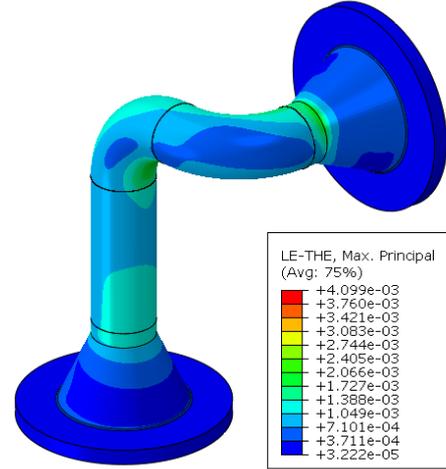


Figure 6.2: A contour plot of $\epsilon_{\max,LE-THE}$ in the example Flow Loop.

In Figure 6.3, the plot of not approved elements that was created by the HISC Evaluation plug-in is presented in three subfigures. Figure 6.3a only shows approved, white elements while in Figure 6.3b some surface region elements are removed to show some of the coloured, not approved elements in the internal region. In the last subfigure, Figure 6.3c, a closer look is taken at these not approved elements. As a matter of fact, the highest strain values were in the surface region but because of the less stringent criterion there, compared to the internal region, these values still passed the evaluation.

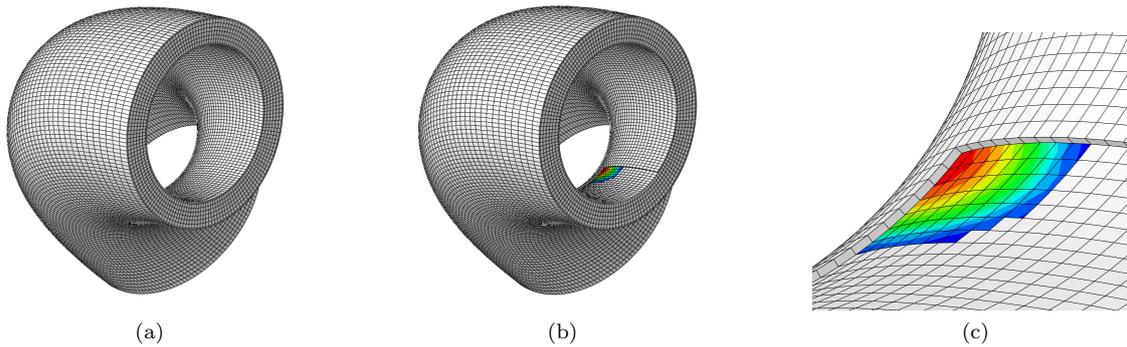


Figure 6.3: A visualisation of not approved elements in Elbow-2. White elements are approved, coloured ones are not. In (a), all elements of Elbow-2 are shown while some of the surface region elements are hidden in (b). A close-up of this area of interest is presented in (c).

Figure 6.4 shows Elbow-2, cut at the cross section where this maximum $\epsilon_{\max,LE-THE}$ was found. Figure 6.4a presents a contour plot of $\epsilon_{\max,LE-THE}$ whereas Figure 6.4b shows a plot of not approved elements in Elbow-2. In the latter figure, some elements are marked in black. This is the most strained through-thickness element row in Elbow-2, considered in the mesh convergence study below.

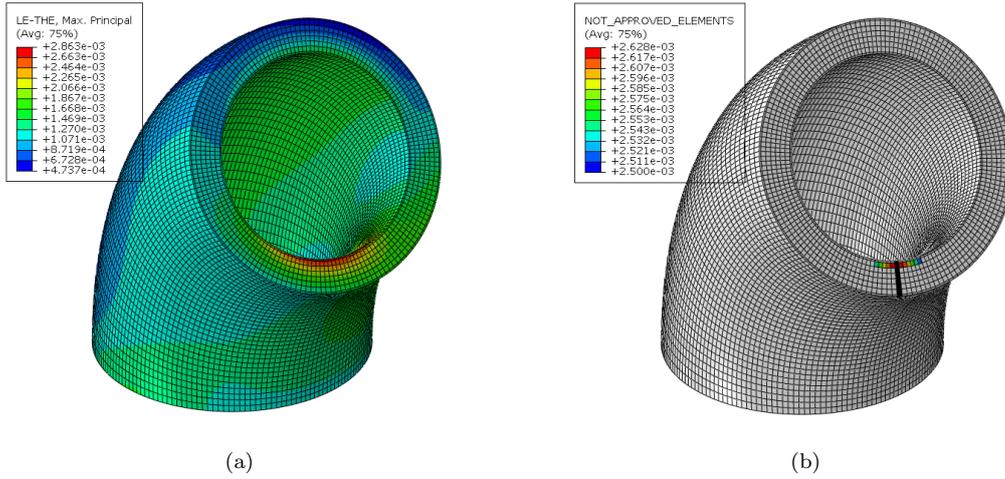


Figure 6.4: Elbow-2, cut in the cross section of highest deformation. A contour plot of $\varepsilon_{\max,LE-THE}$ is shown in (a) and not approved elements in (b). Note in (b) that some elements are marked in black. These are the elements where strain values are taken for the mesh convergence study.

6.3 Mesh Convergence

To verify that the default value, 8, of number of elements through the thickness of piping parts is sufficient, a mesh convergence study was conducted. The meshes of Pipe-1, Elbow-1 and Elbow-2 were refined by modifying the mesh parameter, described in Section 5.1.2, of these parts with the Flow Loop plug-in. The strain value of interest, $\varepsilon_{\max,LE-THE}$, was then extracted from the integration points of the elements through the thickness at the most strained location in Elbow-2. These elements are marked in black in Figure 6.4b and the strain values are in Figure 6.5 plotted against the distance from the inner surface to the corresponding integration points. The strain distribution is plotted for three different meshes and it may be noted that the different curves are more or less identical to each other.

