Numerical Simulations of Full Scale Tests

Investigating Effect of Fault Movement on Buried Pipeline

Lei Zhang

Delft University of Technology, Faculty CEG

Delft, The Netherlands, 2015
Abstract

This master thesis investigates the response of buried pipelines under ground-induced deformations through the finite element simulations of a testing program undertaken at TU Delft Stevinlab II as part of the GIPipe research project.

Starting with the analysis of collapsing steel rings which simulate the pipe-soil interaction, finite element based methods are applied to the analysis and design of shell structures. Then to come is the non-linear analysis of a pipeline under displacement control finally deformed with S shape curved and two critical parts included. Decisions of many properties and options will be made for GIPipe model building part; the operation procedure involves developing each part of the test to implement imposing geometrical imperfection prior to a physical and geometrical non-linear analysis for the pipelines, many of which have failed in local buckling in the most critical parts and assessing structural imperfection sensitivity from a proper scale factor and the exact position. For the pipelines that have the internal pressure, an additional setting step will performed during model building part.

Most of the tests have shown local buckling in the compression part of pipe cross section and several tests result in large axial strain. The author introduces an initial imperfection that aids the model in forming a local buckle in the finite element model. These imperfections are calibrated through the axial strain-fault displacement relationship measured by strain gauge in the tests. By varying the initial imperfection mode shape and scale factor it is discovered that the appearance of the local buckle is decided by many conditions mentioned above in every single test model simulation. Through the entire operation process of the deformed pipelines, the elastic-plastic ring springs behaviour is a key aspect needing to be implemented carefully.

Finally, comparisons of the axial strain and circumferential strain along the pipeline will be made to validate the finite element model.

Thesis Committee

Committee chair: Prof. ir. F.S.K. Bijlaard, Faculty CEG, TU Delft
Committee member and supervisor: Ir. S.H.J. van Es, Faculty CEG, TU Delft
Committee member: Ir. A.M. Gresnigt, Faculty CEG, TU Delft
Committee member: Prof. M.A.N. Hendriks, Faculty CEG, TU Delft
Graduation coordinator: Ir. L.J.M. Houben, Faculty CEG, TU Delft
Preface

This report presents the results of my Master Thesis. This graduation project was performed to obtain my master’s degree in Civil Engineering at Delft University of Technology. The subject of this report concerns the model building and the verification to complement and extend the full scale tests taken in Stevinlab II, TU Delft.

I would like to thank some people which supported me during this graduation project. First and foremost, I want to express my sincere gratitude to my advisor Sjors van Es for guiding me through this thesis. Without his detailed comments, advices, lots of patience and of course the experimental data I couldn’t finish this report successfully. And I also want to thank Professor Bijlaard, Mr. Gresnigt and Mr. Hendriks; they could always take time to give important and constructive recommendations for me in writing this thesis. Finally, I want to thank my family and friends which supported me during my study especially my boyfriend which gave me the encouragement and suggestions in spite of the long distance and the existence of the time difference to help me carry out this project.

As I will be leaving this place back to my country, I would like to leave this with you for your reading pleasure.

Lei Zhang

Delft, The Netherlands

2015.6
Content

Abstract.................................................................................................................................i
Preface ..................................................................................................................................iii
Content..................................................................................................................................v
1 Introduction ............................................................................................................................1
  1.1 Background .........................................................................................................................1
  1.2 Objective .............................................................................................................................3
  1.3 Problem description and existing knowledge .....................................................................4
  1.4 Limit state for buried pipeline ...........................................................................................7
  1.5 Report outline .....................................................................................................................8
2 Full Scale Pipeline Tests ............................................................................................................9
  2.1 Test setup overview .............................................................................................................9
  2.2 Ring spring devices .............................................................................................................11
  2.3 Laboratory experiments data .............................................................................................12
    2.3.1 Pipe types ......................................................................................................................12
    2.3.2 Soil types .....................................................................................................................12
    2.3.3 Internal pressure application ......................................................................................12
    2.3.4 Axial force under pipeline deformation ......................................................................13
  2.4 Test control methods ..........................................................................................................14
  2.5 Test results and measurements .........................................................................................15
    2.5.1 Test results ..................................................................................................................15
    2.5.2 Strain measurements ..................................................................................................15
    2.5.3 Ovalisation measurements .........................................................................................17
  2.6 Chapter conclusion .............................................................................................................17
3 Finite element analysis method ............................................................................................19
  3.1 Introduction .......................................................................................................................19
  3.2 Initial geometrical imperfection analysis .........................................................................20
  3.3 Nonlinear buckling analysis ...............................................................................................20
  3.4 Finite element analysis procedure ....................................................................................22
  3.5 Chapter conclusion .............................................................................................................22
4 Model Building and Results of Ring Springs .......................................................................23
  4.1 A series of steel rings ..........................................................................................................23
  4.2 Geometry and boundary conditions of FEA model ............................................................23
  4.3 Material property settings of the FEA model ....................................................................25
  4.4 Results of ring springs model ............................................................................................26
  4.5 Chapter conclusion .............................................................................................................28
5 Model Building of Full Scale Tests .......................................................................................29
6.3 Chapter conclusion........................................................................................................88
7 Conclusions and Recommendations.............................................................................90
  7.1 Conclusions..............................................................................................................90
    7.1.1 Nonlinear geometrical and material properties...............................................90
    7.1.2 Pipe-soil interaction.........................................................................................90
    7.1.3 Initial imperfections.........................................................................................90
    7.1.4 Advanced numerical model for investigating full scale tests........................91
    7.1.5 Conclusions from result comparison..............................................................91
  7.2 Recommendations for further studies.....................................................................92
8 Reference....................................................................................................................93
Appendix A: Specimen layout and properties.................................................................96
Appendix B: Material Tensile Test Data........................................................................99
Appendix C: Detailed Initial Imperfection Shape.........................................................108
Appendix D: Model building in ABAQUS....................................................................112
Appendix E: Comparison of results at strain gauges.....................................................119
1 Introduction

1.1 Background

Buried pipelines may suffer from serious damages during passing through harsh-environment regions. Large plastic deformation may occur due to landslides, ground settlements, liquefaction or large fault movement. This master project is a part of the “GIPIPE” project, which aims to develop safe design guidelines for buried steel pipelines under ground-induced deformations (GIPIPE research group, 2015). The GIPIPE is a research project that investigates the mechanical behaviour and structural integrity of buried steel pipelines, subjected to severe ground-induced permanent action with tectonic (quasi-static) effects, slope movements, and excavation-induced displacements included, using advanced experimental techniques and high-performance numerical simulations.

In the context of strain-based design, consideration of pipe-soil interaction is fundamental for determining extreme ground-induced actions (strain demand) towards pipeline safety. GIPIPE project through a multi-disciplinary partnership, considering the particularities of buried steel pipelines with emphasis on soil-pipeline interaction effects, combines geotechnical engineering concepts with pipeline engineering practice. The final objective of GIPIPE is the development of a complete set of design guidelines and operational recommendations for buried steel pipelines, in areas where ground-induced actions are likely to occur (GIPIPE, 2015), towards ensuring pipeline integrity against geohazards, reducing the risk to the population and the environment, increasing pipeline operations reliability and safeguarding the unhindered transportation of energy and water resources (Gipipe Project, 2015).

An active fault is a ground-induced permanent action between two portions of the earth crust along which relative movements can occur. The movement is concentrated in relatively narrow fault zones. Principal types of fault movement include strike-slip, normal-slip and reverse slip, shown in Figure 1.1. Normal faults occur due to tensile stress in each crust and reverse and trust faults occur due to compressive stress. Shear stresses produce strike-slip faults. In this project, the considered type of fault movement is the strike-slip which deforms a continuous pipe primarily in bending. Additionally a normal force which is tension or compression depending on the pipe-fault intersection angle occurs.

![Figure 1.1: Different types of fault movement](Source: “Where’s the San Andreas Fault”, 2006).
Analyzing the pipe response to the postulated movement, and the resulting tensile, bending, and compressive loads applied to the buried pipe is necessary for a safe design. For large deformations, the calculations could be done by finite element analysis model of pipeline and pipe-soil interaction. Finally, the computed response stresses, strains, buckling and ovalisations are compared to allowable limits established based on the required performance of the pipe following the fault movement. The design for loads and deformations associated with buried piping depends on the accuracy of the predicted ground movement and soil properties as well as the accuracy of the pipe-soil interaction model. Parametric variations of the model input data are usually necessary to bound the problem.

In the part done by Delft University of Technology, ten full scale tests aiming to simulate pipelines under severe shear-type deformation are performed. The shear-type deformation is the effects of events such as strike-slip fault movement on a buried pipeline (Figure 1.2). For the experimental tests, using real soil will be the most realistic representation of reality. However, by burying the pipe it is hard to see the appearance and the changing states of the local buckle or measure the force applied to the pipe by soil. To simulate the soil-pipe interaction, the real soil is replaced with nonlinear springs in the form of collapsible rings. Only horizontal movement of the pipe through the soil is considered. An angle between the fault movement plane and the plane perpendicular to the pipe axis is the fault movement angle, called $\beta$. The strike-slip fault plane with an angle $\beta$ which is not always perpendicular ($\beta = 0$) to the pipeline longitudinal direction. All relevant boundary conditions for the buried pipeline are created in the laboratory:

- Moving fault with a maximum displacement of approximately 1500 mm.
- Tensile force in pipe due to lengthening/shortening caused by fault movement.
- Pipe soil interaction by non-linear springs.
- Internal pressure.

In the full scale tests, one of the two soil masses in the box moves as the fault movement. The large moving boxes were built to represent and control the displacement of the moving fault, which is connected to the pipeline segment by twelve nonlinear spring behaviour devices as mentioned. These devices are constituted
by steel rings. For the different kinds of elastic-plastic properties from the different kinds of soil, steel rings, which will be called “ring springs” in the following text, in several diameters and lengths were used to establish the relation between the pipe and the moving box, like the soil surrounding the pipeline in reality. These ring springs give a lateral resistance against pipeline from fault movement, similar to the soil-pipe interaction in practice. This kind of discrete modeled soil (ring-springs) is designed to get a similar pipe-soil interaction as continuous soil model. For building experiments as mentioned, 10 unique tests for 2 types of pipelines in various situations were performed in the lab. In order to make a validated and reliable numerical model for studying the effect of buried pipe under large deformation, the result comparison investigation from these experiments and finite element model would be made. For the further research, the methodology could be improved and applied other types of permanent ground-induced actions, such as normal and reverse faults, as well as to buried pipelines subjected to landslides, differential settlement or lateral spreading.

1.2 Objective

The main purpose of this report is to build an advanced nonlinear analysis model to simulate the full scale tests investigating the effect of ground induced deformation on buried pipelines. This validated model could also be employed in a parametric study. Earlier studies of GIPipe project (Vazouras, 2012) have already developed the finite element model to simulate the pipeline, soil and their interaction through contact algorithm. It is different from previous studies that nonlinear behaviour springs are employed instead of the soil mass in the full scale tests in this report.

To achieve the objective, a numerical model for describing pipeline deformation under strike-slip fault action would illustrate the development of large strain at local buckling. This model is an enhancement of the simplified model presented in the experimental tests designing report (Van Es, 2015), and the following aspects are going to be taken into consideration:

- Dimensions of elements and development of connections;
- Applicable material properties of the pipe segments;
- Soil-pipe inelastic interaction behaviours;
- Application of the boundary conditions;
- Presence of internal pressure;
- Linear elastic buckling analysis as the initial imperfection of a local buckle;
- Development of large inelastic strains in the steel pipeline.

In this report, to prove that the built model can represent the experiment and simulate the whole test process integrally and correctly, investigations are expected to get the consistent value from test results and finite element model results: the relationship graph between the displacement of the fault movement and the axial strain along the pipeline length; the relationship graph between the displacement of the fault movement and

3
the circumferential strains at specific positions when the internal pressure present. The strains will be measured through the strain gauge during full scale tests.

Two results should coincide with each other as well as the occurrences and the positions of local buckling, otherwise modification should take place to make sure the validity and usability of the finite element analysis model.

1.3 Problem description and existing knowledge

The present study focuses on the structural response of continuous (welded) buried steel pipelines crossing active strike-slip faults. Those pipelines are subjected to an imposed soil mass movement ended in a curved S-shape. In this particular way, pipeline bending deformation may occur with the development of high stresses and strains associated in the critical part of pipeline wall. These strain and stresses are well beyond the elastic range of pipe material and may cause the pipelines failure. High tensile stresses may cause fracture of the pipeline wall, especially at defected locations or welds, whereas high compression stresses may cause buckling, either in the form of beam-type (global) instability or in the form of pipe wall wrinkling, a shell-type instability, referred to local buckling or kinking. Local buckling will occur before other types of structural instabilities in the case of pipelines with high D/t values especially when embedded in soil.

![Figure 1.3: Left: Definition of the fault displacements and angle \( \beta \). (Source: Dimitrios, 2007) Right: Fault displacements at angle \( \beta=0^\circ \).](image)

During a ground-induced displacement, local buckling may occur in the pipelines. A number of investigations have been performed and analysis methods are proposed for consideration of tensile and bending behaviours of the pipeline due to strike-slip fault movement. For buried pipeline subjected to fault movements that result in tensile stresses in the pipeline, Newmark and Hall (1975) first proposed a small deflection model and adopted static soil pressure and static friction force as shown in Figure 1.3 (Left) with a total fault movement \( U_{\text{fault}} \), in which a pipeline intersects a right lateral strike-slip fault at an angle \( \beta \), the fault movement plane mentioned in section 1.1. For a pipe-fault intersection angle \( \beta \geq 0^\circ \), the strike-slip fault results in tensile axial force and bending strains in the pipe. It is assumed that the pipeline is firmly attached to the soil at two anchor points. The study found that the resistant capacity of a buried pipeline to fault movement is dependent upon the soil characteristic, angle of crossing a fault, slip length and property of material, and the pipe resistance is maximized by minimizing the longitudinal and lateral resistance of the
soil to the pipe motion. Kennedy et al (1977) extended the ideas of Newmark and Hall, and incorporated the effects of lateral interaction and take large deflection theory into consideration for evaluating the maximum axial strain. In this model, no flexural resistance was considered. Only the axial tensile force at the point of inflection was used for constraint equilibrium. The study assumed that the pipeline is a flexible cable which was resisted by the uniform passive soil pressure, deformed as a single curve in both sides from middle point with constant curvature approaching asymptotically to the undeformed portion of the pipeline, Figure 1.3 (Right). It was evident that Kennedy et al. criteria are met only when the pipeline is subjected to large fault movements and is able to undergo large tensile strains without rupture. Another investigations regarding tensile behaviour due to large fault movement based on the beam model is the O’Rourke (1980) finite element approach. In common cases the maximum allowable strain is seriously reduced in order to account for thermal effects or metallurgical alterations induced by welding, which are well out of the range specified by the Kennedy et al. criteria.

Wang and Yeh (1985) refined and improve the existing results, in which the pipeline bending stiffness is taken into account. Generalizing the formulations for buried pipelines across any strike-slip faults that may cause either tension or compression failure of the pipeline, their methodology relies on partitioning of the pipeline into four distinct pipe segments (Figure 1.4): two in the high curvature zones on both fault trace sides and another two outside this zone. The former segments are assumed to deform as circular arcs while the latter ones are treated as beams-on-elastic-foundation. The effect of the axial force and the bending moment interaction and of large deformation will also be included. The study considered the problem in two fold nonlinear: geometry nonlinearity and material nonlinearity using the proposed technique and proper parameters which would yield more realistic results. Audiber and Nyman (1977) have performed experiments on the soil resistance against the horizontal motion of pipelines. It is found that the soil pressure-displacement relationship was nonlinear, showing a increasing of displacement at higher earth pressure. Later on, Wang and Yeh (1986) noticed to study the true failure mechanisms of buried pipelines due to fault movement, the effects of the nonlinear soil-pipe interaction should be considered.

**Figure 1.4:** Four pipe segments methodology with the surrounding soil interaction under strike-slip fault. (Source: Dimitrios, 2007)

In reality, Takada (1998) indicated that pipelines behave more like a shell when crossing a fault, especially for large D/t ratio pipelines. Until 1979, the vast majority of the work of a buried pipeline undergoing fault movement has modeled the system as a beam on an elastic-plastic of fracture problem of a pipe. Muleski (1979) modeled a buried pipe as a thin circular cylindrical shell in a resisting soil medium giving rise to other displacements and stresses in the pipe. In his model, account is taken of the displacements arising from the curvature of the pipe and therefore it appears that it could provide more realistic stress field than
the beam model. Therefore, it is necessary to analyze this problem by a shell model instead of through a beam-type approach for describing pipeline deformation, since the latter is less complete. Both Ariman and Lee (1991) and O’Rourke and Liu (1999) evaluated pipe strain using the finite element method which modeled the pipe as a thin cylindrical shell. All of these approaches mentioned above are mainly used to evaluate the pipe strain. Consideration of nonlinear property of the lateral soil spring and the pipe is the key problem in recent researches. Takada (2001) built a new simplified method for obtaining the maximum strain in steel pipes considering nonlinearity of material and geometry of pipe section, adding the effect of the large deformation of the pipe cross section. It is shown that the maximum strains in steel pipes are more sensitive to the bending angle than crossing angle.

O’Rourke and Liu (1999) indicated that the interaction at the soil-pipe interface can be modeled as an elastic spring as long as the relative displacement is less than the maximum elastic deformation, which is only applied for small to moderate levels of ground deformation. Alternate relations for the simulation of this interaction are the three soil spring constants for modeling the soil-pipe interaction as an elasto-plastic system. Lee and Bohinsky (1996) simulated the constraints of the surrounding soil by series of equivalent springs with nonlinear characteristics attached to the pipe in the longitudinal and the transverse directions. It is necessary to define the yield condition of the soil for large lateral displacements. Karamitros (2007) and Trifonov (2010) considered the soil-pipe interaction could be described by elastic-perfectly plastic behaviour during simulating the numerical model and verification. Recently, researches are focusing on the finite element method for the buried pipe under permanent fault movement using the soil-pipe interaction in the nonlinear behavior. Zhao (2010) established a finite element model with contact elements to analyze the response by consideration of soil-pipeline nonlinear interaction in the study. Liu (2008) made the soil-pipe interaction in axial, horizontal and vertical directions model with spring elements tie to every node following the soil spring properties calculated according to the ALA-ASCE guidelines, which is the elastic-perfectly plastic behaviour. In the GIPPIPE project, through a finite element modeling of the soil-pipeline system which accounts rigorously for the inelastic behaviour of the surrounding soil, Vazouras et al. (2010) developed a model of buried steel pipeline under fault movement assuming fixed conditions at the two ends of the pipeline. The soil-pipe interface in their model is simulated with a contact algorithm, which allows separation of the pipe and the surrounding soil. Vazouras et al. (2015) extended their work to a refined numerical model accounting for appropriate end effects. All of the finite element analysis models built by Vazouras et al. are using the elastic-perfectly plastic constitutive model accompanied by a frictional resistance or soil-pipe adhesion considered in the contact surfaces instead of the spring model.