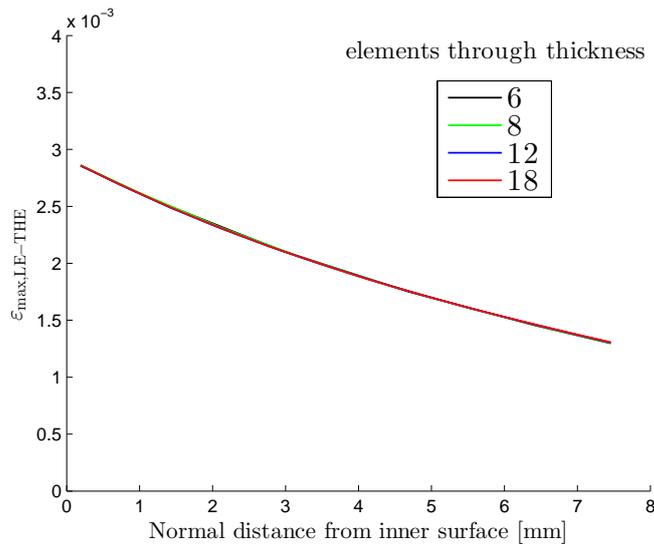


Figure 6.5: Distribution of $\varepsilon_{\max,LE-THE}$ along the thickness of Elbow-2 at the most deformed region for different meshes. Note that the different curves are not separable in this plot.

Because of the surface partitions described in Section 5.1.2, the elements closest to the surface of a standard part are of the same thickness for up to 18 through-thickness elements. Hence, the strain value in the integration point of the most strained element at the inner surface may be directly compared between the different meshes since they are taken in the same location. This is done for six different meshes in Figure 6.6 and noting that the scale on the vertical axis is narrow, it may be observed that this maximum $\varepsilon_{\max,LE-THE}$ is more or less constant.

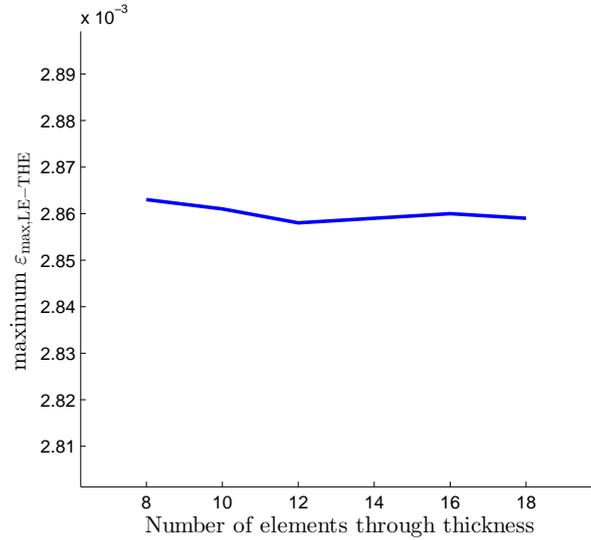
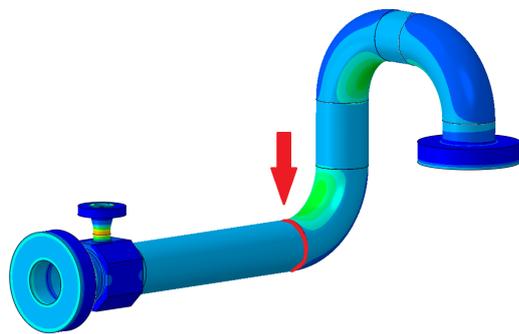


Figure 6.6: Maximum value of $\varepsilon_{\max,LE-THE}$ in the example Flow Loop for different meshes. Note the scale of the vertical axis, the curve in principal represents a constant value.

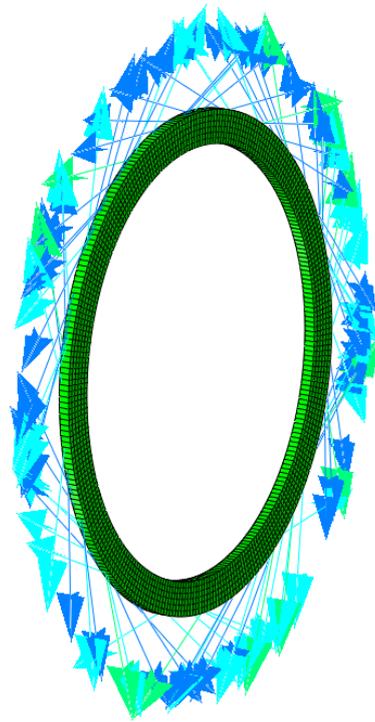
6.4 Principal strain directions

An investigation was made of a Flow Loop model subjected to internal pressure and temperature load and with both flanges at the ends fixed with a zero displacement Flange BC. The Flow Loop is shown in Figure 6.7a, also showing a red arrow pointing at a slice of piping at the location of a girth weld between a pipe and an elbow. This slice is cut out and the directions of the maximum principal strains in $\varepsilon_{\max,LE-THE}$ are plotted as arrows in Figure 6.7b. The arrows show that the principal strain directions are in the same direction as the hoop direction.