To generalize the formulations for buried pipelines across the fault movement with the pipe-soil interaction which is applied by the nonlinear behaviour springs in tests, it is necessary to develop a model included these springs and buried pipelines under bending formation. Thus, the problem is using geometrical and material nonlinear analysis model to simulate and investigate the effects of the full scale tests taken in TU Delft.
1.4 Limit state for buried pipeline

For buried pipes, as for most structures, performance limits are directly related to stress, strain, deflection, or buckling. Since an actual crack due to high tensile strain is not likely occurred, loss of containment of the continuous pipeline due to fault movement is the main damage phenomenon. According to Kennedy (1997), X-grade pipeline steels generally can accommodate a strain level of 5% or more and a local strain of 15% or more without rupture. An acceptable tensile strain ranging from 2% to 6% was generally suggested depending on degree of quality control and extent of weld inspection. However, tests have shown that pipes can take substantially less strain under compression than tension (Lee, 1996). Usually, the formation of a buckle does not necessarily imply loss of containment, but further deformation, like folding, of the buckled area, excessive tension at the opposite side of the local buckling may cause pipe wall fracture as well (Vazouras, 2015). Either due to tensile rupture failure or buckling, apart from the detrimental effects, an ecological disaster may also result from the leakage of environmentally hazardous materials like gas or liquid waste.

The buried pipeline in this report is a steel pipeline. Loss of containment is the most important limit state. Taking both ultimate limit states and the serviceability limit state into consideration, the failure modes that may occur are (Gresnigt, 1986):

- Development of leakage due to cracking or rupturing.
- Development of inadmissible large deformations, such as excessive out-of-roundness and buckling. This failure mode is especially important from the pipeline manager's point of view. Since buckling may, for example, be associated with very large local strains, buckling increases the danger of cracking and leakage.

When either of these two failure modes occurs, a so-called limit state is attained. This is a state in which the structure is deemed to have become unserviceable or in which one or more of its parts have ceased to perform the function for which they were designed.

For this report the strain is the most important aspect for verification of the results from the test and the finite element method simulation. In Eurocode3 part4-3: pipelines, the plastic strain limitation verification should be as follow:

1. The maximum tensile strain $\varepsilon_{mx}$ should not exceed the limit strain $\varepsilon_{i,Rk}$ which is prevented by limiting the strain to some values in practice, defined by:

$$\varepsilon_{i,Rk} = z\%$$  \hspace{1cm} (1.1)

The value $z = 0.5\%$ is recommended in the Eurocode 3, Part4-3.

2. It should be demonstrated that the pipe wall with weld zones and allowed discontinuities has the strain capacity (limit strain) required for the structural analysis.
1.5 Report outline

Chapter 1 has given the background of this report, including the introduction of the GIPIPE project and earlier studies about buried pipeline under fault movement. In the following Chapter 2, the full scale pipeline under strike-slip fault movement test setup will be described firstly as well as the measurement equipment during the test progress and the test results. Then, Chapter 3 briefly surveys the motivation for the methodology of local buckling analysis. An explanation of the method how to apply finite element analysis to the investigation is exhibited. The finite element analysis method will start with a simpler nonlinear behaviour shell cylinder numerical model in Chapter 4. This model is simulated as the pipe-soil interaction ring springs in the full scale experiments. In the main part of the report is the investigation of the full scale tests shown in Chapter 5.

The structural responses of both steel pipelines under strike-slip fault movement and the big size cylinders as nonlinear springs are examined numerically, using the general purpose finite element program ABAQUS. The constraint method which established the connections between the ring springs and the pipeline, and the way to establish the nonlinear behaviour of this kind of springs in the full scale test model will be settled in Chapter 5. Through the parametric studies, the geometrically and materially nonlinear properties, mesh sizes and thickness layers are analyzed for constructing the finite element model, so that the pipeline performance criteria are evaluated with a high-level of accuracy.

After developing the entire model of the test, results are obtained of the relationships between the axial strain and the fault displacement at the same specific positions measured by strain gauge in practice. The results are for X65 and X60 steel pipelines with typical values of the diameter-to-thickness ratio around 50 under different soil type conditions. The strain-displacement relationship comparisons from the test and numerical model will be taken into account to investigate the accuracy of the finite element model in Chapter 6. Then some significant conclusions and recommendations for further research will be put forward in Chapter 7 of the report.
2 Full Scale Pipeline Tests

2.1 Test setup overview

The experimental program is carried out at TU Delft Stevinlab II. There are 10 full scale tests in the laboratory in total for simulating the formulation of the buried pipelines under ground-induced deformation. The test setup itself is a large pipeline bending test designed to give a displacement of a moving box. As shown in Figure 2.1 there are two big boxes constituted by assembled steel frames, one in the left side around the pipeline is fixed and the other in the right is moving. Ring springs are settled in between the box and the pipeline. The combination of box and the connected ring spring is a simulated soil mass in order to simulate the fault movement. The displacement which is dictated by the available hydraulic actuators will lead the ring springs to deform and behave in its elastic-plastic (nonlinear) properties and make the pipe deform in an S-shape (Figure 2.2). If the strain in the compressive side of the pipeline is too large, wrinkling (instability) will occur. During the test design phase, boundary conditions and applied loads are listed as follows:

- Boundary condition: displacement of the moving box (fault movement), limitation is 1500 mm;
- Applied loads: internal pressure in the pipeline and axial force in the two ends of the pipeline.

Figure 2.1: Top view of test setup, ringsprings are not drawn. (Source: Van Es, 2015)

Figure 2.2: Left: The schematic of test setup. The grey box is the moving box. Right: Pipeline and nonlinear behaviour springs after the displacement of moving box.
During the pipeline deformation under ground-induced displacement, internal pressure might be present in the pipeline, resulting expands in hoop direction which is not caused by the fault movement. Therefore, stress and strain arise in the pipeline. For the other applied load, the axial force in the two ends of pipeline is the superposition of two forces: the axial force from the internal pressure and the applied axial force with hydraulic actuators (Figure 2.3). More detailed information will be introduced in the following sections.

In every full scale test, 12 ring springs were applied to replace the soil property. Ring dimensions and locations were decided depending on the type of surrounding soil and the angle of fault plane according to the project design specification. For the buried pipeline, the interaction force from soil should be in the all 3 directions. However, in order to get a simplified strike-slip fault model and the most efficient model to simulate the fault movement induced soil-pipe interaction, the springs are arranged only in the part of pipeline has the interaction force with the soil in the transverse direction in the horizontal plane. These 12 ring springs is attempted to represent the realistic pipe-soil interaction resulting in acceptable accuracy and costs. With regard to this transverse displacement, pipelines would also displace transversely in the horizontal direction as same to fault movement direction. The test setup in practice is shown in Figure 2.4 Left, and the right one is the deformed pipeline after the displacement of the moving box.

Figure 2.3: Photographs of the end hydraulic actuator on the right side from fault.

Figure 2.4: Photographs of the test setup before (left) and after (right) the fault movement.
2.2 Ring spring devices

The 12 nonlinear behaviour springs (6 on each side of the fault), symmetrically distributed in between the moving box and pipeline, are made of steel cylinders in different diameters and thicknesses. It appears that when a steel ring is flattened, the behaviour closely resembles to the reference behaviour which is the target for the pipe-soil interaction in the test setup (Van Es, 2015). As mentioned in the overview of test setup, the pipeline is deformed by the displacement of moving box through the intermediate ring spring flattening. The ring springs just replace the soil property as the surrounding constraint.

These ring spring devices are tensile constructions with compression loading transferred to the two edge ends to characterize property behaviour close to the elastic-plastic properties of soil. During the fault moving there is a relative displacement between the pipeline and the moving box, therefore tensile forces will appear in the two ends of the ring spring device. This tensile force made the ring spring compress itself to achieve the elasticity at the first stage and then the plasticity depending on the material property of the steel ring. The relationship of ring force and ring deformation (flattening displacement) is capable compare to the real soil properties. Flattening as the final deformed pattern and the relationship diagram is shown in Figure2.5 respectively. “F ring” is the F shown in Figure 2.5 Left. “U ring” is the total displacement of the top and bottom central line of the steel ring in the force direction. Photograph of the ring spring devices are shown in Figure 2.6.

![Figure 2.5: Left: Tensile construction with compression loading of ringsprings (Source: Van Es, 2015). Right: Force and deformation behaviour of one ring spring in Test 1.](image1)

![Figure 2.6: Photography of ring spring device.](image2)
2.3 Laboratory experiments data

2.3.1 Pipe types

Since the available pipeline base material pipes are shorter than the length necessary for the test specimens, girth welds will be present in the test specimen. To investigate the influence of such a girth weld on the pipeline behaviour, in some cases, the weld will be placed in the most highly strained cross section (Van Es, 2015). Two steel types were available: 219×5.6 X65 and 406×7.4 X60, more details in Table 2.1. The length of every tested pipeline is approximately 20 meters. The material properties will be introduced in Chapter 5.2.

Table 2.1: Overview of the specifications of available pipeline material.

<table>
<thead>
<tr>
<th>Type</th>
<th>D</th>
<th>T</th>
<th>D/t</th>
<th>Grade</th>
<th>Fabrication</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>8.58”(219.1mm)</td>
<td>5.59mm</td>
<td>39</td>
<td>X65</td>
<td>HFW</td>
</tr>
<tr>
<td>2</td>
<td>16” (406.4mm)</td>
<td>7.4mm</td>
<td>55</td>
<td>X60</td>
<td>HFW</td>
</tr>
</tbody>
</table>

Pipelines are connected from several pipe segments through a girth weld. So the pipes used in the finite element analysis model also need to be cut in several parts and each part has its own geometric (an average diameter and an average thickness) and material property. Besides the ten full scale tests, four tensile specimen tests in the axial and hoop direction are taken place. For the model building, homogenous material property in shell section is applied. The input data for the property in the finite element model will be the tensile specimen stress-strain relationship results in specimens from axial direction. The pipeline material property is one of the parameters which lead various pipeline deformations during the fault movement. The summary of the magnitude of diameter, thickness, length of the pipe segments and their layout of every continuous pipeline are shown in Appendix A.

2.3.2 Soil types

Different soil parameters will influence the response of present soil acting on a pipeline under the fault movements. Within the 10 full scale tests, two properties of soil have been defined to be taken into consideration: soft to firm clay and loose sand. It is simply designated as “clay” and “sand” during the design phase. In practice, the exact soil resistant behaviour against a moving pipeline depends not only on the soil type, but also the pipe burial depth. For all tests, a burial depth of 2.5 m of the pipe centerline is assumed. The approach to achieve the accurate soil properties is to apply the two kinds of soil types into the ring spring devices through various geometric properties and locating at specific positions.

2.3.3 Internal pressure application

Significant internal pressure is present in the four pipeline tests. The longitudinally constrained pipeline resisted the lateral contraction results in a defined normal stresses in three levels. These three levels are including the ratio of hoop stress $\sigma_{\text{hoop}}$ and specified minimum yield stress of 0, 0.25 and 0.5.
2.3.4 Axial force under pipeline deformation

When the surrounding soil fault starts, the axial force always appears along the pipe due to the soil constrained. In the tests, because of the pipe-soil interaction is applied by cylinders, the axial force should be applied in addition. The strike-slip fault plane angle \( \beta \) in Figure 2.7 and Table 2.2 is assumed to only influence the axial force at the two ends of the pipeline instead of the pipeline curved angle. The ground induced deformation under the perpendicular movement leads to axial forces in the pipeline due to elongated pipeline length. In reality, the plane of fault movement is not always exactly perpendicular to the pipeline axis. In case of the plane is non-perpendicular to pipeline axis, a tensile or compressive normal force may arise. In all the ten tests, the fault movement plane is perpendicular to the pipeline. However, since very small angle variations already give large differences in the axial force in the pipeline, the effect of such angles can still be approximated by only varying the axial normal force (Van Es, 2015).

![Figure 2.7: Different axial force types under several plane angles of fault movement. (Source: Van Es, 2015)](image)

The effect of the axial force is significant. For a pipeline under high tension, the high axial forces reduce bending and thus the axial strain, while low axial forces do not reduce bending. Compressive axial forces also occur that possibly may lead to instability of the pipe wall. The recognized axial forces in a real event are:

1. \( F_s \): Axial force due to the formation of an S-curve along the pipeline length. This force is tensile and arises due to the formed shape length is longer than the original straight pipeline. This force only comes into effect for longitudinally constrained, long and straight pipelines. In the experimental study and the numerical model, all pipelines are regarded as such situation.

2. \( F_\beta \): Axial force due to a non-perpendicular angle \( \beta \neq 0 \) between soil movement and pipeline. For negative angles, the resulting axial force in compressive, for positive angles, the resulting axial force is tensile. The soil fault angle as Figure 2.7 shows.

3. \( F_P \): Axial force due to constrained expansion in hoop direction from internal pressure. This contraction is restrained in case of a longitudinally restrained pipe, long, straight pipelines in a result with an axial tensile force.

Whether the axial force is tensile force or compressive force will depend on the summary of all the three aspects above. Magnitude of the applied axial force to simulate realistic force can be measured during the whole testing progress and output from the computer. This result data is the input data of the numerical model.

Table 2.2 shows the summarized physical properties of the surrounding soil and the angle of the strike-slip fault plane. The left data in the “buckle deformation” is the displacement of fault when first
local buckling happened. In case that a girth weld was placed in the most highly strained cross section, the identical cross section on the other side of the moving soil boundary (identical by symmetry) was always in a plain pipe part to make a direct comparison according to the experimental design. The results of tests will be showed and analyzed in the comparison part with the finite element method simulation result together.

Table 2.2: Overview of the performed test boundary conditions.

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Pipe type</th>
<th>Soil type</th>
<th>Fault plane angle β</th>
<th>Internal pressure</th>
<th>Buckle deformation* [mm]</th>
<th>Local buckle</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>Sand</td>
<td>0°</td>
<td>0</td>
<td>-1480</td>
<td>No</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>Sand</td>
<td>0°</td>
<td>0</td>
<td>L:910 R:840/1480</td>
<td>Yes</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>Clay</td>
<td>-3.25°</td>
<td>0</td>
<td>L:1250 R:810/1480</td>
<td>Yes</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>Sand</td>
<td>-1.50°</td>
<td>0</td>
<td>L:690 R:800/1470</td>
<td>Yes</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>Sand</td>
<td>-2.25°</td>
<td>25%</td>
<td>L:450 R:580/700</td>
<td>Yes</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>(rupture failure)</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>Sand</td>
<td>0°</td>
<td>50%</td>
<td>-1470</td>
<td>No</td>
</tr>
<tr>
<td>7</td>
<td>2</td>
<td>Clay</td>
<td>0°</td>
<td>50%</td>
<td>-1470</td>
<td>No</td>
</tr>
<tr>
<td>8</td>
<td>2</td>
<td>Sand</td>
<td>0°</td>
<td>0</td>
<td>L:890 R:700/1480</td>
<td>Yes</td>
</tr>
<tr>
<td>9</td>
<td>2</td>
<td>Clay</td>
<td>-2°</td>
<td>0</td>
<td>L:1160 R:1030/1480</td>
<td>Yes</td>
</tr>
<tr>
<td>10</td>
<td>2</td>
<td>Sand</td>
<td>-2°</td>
<td>25%</td>
<td>L:680 R:640/1470</td>
<td>Yes</td>
</tr>
</tbody>
</table>

*Left value is the fault movement when local buckling appears; right value is the final fault movement. The fifth test stopped for the pipe wall rupture failure.

2.4 Test control methods

Test is controlled by some specific definitions during the fault movement.

-A displacement control of the moving box is increased in steps of 1 cm at a speed of 2 mm/s until the limitation 1500 mm is reached through hydraulic actuators 2 and 3 (the actuator number is marked in Figure 2.2).

-A force control unit adjusts the behaviour of actuator 1 providing the designed axial normal force to the pipeline end. This axial force for every test also needs to be input to the numerical model. The quantitative value of the axial force will be displayed in the section 5.5 when building the model.

-Another important control unit which is the deformation control manages the behaviour of actuators 4 on the opposite pipeline end side to actuators 1. The controller regulates the actuators on the basis of an axial displacement measurement of the pipe in the centre of the test setup (Figure 2.8), that the middle cross section of the pipe remains in the middle of the test setup. As a result, the retreat of both pipe ends due to the formation of an S-curve is equal in theory (Van Es, 2015).
2.5 Test results and measurements

2.5.1 Test results

After testing, most of pipelines are deformed in a symmetrically S-curve shape with high strain in the tensile side of pipe cross section and local buckle in the compressive side. Since the setup and all the boundary conditions are designed symmetrically on the two sides of fault, the most deformed length, which is introduced as the “critical part” in the following description, is approximately symmetrical to the center of the pipeline. The distance from critical part to center fault plane is around ±1500 mm for pipe type 1 and ±2500 mm for pipe type 2.

The shape of the local buckling can be introduced as a main dent and two dents in a line beside the main dent in the circumferential direction. In-between is the bulge part of the local buckling. It is different when the internal pressure is applied: the local buckling will only include one bulge shape. Theoretically, the local buckling position should be symmetric, but due to the various initial imperfection of the pipeline, like girth weld in one of the two sides or some other imperfections in the facility for instance, connections of the ring springs and their transmission units, and different yield strength of steel, it can’t be perfect symmetrical at all. More seriously, these issues may cause early local buckling failure.

Besides monitoring the displacement of the fault movement and the force in hydraulic actuators during the test, many other data was collected as well. These measured data included real-time axial strain in the longitudinal pipeline direction, the circumferential strain when internal pressure is applied, ovalization at various specific locations, the force-deformation relationship of the ring springs, etc.