Another investigation was made of the Flow Loop model described previously in this chapter, see Section 6.1, with loads and boundary conditions as described in Table 6.1. A slice at the girth weld between Elbow-1 and Elbow-2 is marked in red in Figure 6.8a. Note that this Flow Loop model includes a non-zero displacement Flange BC on one of the flanges. The resulting maximum principal strain directions were plotted and are presented in Figure 6.8b. The result shows that the principal strains are not directed in the hoop direction at this girth weld.

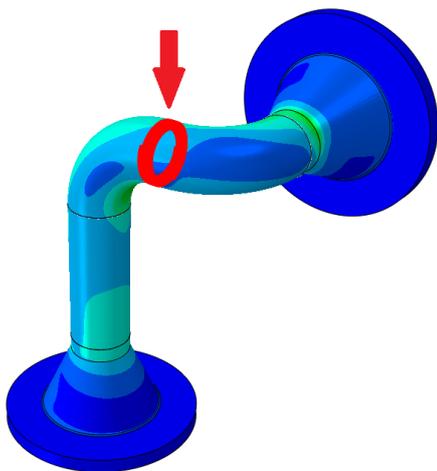


(a) The arrow is pointing at the investigated girth weld.

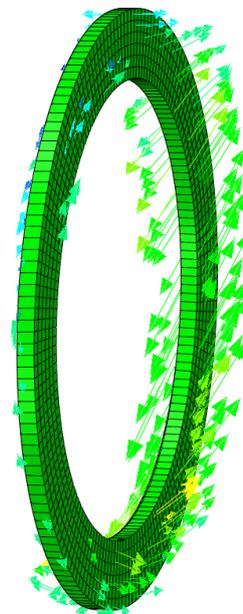


(b) Principal strain directions at the girth weld.

Figure 6.7: Maximum principal strains, $\varepsilon_{\max,LE-THE}$, of a Flow Loop.



(a) The arrow is pointing at the investigated girth weld.



(b) Principal strain directions at the girth weld.

Figure 6.8: Maximum principal strains, $\varepsilon_{\max,LE-THE}$, of a Flow Loop where one of the flanges are subjected to non-zero displacement.

7 Concluding remarks and discussion

The goal of the project described in previous chapters was to reduce the time needed to analyse Flow Loops with respect to HISC by automating simulations. It was also desired to document the work well and efficiently.

In the end, the project resulted in two plug-ins for Abaqus CAE. These plug-ins were created by means of Python scripting and make it possible to build Flow Loop models and evaluate their output against HISC RP very rapidly. The plug-ins also leave trace through writing report files automatically. Hence, altogether, the project must be considered successful.

In excess of reducing the time consumption, the automation also ensures quality of the analyses through minimising the risk of user errors when modelling and standardising the analysis procedure. Parameters specified are often taken from standards and will then most likely be accurate. A thoroughly worked through interpretation of the HISC RP is now in the backbone of the plug-ins which will from now on align verifications of Flow Loops and make them less dependent on the engineer conducting the simulation.

7.1 The Flow Loop plug-in

In the first of the two plug-ins, the Flow Loop plug-in, finite element models are built with three-dimensional solid elements, predominantly of the type C3D8R. This element type gives the models accurate stiffness but also induces a risk of so called hourglassing which is a non-physical phenomenon in finite element modelling. Hourglassing, however, is not an issue if the mesh resolution is sufficiently high which it has been proven to be in the models created with the Flow Loop plug-in. The choice of using C3D8R is thus motivated.

The material definitions and the FE modelling in the Flow Loop plug-in must be considered to follow the requirements of the HISC RP as long as special blocks that are imported are modelled correctly.

The Flow Loop plug-in allows the user to rapidly apply the most common BC and load types to Flow Loop models. There are, however, other types that are not supported by the plug-in but are present in the real applications and also mentioned in the HISC RP, an example is tie-in loads. It would be possible to add these as functionality to the plug-in. That it was not added during the project was just a matter of that a limited amount of time was available. It should also be emphasised that any BCs and loads may be added to a Flow Loop model outside the plug-in while still using it to build all other parts of the model.

7.2 The HISC Evaluation plug-in

Before this project was carried out, a common way of verifying Flow Loops with respect to HISC has been to manually evaluate the maximum strain value in the Flow Loop model against the most stringent of these criteria. Since there are different strain criteria of approval in different regions of a part according to the RP, this approach is over conservative.

With the second plug-in, the HISC Evaluation plug-in, the evaluation is done with different criteria depending on whether the strain value is taken close to a material surface or a weld or both or neither. This gives a less stringent but more accurate HISC evaluation of a Flow Loop. Up to this point, there have been problems with getting Flow Loops passed the HISC verification according to experienced engineers. If this problem, e.g., was due to that the criteria close to a material surface was never considered, the HISC Evaluation plug-in may have solved it. These criteria are namely much less stringent than for the interior material.

The strain values that are evaluated against criteria are taken in the integration points of the elements in the evaluated model. The integration points vary in number with element type and are located in the interior of the elements. In particular, a single one is located in the mid-point of the element type C3D8R, as described in Section 4.4.1. Since the parts that are evaluated in the HISC Evaluation plug-in are built by this element type, this means that the strain is considered constant throughout each element.

This is clarified in Figure 7.1 where the integration point values of $\varepsilon_{\max,LE-THE}$ are marked by red circles. The red line drawn between these values represents the computed strain distribution along the thickness of an

elbow, taken from the example Flow Loop in Chapter 6. In contrast to this curve, the blue line shows the distribution that is actually considered by the HISC Evaluation plug-in. The discontinuous jumps on the blue curve are located at the element boundaries. To separate the surface regions from the interior region of the thickness where different strain criteria apply, dashed black lines have been drawn.

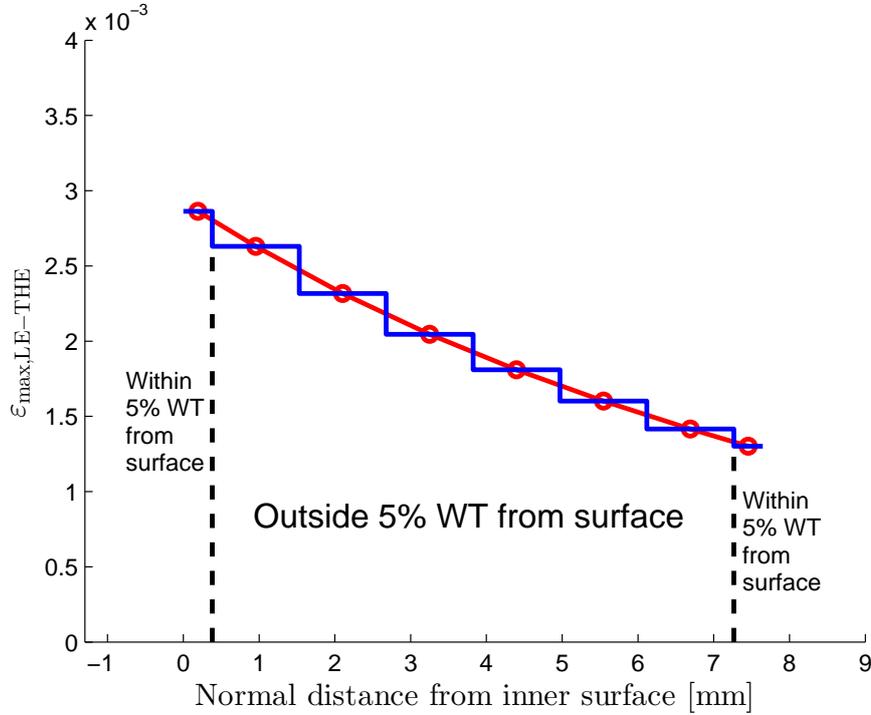


Figure 7.1: *Distribution of $\varepsilon_{\max,LE-THE}$ along the thickness of an elbow. In red, a curve is drawn between the integration point values which are shown as red circles. The piecewise constant blue curve represents the strain distribution assuming that the strain is constant throughout each element. The boundaries of regions with different HISC criteria are shown as dashed black lines.*

Comparing the red and the blue curves in Figure 7.1, one may see how the deformation often is higher at the boundaries of an element than in the integration point. Hence, the strain value at some location in a certain element may be higher than the one evaluated against the HISC criterion in the region of that element. Consequently, an element like this and in turn a Flow Loop might pass the HISC evaluation when it should fail. The same problem occurs in the along direction of a piping part.

A higher mesh resolution would give integration points closer to the region boundaries, but since it has been shown that the default value of 8 elements through the thickness is sufficient to obtain an accurate strain distribution, increasing the number of elements is not recommended. Instead, the HISC evaluation algorithm could be improved by implementing interpolation between and extrapolation from the integration points to find the highest values of strain within each region of the HISC criteria.

Another thing that has not been treated in this project is to neglect residual strain in the hoop direction of girth welds, a possibility that is given in the HISC recommended practice, described in Section 3.4.3. The results in Section 6.4 showed that for a Flow Loop only subjected to internal pressure and temperature loads, the maximum principal strains were parallel to the hoop direction. The second investigation, however, showed that this is not always the case. When a Flow Loop's ends are assumed to be subjected to misalignment from e.g. installation, the principal strain directions at the welds will not be in the hoop direction and the residual strains can not be neglected. The conclusion is that the HISC Evaluation plug-in will sometimes be over conservative since the highest estimated value of the longitudinal strain, $\varepsilon_{res} = 0.25\%$, in the region between the weld and L_{res} always is used.

The HISC Evaluation plug-in is very useful for visualisation. It clearly shows which elements that fail or pass the HISC criteria. The engineer using the plug-in can now get a great overview with the same resolution as the mesh size, see e.g. Figure 5.18b. This not only saves time, but also effectively gives hints on how to improve the design of the Flow Loop to get it passed the HISC verification.

All parts evaluated by the HISC plug-in were assumed to have a micro-structure with fine austenite spacing, due to the methods used for manufacturing these parts. As mentioned, materials with fine austenite spacing have a greater resistance towards HISC. Hence, higher maximum principal strain values are allowed, as confirmed by the criteria in Table 3.1. It is important that the engineer using the plug-in is aware of this, in case other manufacturing methods are used, leading to a micro-structure with coarse austenite spacing instead. As an example, flanges are often manufactured by forging which is not listed as a method leading to fine austenite spacing.

7.3 Future work

As a continuation of this project, the following work is suggested to improve and further develop the methods and the plug-ins.

- To include the effect of principal strain direction at welds. The angle between this direction and the direction of ε_{res} perpendicular to the weld could be used to modify the HISC RP strain criteria to be less conservative.
- To improve the HISC evaluation algorithm by implementing interpolation and extrapolation of strain values between and from the integration points.
- To add functionality to the Flow Loop plug-in so that all BCs and loads of Flow Loop models are included.
- To extend the analysis tools to include more requirements than just the HISC standard, such as against top loads and fatigue.
- To automate steps in the post-processing, such as plotting for reports.