2.5.2 Strain measurements

Axial strain is a good predictor of explicit deformation for such a large deformed structure. It is also the most important result used to compare to the result from finite element analysis model. In the comparison
part of this report, strain will be employed to validate the model.

Under the displacement of the moving fault and the nonlinear behaviour of the ring springs, in the aspect of axial strain the pipelines will result through medium strain (1%), high strain (3%), or occurrence of local buckling in the compressive part even the rupture failure in the tensile part in the final curved shape.

The axial and circumferential strain are measured from many strain gauges, called RK XX (serial number) in the test setup (see Figure 2.9), distributing along the relative critical part in the front and back side of the pipeline. The relative critical part length is ranged from the left-most ring spring to the right-most ring spring.

![Photograph of strain measurements.](image)

Figure 2.9: Photograph of strain measurements.

When a local buckle forms in some part of the pipeline, energy released, so that the axial strain in the compressive side of pipeline cross section will no longer increase from that moment. Or, there will be an inflection point in the strain-fault movement relationship graph in the tensile side of pipeline cross section. For other tests which only have the high strains in either tensile part or compressive part, detected deformations through strain in longitudinal and circumferential directions of the pipeline are worthy in studying to validate the model.

An overview of the final pattern with regard to the axial strain analysis is shown in Table 2.3. For the reason of large axial normal force or the internal pressure, there is no local buckling but only large strain in Test 1, Test 6 and Test 7. The local buckling position is measured from the center of deformed S-shape pipeline. Largest strain position is obtained from the strain gauge in the pipeline compressive part.

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Pipe Type</th>
<th>Local buckling or largest compressive strain position distance from pipeline center [mm]</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test 1</td>
<td>1</td>
<td>-1500 1750</td>
<td>No buckles</td>
</tr>
<tr>
<td>Test 2</td>
<td>2</td>
<td>-2560 2555</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 2.3: Overview of the most critical part in the two sides of pipelines.
<table>
<thead>
<tr>
<th>Test</th>
<th>ID</th>
<th>Lower</th>
<th>Upper</th>
<th>Observation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test 3</td>
<td>1</td>
<td>-2240</td>
<td>2340</td>
<td>Right buckle at girth weld</td>
</tr>
<tr>
<td>Test 4</td>
<td>1</td>
<td>-1615</td>
<td>1875</td>
<td>-</td>
</tr>
<tr>
<td>Test 5</td>
<td>1</td>
<td>-1445</td>
<td>1450</td>
<td>Left buckle at girth weld</td>
</tr>
<tr>
<td>Test 6</td>
<td>1</td>
<td>-1630</td>
<td>1630</td>
<td>No buckles</td>
</tr>
<tr>
<td>Test 7</td>
<td>2</td>
<td>-3625</td>
<td>3625</td>
<td>No buckles</td>
</tr>
<tr>
<td>Test 8</td>
<td>2</td>
<td>-2900</td>
<td>2530</td>
<td>Right buckle at girth weld</td>
</tr>
<tr>
<td>Test 9</td>
<td>2</td>
<td>-3625</td>
<td>3630</td>
<td>-</td>
</tr>
<tr>
<td>Test 10</td>
<td>2</td>
<td>-2565</td>
<td>2220</td>
<td>-</td>
</tr>
</tbody>
</table>

### 2.5.3 Ovalisation measurements

Ovalisation measurements are monitored at various locations along the pipeline. Two types of measured method are exhibited in the ten tests depending on the existence of internal pressure. For unpressurized pipelines, a rod is settled through the pipeline, measuring the vertical ovalisation shown in Figure 2.10 Left. Since the pipe is unpressurized, the ovalisation is assumed to be equal in horizontal and vertical direction according to the experimental design, except at the location of a local buckle (Gresnigt, 1987). In case of pressurized pipelines, a rod through the pipe is not possible. Assumption that horizontal and vertical ovalisation are approximately equal is no longer valid. Therefore, in case of internal pressure presence, ovalisations have been measured with the aid of external brackets fitted with a bending strip equipped with strain gauges (Van Es, 2015), shown in Figure 2.10 Right. The locations of the strain gauge and ovalization measurement vary between the different tests. Since the result comparison in Chapter 6 for each test has checked the strain of the pipeline, only the coordinate positions of strain gauges will be indicated.

![Figure 2.10: Left: Photograph of ovalisation measurements. Right: Photograph of ovalisation measurements for pipe with internal pressure.](image)

### 2.6 Chapter conclusion

This chapter introduced the physical properties of the ten experimental tests, which is included the test setup dimensions and the material properties of the pipe segments. Except for the pipeline itself, the pipe-soil interaction is another important part of the tests. Soil is presented by the ring springs in the tests. Having known the rationales about the ring spring devices is helpful to build the pipe-soil interaction
element and its property in the numerical model.

Furthermore, axial force at the two ends of the pipeline and the internal pressure would be provided in the setup. Strains, ovalisations, force and displacement are the obtained values from the full scale tests analysis and measured through the measurement equipments during the test.
3 Finite element analysis method

3.1 Introduction

Modern technological advances challenge engineers to carry out increasingly complex and costly projects, which are subject to severe reliability and safety constraints (Dhatt, 2012). For a proper understanding, analysis needs mathematical models that allow them to simulate the behaviour of complex physical systems. Engineering sciences are used to describe the behaviour of physical systems in the form of partial differential equations. The finite element method has become one of the most frequently used methods for solving such equations. It consists of using a simple approximation of unknown variables to transform partial differential equations into algebraic equations.

In the GIPPIPE project, these numerical models are used during the design phase of the projects to predict the rough deformation shape of the pipeline under fault movement and during the investigation phase to verify the test result. The final explicit finite element analysis model could be extended to the design phase of other strike-slip fault tests. Since the finite element analysis (FEA) is employed to investigate the buckling behaviour of the pipeline and the force-deformation relationship of the ring springs, the finite element package ABAQUS is used, with implicit methods, implemented in ABAQUS/Standard (Version 6.10).

Both the ring spring under compressive force and buried pipeline model under fault movement will be considered in two numerical models using shell structures. The nonlinear material properties and geometric dimensions are based on the tensile specimen tests which will be introduced in the relevant model building part in this report. The shell element type, mesh size, integration points along the shell thickness and the most important static analysis method have to be defined in most appropriate ways.

No structure is perfect. Shell structures behaviour is especially sensitive to the imperfections. The buckling load of thin-walled shells is overestimated by theoretical methods mainly due to the high sensitivity to geometrical imperfections of such structures (Casado et al., 2014). In the numerical model, a geometrical imperfection should be applied on the perfect pipeline as the initial imperfection before the geometrical nonlinear analysis. The buckling analysis will behave based on the initial imperfection trending towards to the local buckle as same as test results.

Buckling of shells which happens with small initial displacements is under a geometrical nonlinear behavior. Three different types of analysis are utilized and compared: elastic linear buckling analysis, geometrical nonlinear static analysis Riks method and geometrical nonlinear static analysis general method. At first FEA step, the initial geometric imperfections are not considered in the ring spring model, so this finite element model will just use the nonlinear general method. The model building procedure and results comparison are described in Chapter 4. Then the geometric imperfections are accounted for in the full scale pipeline model to evaluate their effects on critical compressive part. Parameters are discussed based on the full scale numerical model and result comparisons of Test 1 are taken for reference.
3.2 Initial geometrical imperfection analysis

It is hard for a perfect shell element to compute convergence during instability analysis in finite element analysis. The elastic linear buckling analysis is used to predict the initial imperfection of thin shell and the corresponding buckling shapes. Before the nonlinear analysis of the full scale model, the elastic buckling behavior of thin wall pipeline is obtained with shell finite element eigen-buckling analysis in ABAQUS. The buckling load or shape is generally used as a parameter to estimate the critical buckling loads in determining the nonlinear buckling strength and final deformation of stiff structures. Usually, the buckling shape is used for the description of the imperfections when the maximum amplitude of the imperfection is known but its distribution is not known. Superposing of multiple buckling shapes may be used as the initial geometric imperfection in nonlinear buckling analysis (Sarawit, 2003).

Before the fault movement test, no geometrical imperfections are measured in advance. So the initial imperfection could only be estimated from the final deformation of the pipeline. The elastic linear buckling analysis response could predict the deformation shape prior to nonlinear buckling analysis. However, even when the response of a structure is nonlinear before collapse, a general eigenvalue buckling analysis can provide useful estimates of collapse mode shapes (Analysis User’s Manual, 6.10).

3.3 Nonlinear buckling analysis

In the study where problems involve geometric nonlinearity and material nonlinearity prior to failure, a pre and post buckling analysis is needed to investigate the load-deflection behaviour. Several convergence approaches are possible depending on the selected algorithm and how the boundary conditions are applied. When the loads can be applied by means of the prescribed displacements, and no snapback behaviour occurs, a displacement increment method (where proportional displacements are applied) is used. In other cases, the modified Riks method (where proportional loads are applied) is used in order to be able to pass limit points (Sarawit, 2003). Both approaches are effective in obtaining nonlinear static equilibrium states during the unstable phase of the response.