References

- [1] DET NORSKE VERITAS. Design of duplex stainless steel subsea equipment exposed to cathodic protection. *Recommended Practice* (2008).
- [2] J. M. Schicks. Gas hydrates. *Annu. Rep. Prog. Chem., Sect. C: Phys. Chem.* **106** (0 2010), 101–117. DOI: 10.1039/B811059J. URL: <http://dx.doi.org/10.1039/B811059J>.
- [3] ASME B36.10M. Welded and Seamless Wrought Steel Pipe. *The American Society of Mechanical Engineers* (2004).
- [4] M. L. Lennart Hågeryd Stefan B. *Modern produktionsteknik Del 1*. Swedish. Stockholm: Liber, 2010. ISBN: 9789147050918.
- [5] ASME B16.9. Factory-Made Wrought Butt Welding Fittings. *The American Society of Mechanical Engineers* (2007).
- [6] *Benkan, Manufacturing of Butt Weld Fittings*. URL: <http://www.benkan-japan.com/en/index.html>.
- [7] ISO 10423. Petroleum and natural gas industries - Drilling and production equipment - Wellhead and christmas tree equipment. *INTERNATIONAL STANDARD* (2009).
- [8] Y. Li et al. Numerical study on girth weld of marine steel tubular piles. *Applied Ocean Research* **44.0** (2014), 112 –118. ISSN: 0141-1187. URL: <http://www.sciencedirect.com/science/article/pii/S0141118713001016>.
- [9] H.D.Solomon. Stainless Steels - Duplex. *Encyclopedia of Materials: Science and Technology (Second Edition)* (2001).
- [10] G Chai et al. Mechanisms of Hydrogen Induced Stress Crack Initiation and Propagation in Super Duplex Stainless Steels. *steel research int.*, **80** (2009), 482 –487.
- [11] H. Mattsson et al. *Materialteknik - Korrosion*. Department of Materials and Manufacturing Technology, Chalmers University of Technology, 2006.
- [12] W. Callister. *Materials science and engineering, an introduction*. John Wiley & Sons, Inc., 2007.
- [13] ASME VIII. Rules for Construction of Pressure Vessels. *The American Society of Mechanical Engineers* (2004).
- [14] DET NORSKE VERITAS. Submarine Pipeline Systems. *Offshore Standard* (2012).
- [15] ASME II, part D, Properties (Metric). Materials. *The American Society of Mechanical Engineers* (2010).
- [16] Simulia. *ABAQUS Documentation - Abaqus Analysis User's Guide, Mesh tie constraints*. Dassault Systèmes, 2013.
- [17] Simulia. *ABAQUS Documentation - Abaqus Analysis User's Guide, Coupling constraints*. Dassault Systèmes, 2013.
- [18] N. Saabye Ottosen and H. Petersson. *Introduction to the finite element method*. English. New York: Prentice Hall, 1992. ISBN: 0134738772, 9780134738772.
- [19] Simulia. *ABAQUS Documentation - Getting Started with Abaqus*. Dassault Systèmes, 2013.
- [20] Simulia. *ABAQUS Documentation - Abaqus Analysis User's Guide, Solid elements*. Dassault Systèmes, 2013.
- [21] Simulia. *ABAQUS Documentation - Abaqus Theory Guide, Element Library: overview*. Dassault Systèmes, 2013.
- [22] *The official home of the Python Programming Language*. URL: www.python.org.
- [23] Simulia. *ABAQUS Documentation - Abaqus/CAE User's Guide, Verifying your mesh*. Dassault Systèmes, 2013.