In finite element analysis model of the full scale test, for numerical post buckling analysis the modified Riks method is applied. Modified Riks method is generally used to predict unstable, geometrically nonlinear collapse of a structure. It is often needed to seek nonlinear static equilibrium solutions for unstable problems in these tests, where the response of the load-displacement can show high nonlinear behaviour—that is, during periods of the response, the load and/or the displacement may reduce as the solution progresses and the structure must release strain energy to remain in equilibrium. The modified Riks method is an algorithm that gives effective solution of such cases. According to ABAQUS Analysis User’s Manual, if there is concern about material nonlinearity, geometric nonlinearity prior to buckling, or unstable post buckling response, a load-deflection (Riks) analysis must be performed to investigate the problem further.

The nonlinear analysis Riks method uses the load magnitude as an additional unknown; it solves
simultaneously for loads and displacements. Therefore, another quantity must be used to measure the progress of the solution; ABAQUS/Standard uses the arc length “l”, along the static equilibrium path in load-displacement space. This approach provides solutions regardless of whether the response is stable or unstable; the load displacement relationship can be propagated as shown in Figure 3.1 instead of a monotone increasing relationship as in General Static analysis method. From the Figure 3.1 as can be seen, for the same load 1.0 different displacement could happen at both point A and point B due to the feasible snapback of the whole model.

![Figure 3.1: Typical unstable static response. (Source: ABAQUS/Standard Analysis User's Manual, 6.10)](image)

However, for some situations there is no local buckling, which means the load is continuous increased with respect to the growing displacement during the entire analysis; In that case general static analysis could be applied as well. For Test 1, there is no local buckling, so at the first time general static analysis is chosen for model calculation convergence and then the Riks method. From the result comparison of Test 1 shown in Figure 5.9 of section 5.3.1 it can be seen that there is almost nothing differences between these two analysis methods for tests without local buckling. For a more systematical analysis in this series of tests, modified Riks method will be the only nonlinear analysis method applied on all the ten tests in the report.

Axial strain along the front and back central line in longitudinal direction resulted from the test and finite element model would be compared to validate the analysis. The coordinates of the strain gauges will be introduced in the following chapter 6. The position of local buckling is shown in the figures from Appendix C and described in Table 2.3. Test analyzed results are shown in the Appendix E: axial strains is drawn on the vertical coordinate, the displacement of the fault movement is drawn on the horizontal coordinate. The gradient of strain-displacement curve before the local buckle is constant.
3.4 Finite element analysis procedure

The procedure flowchart for the finite element model building and the analysis criterion is shown in Figure 3.2.

![Finite element analysis procedure flowchart](image)

Figure 3.2: Proposed procedure for identifying and analyzing pipeline under fault movement.

3.5 Chapter conclusion

This chapter introduced the finite element analysis method. Computer numerical simulation using the finite element analysis approach provides many advantages over conducting physical experiments, especially in parametric studies. In some cases, physical experiments are still conducted to verify the analytic approach and the assumptions made. The finite element analysis program ABAQUS is mainly used in this project.

Elastic linear buckling analysis is applied to obtain suitable initial imperfections to aid local buckle formation in the nonlinear test analysis. For the nonlinear buckling analysis, modified Riks method is employed rather than general static analysis method to have a displacement control pipeline deformation progress.
4 Model Building and Results of Ring Springs

4.1 A series of steel rings

When a load is placed on a flexible pipe, the pipe deflects and provides resistance because of its stiffness. It is even possible, using the property of large size pipe to think of soil as being a nonlinear behaviour spring that resists movement or deflection because of its stiffness.

As introduced in the test setup, these tests is designed to describe the pipeline behaviour under the fault movement, and the surrounding soil is a very important aspect this project. Some mechanical devices are applied to simulate the soil on the basis of its property. The experiment is designed to mimic the soil model of Brinch Hansen (1961) which is commonly used to find the maximum possible horizontal resistance of soil against lateral movements of the foundation. If a pipeline moves through soil, the model assumes linear elastic behaviour of the soil until a maximum capacity is reached (Van Es, 2015). In the test, a serious of cylinder structures (large size pipe) which are made of steel is modeling the soil-pipe interaction. It appears that when a steel ring is flattened (see Figure 4.1), the behaviour closely resembles the bilinear diagram that is defined by the Brinch Hansen model. By correct of the ring diameter, wall thickness and material mechanical behaviour, the desired soil properties can be represented by such a ring.

![Figure 4.1: Overview of the formation of four plastic hinges in the ring wall. (Source: Van Es, 2015)](image)

4.2 Geometry and boundary conditions of FEA model

Soil stiffness is one of the most important parameters affecting the response of a pipeline. The effect of soil stiffness on a bending pipe is greater in transverse direction than in axial direction in normal case. To make sure different types of ring springs used for simulating the soil behaviour are working as the designed property in the test, finite element models are built to investigate.

For having a better understanding of the shell structure, a simple model which is used as the springs in the later full scale model for the ground-induced pipeline in Test 1 was first built and analyzed in the report.
before the ten finite element models for full scale tests. These rings are the short segments cut from cylinders in different sizes. Their D/t ratio is approximately 50. Shell elements were chosen for the ring-spring simulation. The model is a quarter of the entire ring segment with symmetric constraints in the two directions aiming to a simpler and effective analysis.

In order to have a uniform distribution tensile force $F$ on the upper and lower lines along the direction of pipe length, transmission part (rectangular hollow sections and steel rods) is applied to the ring springs. Not all the ring springs has the transmission part. The ring springs far away from the middle point have longer ring length to get higher soil stiffness and restrained the pipe in the opposite transverse direction.

For the ring springs in the middle between them (the third and the fourth ring in general, with the number in ascending order from the middle point in one side), no transmission part is applied. However, transmission part might be applied on all the ring springs to work effectively. In this chapter only the ring springs in Test 1 are investigated, the detailing dimensions of the six rings in one of the two sides for the first test are showed in the Table4.1.

<table>
<thead>
<tr>
<th>Table4.1: Dimension of the ring springs in the first test. [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cross section</td>
</tr>
<tr>
<td>T1R1</td>
</tr>
<tr>
<td>T1R2</td>
</tr>
<tr>
<td>T1R3</td>
</tr>
<tr>
<td>T1R4</td>
</tr>
<tr>
<td>T1R5</td>
</tr>
<tr>
<td>T1R6</td>
</tr>
</tbody>
</table>

To achieve this kind of load condition, the central line of the one of top or bottom transmission part should be fixed in the x, y, z directions. This boundary condition is achieved through the central line of the ring for the ring springs without the transmission part. Then, displacement in x direction controls the final displacement of the other non-fixed central line. Symmetrical boundary conditions are:

-Along the perimeter and the cross section of the two transmission parts if existed about the z direction;

-Along the ring length and the middle surface of the transmission parts if existed about y direction.

In the test, there is no an extra surface in the middle of the transmission part. However the stiffness of rectangular hollow section is much larger than the shell element, so this simulated model could be effective. See the boundary conditions in the left one of Figure 4.2.

This finite element analysis model is one of the simplest models to get a quick knowledge of the whole nonlinear analysis procedure and how FEA program works. The shell element shown in Figure 4.2 Right is final stress distribution pattern and the final deformation of the ring spring.
4.3 Material property settings of the FEA model

In the process of building the FEA model, true stress and true strain are asked to input in the material property part. Since the results from the steel tensile specimen test are the engineering stress and engineering strain, some conversations should be taken in the preparation step. The true stress and true strain can be obtained from the equation listed below (engineering stress=nominal stress):

\[
\sigma_{\text{true}} = \sigma_{\text{nom}}(1 + \varepsilon_{\text{nom}}) \tag{4.1}
\]

\[
\varepsilon_{\text{true}} = \ln(1 + \varepsilon_{\text{nom}}) \tag{4.2}
\]

Then the plastic strain can be calculated by

\[
\varepsilon_{\text{true,pl}} = \varepsilon_{\text{true}} - \frac{\sigma_{\text{true}}}{E} \tag{4.3}
\]

Axial force and the displacement along the deformation direction of the rings are the analysis result. The results of the experiment and the FEA model are shown in the Figure 4.3. The name T1R2 means that the second ring from the middle point in Test 1, the rest can be obtained in the same way. From the result in Figure 4.2 it is seen that the largest stress is arisen in the 4 critical point shown in Figure 4.1 before. After the plastic hinge arose, the deformation was increasing without a growing force which is similar to the theoretical soil-pipe interaction properties. Compare to the force concentration point in the two side face, the two points on the top and bottom which is the side transmit load will show the plastic properties
firstly, see Figure 4.3, Right. In some particular rings in ten tests the plastic hinge appeared in these positions would be ruptured, leading the energy released immediately for the whole ring and the loading transmission to the pipeline. Furthermore it can’t take more force after the rupture failure. It can be learn from that, a proper design for every single ring is also important in the preparation stage to have a complete simulation of the soil-pipe interaction for fault movement.

Figure 4.3: Left: Rings in setup before the test. Right: A ring with a ruptured plastic hinge after the test.

4.4 Results of ring springs model

It can be seen from the comparison graph (Figure 4.5) that there is a gap between two curves. Differences from test result and simulation result always exists. Some reasons can be the explanation for this numerical model, for instance:

- Variation of material properties along the pipe of which the rings were made.
- The calculated material properties which could be improved may end with different deformations.
- The contact condition between the hollow rectangular section and the ring. In the ring spring experiments, load is applied through two steel rods which are simplified to the line loading with the displacement control method.
- The residual stresses along the entire ring which are neglected in the numerical model.

There are some finite element tests for the second ring from Test 1 for using different mesh size or thickness layers (the definition of these parameters will be introduced in the later chapter in full scale model). The differences between the results are very small, see Figure 4.4.

Unavoidable experimental error and the simplified FEA model would also introduce the difference between the test result and numerical result. So in the later ten full scale GIPIPE models building, the force and displacement results measured by every single real test (the blue line in the relationship graph) will be indicated as the spring property data.
Figure 4.4: Ring deformation and force relationship result of ring2 in Test 1 under different mesh size or thickness layers.

Figure 4.5: Comparison result graphs of the results from experiments and FEA model
4.5 Chapter conclusion

This chapter mainly discussed the numerical simulation and its force-displacement relationship result of the devices used as the ring springs in the full scale pipeline tests. Using shell element in the finite element analysis is a good way to investigate the nonlinear behaviour rings with the ratio of diameter-to-thickness of 50.

The result comparison still shows some distinguishes between the experiment test and FEA model. So in the following GIPIPE model, the ring spring behaviour will apply the force-deformation relationship data measured from the real test to reduce the error.
5 Model Building of Full Scale Tests

5.1 Numerical model description

The finite element analysis is employed to investigate the buckling and post-buckling behaviour of the pipeline shell under fault movement displacement. For this purpose, a numerical model using the finite element package ABAQUS is established. The numerical model considers a complete pipeline structure element, so as not to influence buckling deformation and the buckling shapes, imposing symmetric conditions, external and internal loads. Information about many properties for the full scale tests should be defined during the model building.

The material properties and the geometric dimensions of the pipeline cross section are the same to those values of the real pipeline used. Results from the investigating preliminary analysis show that the effect of the soil movements hardly present at more than 10distance from the fault movement interface. Therefore the specimen length was limited to 20 m (Van Es, 2015). Every pipeline in numerical models has the same dimension in the length to the specimen, which is the distance between (not included) the two end hydraulic jacks, specific magnitude of the pipe segment lengths have been shown in Appendix A.

When building the numerical model, outside diameter of the entire pipeline is constant. Because of the various thicknesses for different segments, several cross sections are defined in each model. Even though the thickness distinguishes are very small, some break changes in strain will take place when the internal pressure employed, this situation will be discussed in more details in section 5.4.

Ring spring device is an element using an interaction function in the finite element package ABAQUS named “connection”. The nonlinear behaviour of the ring spring is defined by the motion properties of the connections.

In case of the displacement control and the force applications, the numerical analysis will be performed as the same to the real test. The initial imperfection shape building methods will be introduced and described in the following sections.

5.2 Material properties

After construction of the pipeline geometry, material properties should be applied first to determine the physical properties of the objective. The steel pipe material is described with a large-strain von Mises plasticity model with isotropic hardening employed, and its calibration is performed through an appropriate uniaxial stress-strain curve from the tensile test on a coupon specimen.

Material properties of the pipeline, that is to say the steel stress strain relationship, in the GIPPIPE model will determine the ratio of axial strain and fault movement displacement in the elastic stage, the time of local buckling appearance and the axial strain as the output comparison result in the plastic stage to
validate the finite element mode through the experiments.

As mentioned in previous chapter, the pipeline in the every test is connected by several (3/4/6) parts of pipe segment through the girth weld. These segments are made of the materials from different types which are listed in Table 5.1.

Table 5.1: Material types used in two different pipeline sizes.

<table>
<thead>
<tr>
<th>Pipe Type No.</th>
<th>Type1</th>
<th>Type2</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>H86916400B</td>
<td>H60410114</td>
</tr>
<tr>
<td></td>
<td>H86916331B</td>
<td>H60413730</td>
</tr>
<tr>
<td>Steel Type No.</td>
<td>H86916024</td>
<td>H60415859</td>
</tr>
<tr>
<td></td>
<td>H86916331B</td>
<td>H60423640</td>
</tr>
<tr>
<td></td>
<td>H86916270B</td>
<td>H60415422</td>
</tr>
<tr>
<td></td>
<td>H86916288B</td>
<td>H60416313</td>
</tr>
<tr>
<td></td>
<td>H86916288</td>
<td>H60416153</td>
</tr>
<tr>
<td></td>
<td>H86916314</td>
<td>H60423631</td>
</tr>
</tbody>
</table>

Result from the tensile specimen tests data (two sets of test result in axial direction, from section 2.3.1) are the engineering stress and engineering strain, which is the orange solid line in Appendix B graphs. To get the true stress and true strain for the reason mentioned in the ring spring FEA model, using equations (4.1) - (4.3) to calculate and get the true yield stress and plastic strain for the plasticity of the steel material properties; and the parameters for elastic property are shown in Table 5.2. In the first several simulation trials, the Young’s Modulus for these various steel types are chosen from 190000 N/mm² and 200000 N/mm² according to the engineering stress and strain graphs. Since the critical procedure is not in the elasticity of the steel, with the purpose of getting a simpler stable model and an ideal continuous initial imperfection result in the linear buckling analysis, the Young’s Modulus for every steel type is consistent with the value. The true stress and true strain are in the orange dashed line in the same graphs for different steel types.

Table 5.2: The elastic property parameters.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus</td>
<td>210000 N/mm²</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.3</td>
</tr>
</tbody>
</table>

From the combination of pipe segments graph in Appendix A, it can be seen that several types of material properties will be used in every single pipeline in one test. The different material properties are compared in the second set of graphs in Appendix B according to the test number to show a clear stress strain relationship of every part of the entire pipe. These little diversities in strain-stress relationship will lead to a discontinuity of the cross section properties at the connection positions. The maximum diversity in each graph is approximately in the strain range after 0.3% (0.003). Various stress-strain relationships will lead to distinguish in the axial strain along the pipeline length especially in the plasticity period. Because of the different material properties between two adjacent pipe segments and the girth weld in between them,
peak stress and strain will turn up in some tests leading to the local buckling located at the girth weld connection position or even rupture failure. This will be discussed in the specific simulation result analysis in the following Chapter 6. That is one of the most likely reasons for the different results from the test and FEA model.

5.3 The application of ring springs

5.3.1 The connections between springs and pipeline in practice

For representing the elastic-plastic material properties of the soil, there are 12 ring springs surrounding the pipeline in a designed order for each test. One of the two ends of the ring spring device is the moving box simulating the soil mass to control the displacement of the fault movement during the test. The other end of the ring is the small white frame unit (see Figure 5.1). The middle point of the two white frame units in both top and bottom horizontal planes connect the loading transmission element which is the two robs across the ring spring and the straps. In order to make sure that the tensile force from the ring spring is distributed equally in both the top and bottom transmission robs, a vertical located rob is applied to make a rigid connection avoiding robs dislocated in practice.

There are three joints allocated in the white frame which can rotate itself around the middle joint in case of pipe incline. Rest of the joints, are connected to the two steel straps. Each strap pulls the pipeline when fault starts to move and ring spring behaves in its material property through wrapping half surface of the pipeline within its length.

Figure 5.1: A pipe surrounding with ring springs.
The positions and the direction of motion of the ring springs are arranged the same as the designed tests. The direction of fault movement is the same for ten tests, so the setting location orientation (see Figure 5.3, Figure 5.4) of the ring spring devices is all consistent, whereas, each test has its own ring spring arrangement plan included the size and the setting coordinate which is always symmetric to the central line of the pipe. Ring springs from both two pipe types are marked in an ascending order (1-6) from the inside which is next to the central line of the pipe to the outer side as shown in Figure 5.3-5.4. Detailed coordinate of ring spring central line (same coordinate to the middle joint) in the length direction for the ten full scale tests will be listed in Table 5.2.

Two kinds of steel straps and their corresponding connected white frame device are used in the tests. The dimensions of the strap width are depending on the pipeline diameter: for pipes from type1 the straps width is 80mm; for pipes from type2, 120 mm was employed. The spacing distance for the two straps (two end joints distance) is listed in Table 5.3. The contacted surface within the strap width of the pipe is bearing the load induced by the ring spring motion.

Figure 5.2: Schematic diagram of the connection for one spring before (left) and after (right) the final displacement under fault movement. The white rectangle in the figure is the “white frame unit”, a connection device in test setup.