Appendix A - Complex Flow Loop with Flow Loop Tree

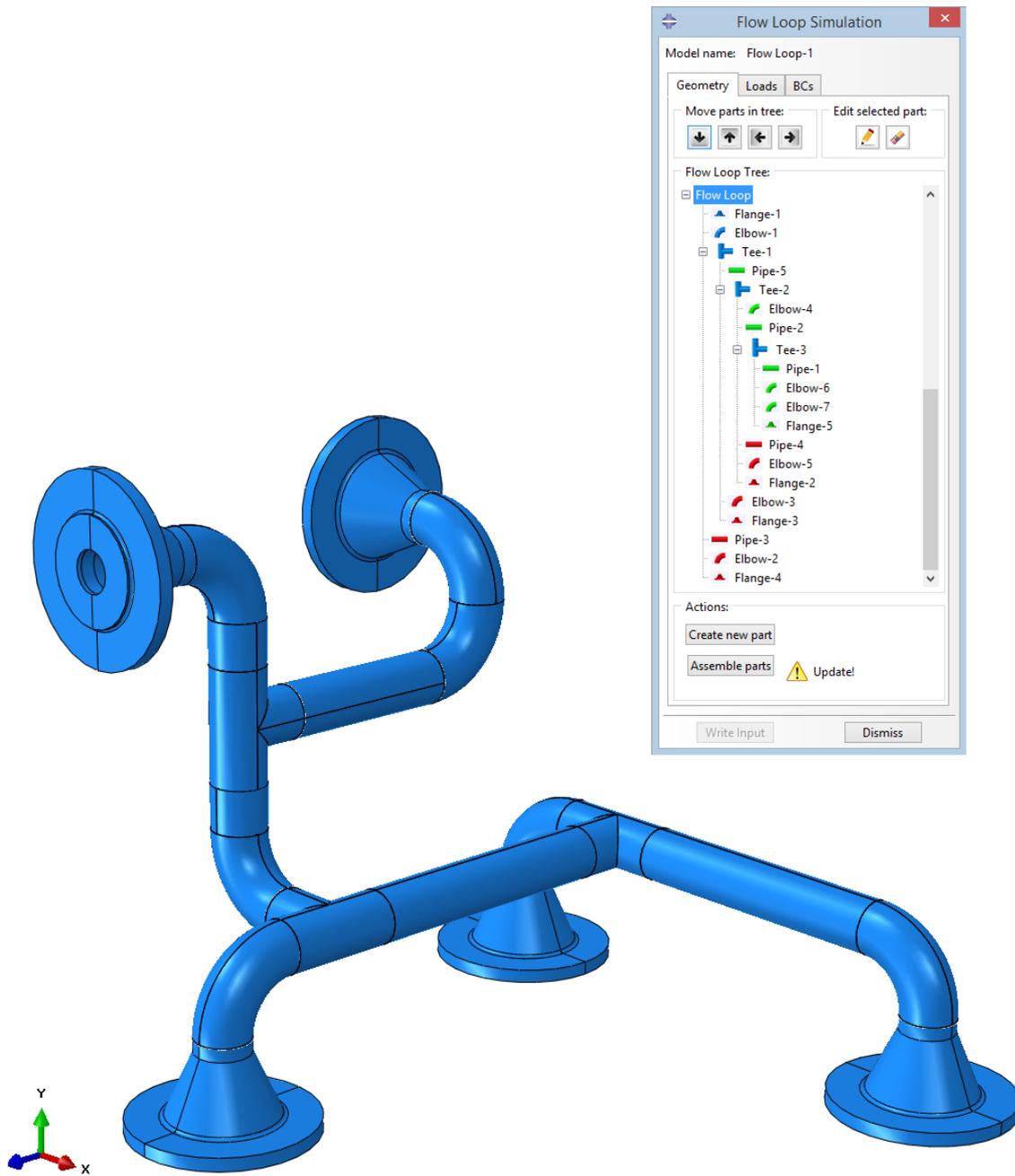


Figure 7.2: Complex Flow Loop with corresponding Flow Loop Tree, created with the Flow Loop plug-in.

Appendix B - model report

```
*****
*
*                               *****
*                               * FLOW LOOP ANALYSIS - REPORT *
*                               *****
*
*                               PROJECT FOLDER:
*
*                               C:/Examensarbete/Abaqus/FlowLoopAnalysis/numEl_8
*
*                               Date: 2014-05-07
*
*                               Copyright (C) 2014 David Andersson, Marcus Gustafsson
*
*****
```

```
*****
***** FLOW LOOP - INPUT DATA *****
*****
```

```
***** PIPES *****
```

```
** Pipe: Pipe-1
```

```
*USER DEFINED INPUT
name..... Pipe-1
outer diameter..... 60.3
thickness..... 8.74
length..... 150.0
erosion..... 0.0
corrosion (inside).. 0.0
corrosion (outside).. 0.0
mill tolerance..... 0.125
material..... UNS_S32760_25Cr_SuperDuplex
```

```
*CALCULATED INPUT SENT TO *.inp-FILE
outer diameter..... 60.3
inner diameter..... 45.005
```

```
***** ELBOWS *****
```

```
** Elbow: Elbow-1
```

```
*USER DEFINED INPUT
name..... Elbow-1
```

```

outer diameter..... 60.3
thickness..... 8.74
angle..... 90.0
center-to-end (AorB) 76.0
erosion..... 0.0
corrosion (inside).. 0.0
corrosion (outside). 0.0
mill tolerance..... 0.125
material..... UNS_S32760_25Cr_SuperDuplex

```

```

*CALCULATED INPUT SENT TO *.inp-FILE
outer diameter..... 60.3
inner diameter..... 45.005

```

** Elbow: Elbow-2

```

*USER DEFINED INPUT
name..... Elbow-2
outer diameter..... 60.3
thickness..... 8.74
angle..... 90.0
center-to-end (AorB) 76.0
erosion..... 0.0
corrosion (inside).. 0.0
corrosion (outside). 0.0
mill tolerance..... 0.125
material..... UNS_S32760_25Cr_SuperDuplex

```

```

*CALCULATED INPUT SENT TO *.inp-FILE
outer diameter..... 60.3
inner diameter..... 45.005

```

***** FLANGES *****

** Flange: Flange-1

```

*USER DEFINED INPUT
name..... Flange-1
OD..... 200.0
B..... 42.82
T..... 20.0
T2..... 15.0
K..... 120.0
Q..... 5.0
J1..... 120.0
J2..... 60.3
J3..... 70.0
r1..... 5.0
r2..... 3.0
erosion..... 0.0
corrosion (inside).. 0.0

```

corrosion (outside). 0.0
mill tolerance..... 0.125
flip..... False
material..... UNS_S32760_25Cr_SuperDuplex

*CALCULATED INPUT SENT TO *.inp-FILE
outer diameter..... 60.3 (at weldneck)
inner diameter..... 45.005

** Flange: Flange-2

*USER DEFINED INPUT
name..... Flange-2
OD..... 200.0
B..... 42.82
T..... 20.0
T2..... 15.0
K..... 120.0
Q..... 5.0
J1..... 120.0
J2..... 60.3
J3..... 70.0
r1..... 5.0
r2..... 3.0
erosion..... 0.0
corrosion (inside).. 0.0
corrosion (outside). 0.0
mill tolerance..... 0.125
flip..... True
material..... UNS_S32760_25Cr_SuperDuplex

*CALCULATED INPUT SENT TO *.inp-FILE
outer diameter..... 60.3 (at weldneck)
inner diameter..... 45.005

***** TEES *****

***** SPECIAL BLOCKS *****

***** FLOW LOOP - CONSTRAINTS *****

***** TIE CONSTRAINTS *****

** Tie: tieElbow-1

name..... tieElbow-1

interaction pair....	Pipe-1 / Elbow-1
master surface.....	outInteractionSurface (Pipe-1)
slave surface.....	inInteractionSurface (Elbow-1)
position tolerance..	COMPUTED

** Tie: tieElbow-2

name.....	tieElbow-2
interaction pair....	Elbow-1 / Elbow-2
master surface.....	outInteractionSurface (Elbow-1)
slave surface.....	inInteractionSurface (Elbow-2)
position tolerance..	COMPUTED

** Tie: tieFlange-2

name.....	tieFlange-2
interaction pair....	Elbow-2 / Flange-2
master surface.....	outInteractionSurface (Elbow-2)
slave surface.....	inInteractionSurface (Flange-2)
position tolerance..	COMPUTED

** Tie: tiePipe-1

name.....	tiePipe-1
interaction pair....	Flange-1 / Pipe-1
master surface.....	outInteractionSurface (Flange-1)
slave surface.....	inInteractionSurface (Pipe-1)
position tolerance..	COMPUTED

***** KINEMATIC COUPLINGS *****

** Coupling: kinematicCoupling_Flange-1

name.....	kinematicCoupling_Flange-1
coupling type.....	KINEMATIC
control point.....	BCPoint (Flange-1)
surface.....	BCSurface (Flange-1)
coupled DOFs.....	ALL

** Coupling: kinematicCoupling_Flange-2

name.....	kinematicCoupling_Flange-2
coupling type.....	KINEMATIC
control point.....	BCPoint (Flange-2)
surface.....	BCSurface (Flange-2)

coupled DOFs..... ALL

***** FLOW LOOP - MESH *****

** Part: Elbow-1

name.....	Elbow-1
part type.....	Elbow
element types.....	Linear HEX C3D8R (if created by the Flow Loop Plugin)
number of elem.....	70848
warning elements....	[]
failed elements.....	[]
average aspect ratio	3.16567230225
worst aspect ratio..	8.088941
largest corner angle	93.3809509277
smallest corner angle	93.380951

** Part: Elbow-2

name.....	Elbow-2
part type.....	Elbow
element types.....	Linear HEX C3D8R (if created by the Flow Loop Plugin)
number of elem.....	70848
warning elements....	[]
failed elements.....	[]
average aspect ratio	3.16567230225
worst aspect ratio..	8.088941
largest corner angle	93.3809509277
smallest corner angle	93.380951

** Part: Pipe-1

name.....	Pipe-1
part type.....	Pipe
element types.....	Linear HEX C3D8R (if created by the Flow Loop Plugin)
number of elem.....	64288
warning elements....	[]
failed elements.....	[]
average aspect ratio	4.34221124649
worst aspect ratio..	8.005868
largest corner angle	91.0977478027
smallest corner angle	91.097748

** Part: Flange-1

```

name..... Flange-1
part type..... Flange
element types..... Linear HEX C3D8R, Linear Wedge C3D6 (if created by
the Flow Loop Plugin)

number of elem..... 43708
warning elements.... []
failed elements.... []
average aspect ratio 2.64728069305
worst aspect ratio.. 8.191407
largest corner angle 141.206192017
smallest corner angle 141.206192

```

** Part: Flange-2

```

name..... Flange-2
part type..... Flange
element types..... Linear HEX C3D8R, Linear Wedge C3D6 (if created by
the Flow Loop Plugin)

number of elem..... 43708
warning elements.... []
failed elements.... []
average aspect ratio 2.64645910263
worst aspect ratio.. 8.191407
largest corner angle 141.206192017
smallest corner angle 141.206192

```

```

*****
***** FLOW LOOP - LOADS *****
*****
***** INTERNAL PRESSURE LOAD *****

```

** Loading step: Step-1

```

internal pressure... 0.0

```

** Loading step: Step-2

```

internal pressure... 85.0

```

```

***** TEMPERATURE LOAD *****

```

** Step: Initial

initial temperature. 18.0

** Step: Step-1

load temperature.... 18.0

** Step: Step-2

load temperature.... 121.0

***** FLOW LOOP - BOUNDARY CONDITIONS *****

***** DISPLACEMENT BOUNDARY CONDITIONS *****
***** CREATED BY THE FLOW LOOP PLUGIN *****

** Boundary condition: flangeBC_Flange-1

name.....	flangeBC_Flange-1
type.....	DisplacementBC
part.....	Flange-1
point/surface (part)	BCPoint (Flange-1)

* Initial Step

u1.....	0.0
u2.....	0.0
u3.....	0.0

* Loading step: Step-1

u1.....	0.0
u2.....	0.0
u3.....	0.0

* Loading step: Step-2

u1.....	0.0
u2.....	0.0
u3.....	0.0

** Boundary condition: flangeBC_Flange-2

name.....	flangeBC_Flange-2
type.....	DisplacementBC
part.....	Flange-2
point/surface (part)	BCPoint (Flange-2)

* Initial Step

u1.....	0.0
u2.....	0.0
u3.....	0.0

* Loading step: Step-1

u1.....	-1.0
u2.....	0.0
u3.....	-1.0

* Loading step: Step-2

u1.....	-1.0
u2.....	0.0
u3.....	-1.0

***** END CAP FORCES *****

Appendix C - HISC evaluation report

```

*****
*
*                               *****
*                               * FLOW LOOP HISC ANALYSIS *
*                               *****
*                               'LONG VERSION'
*
*                               PROJECT FILES:
*
*                               C:\Examensarbete\Abaqus\FlowLoopAnalysis\numEl_8\FlowLoop-1.cae
*                               C:\Examensarbete\Abaqus\FlowLoopAnalysis\numEl_8\FlowLoop-1_HISC.odb
*
*                               Date: 2014-05-07
*
*                               Copyright (C) 2014 David Andersson, Marcus Gustafsson
*
*****

```

```

-----
|
| In this report a HISC analysis is made according to DNV-RP-F112,
| section E, Non-Linear strain criteria. The analysis is made on
| a flow loop created by the Abaqus plugin named Flow Loop Simulation.
|
| The residual strain for the weld (Table C1):
|     eps_res = 0.25 %
|
| The distance LRes is presented below for each part.
|
| The analysis is based on four different criteria (Table E-1):
|
|                               Strain Criteria
|
|                               -----
|                               | Within LRes | Outside LRes | |
|---|---|---|---|
| | Outside 5% WT | 0.5%-eps_res | 0.3% |
| | from surface | = 0.25%      |      |
|                               -----|-----|
| | Within 5% WT | 1% -eps_res  | 1%   |
| | from surface | = 0.75%     |      |
|                               -----|-----|
|
| The strain used to check the criteria is LE-THE. The strain is
| evaluated in each element and if any of the element nodes are
| located inside one of the partitions corresponding to a stricter
| criteria, that criterion is used. If the strain value in an
| element is above the limit the actual part will fail the HISC
| analysis. The results are presented below for each part, every
| failed element label is also printed in the report.
|
| NOTE! Not all part types are analyzed, see report below.
|
-----

```

 ***** FLOW LOOP, HISC ANALYSIS - SUMMARY *****

***** APPROVED PARTS *****

 ** PARTS THAT FULFILLS THE HISC CRITERIA:

part name..... Pipe-1

***** FAILED PARTS *****

 ** PARTS THAT FAILS THE HISC CRITERIA:

part name..... Elbow-1
 part name..... Elbow-2

 ***** FLOW LOOP - HISC ANALYSIS PART BY PART *****

***** PIPES *****

 ** Pipe: Pipe-1

status..... OK, fulfills HISC criteria
 LRes (DNV RP F112)..... 40.5825547495

***** ELBOWS *****

 ** Pipe: Elbow-1

status..... FAIL, not approved according to HISC
 criteria
 LRes (DNV RP F112)..... 40.5825547495
 num. of fail elements..... 10
 num. fail-el. within 5% WT... 0
 num. fail-el. outside 5% WT... 10
 failed element labels..... see further down in this report

 ** Pipe: Elbow-2

```

status..... FAIL, not approved according to HISC
                criteria
LRes (DNV RP F112)..... 40.5825547495
num. of fail elements..... 365
num. fail-el. within 5% WT.... 0
num. fail-el. outside 5% WT... 365
failed element labels..... see further down in this report

```

***** TEES *****

***** FLANGES *****

HISC analysis is not performed on flanges!

***** BLOCKS *****

NOT IMPLEMENTED YET

***** FLOW LOOP - FAILED ELEMENTS *****

** Part: Elbow-1

* Failed element labels outside 5% of wall thickness:

```

51197    51198    51690    51691    51692    51693    52182    52183
52184    52675    -----

```

** Part: Elbow-2

* Failed element labels outside 5% of wall thickness:

```

14350    14842    15334    15826    16318    16809    16810    17301
17302    17793    17794    18285    18286    18777    18778    19269
19270    19761    19762    20253    20254    20745    20746    21236
21237    21238    21728    21729    21730    22220    22221    22222
22712    22713    22714    23204    23205    23206    23696    23697
23698    24188    24189    24190    24680    24681    24682    25172
25173    25174    25664    25665    25666    26156    26157    26158
26649    26650    27141    27142    27634    28126    40350    40351

```

40352	40353	40354	40841	40842	40843	40844	40845
40846	40847	41332	41333	41334	41335	41336	41337
41338	41339	41340	41823	41824	41825	41826	41827
41828	41829	41830	41831	41832	42315	42316	42317
42318	42319	42320	42321	42322	42323	42324	42806
42807	42808	42809	42810	42811	42812	42813	42814
42815	42816	43298	43299	43300	43301	43302	43303
43304	43305	43306	43307	43308	43789	43790	43791
43792	43793	43794	43795	43796	43797	43798	43799
43800	44281	44282	44283	44284	44285	44286	44287
44288	44289	44290	44291	44292	44773	44774	44775
44776	44777	44778	44779	44780	44781	44782	44783
45265	45266	45267	45268	45269	45270	45271	45272
45273	45274	45275	45757	45758	45759	45760	45761
45762	45763	45764	45765	45766	45767	46249	46250
46251	46252	46253	46254	46255	46256	46257	46258
46259	46741	46742	46743	46744	46745	46746	46747
46748	46749	46750	47233	47234	47235	47236	47237
47238	47239	47240	47241	47242	47725	47726	47727
47728	47729	47730	47731	47732	47733	47734	48217
48218	48219	48220	48221	48222	48223	48224	48225
48709	48710	48711	48712	48713	48714	48715	48716
48717	49201	49202	49203	49204	49205	49206	49207
49208	49693	49694	49695	49696	49697	49698	49699
49700	50185	50186	50187	50188	50189	50190	50191
50677	50678	50679	50680	50681	50682	50683	51169
51170	51171	51172	51173	51174	51175	51661	51662
51663	51664	51665	51666	52153	52154	52155	52156
52157	52158	52645	52646	52647	52648	52649	52650
53137	53138	53139	53140	53141	53142	53629	53630
53631	53632	53633	54121	54122	54123	54124	54125
54613	54614	54615	54616	54617	55105	55106	55107
55108	55109	55597	55598	55599	55600	55601	56089
56090	56091	56092	56093	56581	56582	56583	56584
56585	57073	57074	57075	57076	57077	57565	57566
57567	57568	57569	58057	58058	58059	58060	58061
58549	58550	58551	58552	58553	59041	59042	59043
59044	59533	59534	59535	59536	-----		

** Part: Pipe-1