Figure 5.3: Overview of ring spring and pipeline connections in Test 1 with the pipe type 1. (Source: Van Es, 2015)

Figure 5.4: Overview of ring spring and pipeline connections in Test 2 with the pipe size 2. (Source: Van Es, 2015)
Table 5.2: Coordinate in the pipe length direction of each ring spring in ten tests. [mm]

<table>
<thead>
<tr>
<th>No.</th>
<th>L6</th>
<th>L5</th>
<th>L4</th>
<th>L3</th>
<th>L2</th>
<th>L1</th>
<th>R1</th>
<th>R2</th>
<th>R3</th>
<th>R4</th>
<th>R5</th>
<th>R6</th>
</tr>
</thead>
<tbody>
<tr>
<td>T1</td>
<td>-5250</td>
<td>-4000</td>
<td>-3000</td>
<td>-2200</td>
<td>-1400</td>
<td>-700</td>
<td>700</td>
<td>1400</td>
<td>2200</td>
<td>3000</td>
<td>4000</td>
<td>5250</td>
</tr>
<tr>
<td>T2</td>
<td>-7000</td>
<td>-5500</td>
<td>-4000</td>
<td>-2900</td>
<td>-1900</td>
<td>-850</td>
<td>850</td>
<td>1900</td>
<td>2900</td>
<td>4000</td>
<td>5500</td>
<td>7000</td>
</tr>
<tr>
<td>T3</td>
<td>-6000</td>
<td>-4250</td>
<td>-3250</td>
<td>-2350</td>
<td>-1550</td>
<td>-750</td>
<td>750</td>
<td>1550</td>
<td>2350</td>
<td>3250</td>
<td>4250</td>
<td>6000</td>
</tr>
<tr>
<td>T4</td>
<td>-4750</td>
<td>-3500</td>
<td>-2600</td>
<td>-1875</td>
<td>-1150</td>
<td>-580</td>
<td>580</td>
<td>1150</td>
<td>1875</td>
<td>2600</td>
<td>3500</td>
<td>4750</td>
</tr>
<tr>
<td>T5</td>
<td>-5000</td>
<td>-3750</td>
<td>-2900</td>
<td>-2100</td>
<td>-1350</td>
<td>-600</td>
<td>600</td>
<td>1350</td>
<td>2100</td>
<td>2900</td>
<td>3750</td>
<td>5000</td>
</tr>
<tr>
<td>T6</td>
<td>-5125</td>
<td>-3950</td>
<td>-2900</td>
<td>-2100</td>
<td>-1350</td>
<td>-650</td>
<td>650</td>
<td>1350</td>
<td>2100</td>
<td>2900</td>
<td>3950</td>
<td>5125</td>
</tr>
<tr>
<td>T7</td>
<td>-8000</td>
<td>-6500</td>
<td>-5900</td>
<td>-4050</td>
<td>-2700</td>
<td>-1050</td>
<td>1050</td>
<td>2700</td>
<td>4050</td>
<td>5900</td>
<td>6500</td>
<td>8000</td>
</tr>
<tr>
<td>T8</td>
<td>-7000</td>
<td>-5500</td>
<td>-4000</td>
<td>-2900</td>
<td>-1900</td>
<td>-850</td>
<td>850</td>
<td>1900</td>
<td>2900</td>
<td>4000</td>
<td>5500</td>
<td>7000</td>
</tr>
<tr>
<td>T9</td>
<td>-8000</td>
<td>-6250</td>
<td>-5250</td>
<td>-3700</td>
<td>-2200</td>
<td>-850</td>
<td>850</td>
<td>2200</td>
<td>3700</td>
<td>5250</td>
<td>6250</td>
<td>8000</td>
</tr>
<tr>
<td>T10</td>
<td>-6800</td>
<td>-5250</td>
<td>-3800</td>
<td>-2500</td>
<td>-1600</td>
<td>-800</td>
<td>800</td>
<td>1600</td>
<td>2500</td>
<td>3800</td>
<td>5250</td>
<td>6800</td>
</tr>
</tbody>
</table>

Table 5.3: Detailed dimensions of the connection between ring spring and pipe. [mm]

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Pipe Type</th>
<th>Strap spacing</th>
<th>Strap width</th>
</tr>
</thead>
<tbody>
<tr>
<td>T1</td>
<td>1</td>
<td>350</td>
<td>80</td>
</tr>
<tr>
<td>T2</td>
<td>2</td>
<td>350</td>
<td>120</td>
</tr>
<tr>
<td>T3</td>
<td>1</td>
<td>350</td>
<td>80</td>
</tr>
<tr>
<td>T4</td>
<td>1</td>
<td>240</td>
<td>80</td>
</tr>
<tr>
<td>T5</td>
<td>1</td>
<td>350</td>
<td>80</td>
</tr>
<tr>
<td>T6</td>
<td>1</td>
<td>350</td>
<td>80</td>
</tr>
<tr>
<td>T7</td>
<td>2</td>
<td>350</td>
<td>120</td>
</tr>
<tr>
<td>T8</td>
<td>2</td>
<td>350</td>
<td>120</td>
</tr>
<tr>
<td>T9</td>
<td>2</td>
<td>350</td>
<td>120</td>
</tr>
<tr>
<td>T10</td>
<td>2</td>
<td>350</td>
<td>120</td>
</tr>
</tbody>
</table>

5.3.2 The connections between springs and pipeline in model

Reference point is a special node which can be defined at any location in the numerical model program. It is generally employed in the finite element analysis to make a force application point or the connection of some complicated elements. In the simulation model for full scale tests, several kinds of reference points are produced to simplify the ring spring connection device in the model. There are three reference points in total to simulate the three joints of the white frame in practice. Two reference points transfer the load (1/2F in Figure 5.5) from the connection device to the pipe, called the “slave points”. A reference point named “control point” is set in the middle to simulate the middle joint. This control point transfers the load (F in Figure 5.5) from the ring spring to the connection device.

The intermediate “control point” is linked to another reference point very far away from the pipeline through a nonlinear behaviour unit (ring spring device in practice) directly. That reference points formed the boundary of the moving box (see top and bottom frames in Figure 5.6).
Figure 5.5: Left: Load introduction in pipe cross section by flexible steel straps. Right: spread of load over two straps by divider beam. (Source: Van Es, 2015)

Figure 5.6: Several ring springs and their connections during fault movement in test.

For simulating the connection device in the numerical model, a proper relationship between the two kinds of reference points needs to be created. A coupling constraint method is chosen in ABAQUS. The two reference points in the two sides next to the middle control point was constraint by Continuum Distributing Coupling and set free the UR1, UR2 and UR3 degrees to make sure the two side point can rotate itself.

Then it comes to the constraint method between these two slave reference points and straps. Every strap encloses half of the pipeline perimeter (see Figure 5.5). Building all the straps in the model and making the surface to surface contact is very complicated for the finite element program to converge. So the slave points are used to control some specific area of the pipeline to transfer the load induced by the fault movement. From the experimental design report (Van Es, 2015), Continuum Distributing Coupling constraint between the reference point and a set of nodes is applied to replace the behaviour of the straps. The same constraint conditions to make the straps can rotate themselves as well. However, the way how load is transferring and the contact area from the straps to pipe are hard to say in the experimental tests. Therefore, two ways are tried in the first test to see which one is better for applying in the later simulations.
One method is to make the circle (red line in Figure 5.7) in the middle perimeter line in the length of strap width couple to the reference point in the centre of the pipe cross section circle which is defined as the slave point. This constraint way is called “C1” in the report.

The other way is to make two short lines (red line in Figure 5.8) within the length of strap width couple to the reference point in the centre of the pipe cross section circle. These two lines are in the neutral lines on both top and bottom pipeline wall. This constraint way is called “C2” in the report.

Both of the two simulating ways will lead a tensile force in the semi-circle area on one side and a compressive force in the semi-circle area on another side. The schematics from the side and top views are showed in the Figure 5.7 and Figure 5.8 separately. The comparison results from the contrast experiments of using these two coupling methods will be shown in Figure 5.9. The result is the relationship between axial strain in the front neutral line of the pipeline when the displacement of fault movement is 400 mm and 1200 mm.

The second method could always possess a more stable pipe strain response than the first coupling method. For a more systematical analysis in this series of tests, “C2”, the second coupling method, will be the only coupling constraint method applied on all the ten test simulations in this report.
Figure 5.8: The second coupling method for simulating the straps, C2.
Figure 5.9: Axial strain along the pipe length with two kinds of coupling method in Test 1, Mesh size=15, Integration points=9, front side.

C1-Control point is coupled by the middle circle of the straps
C2-Control point is coupled by the top and bottom neutral line of the strap range
G-the static, general method; others are static, modified Riks method in ABAQUS

5.3.3 The way of simulating the properties for springs

The soil pressure under the condition of relative displacement of soil mass and pipeline has an obviously nonlinear characteristic which has been discussed in Chapter 2.2. Hence, the ring spring devices will behave in this nonlinear property during testing. In finite element analysis program, there are two ways to get this kind of nonlinear behaviour in an element:

-By modifying the input file to add the nonlinear spring;
-By building the axial connectors.

Both of the two methods need the ring springs’ deformation-force relationships from test results, as the material properties of the springs or connectors. The first method has some troubles with the editing input file which is less convenient than modification in the user interface. It can only be defined as the nonlinear elastic springs which is not the ideal property to the ring spring devices. Therefore the more flexible and compatible instrument, the basic axial connector is employed for simulating this nonlinear ring spring property.

Refer to the tests that local buckling occurred in the critical part; a sudden energy release shows up at the moment local buckle occurs. In the experimental tests, the interaction of the pipeline and the ring spring is achieved through the contacting surfaces of straps. So this release happened naturally by a little gap in an extremely short period of time between the contact surfaces. Then the force in this unloading period will decrease immediately according to the plastic springback properties of the ring springs. However, in the FEA model, since either the slave points or the control points are constraint to the points far away from the pipeline by the distributing coupling, a proper plastic property setting of the axial connector should be taken to have this energy release.

In order to make connector behave in its elastoplastic property as the ring spring did in the real test, both of elastic and plastic properties should be applied in the connector section property. The definition of the elasticity and plasticity of the axial connectors in ABAQUS has a same algorithm to the definition of the definition of the material properties. The F-U relationship of the connector is calculated measured ring spring F-U relationship during the experimental test. In the elasticity stage of the axial connector, a parameter “D11” is applied to determine the gradient of the F-U curve. In the plasticity stage, the yield force and corresponding plastic motion will be calculated as following:

\[ F_{y0} = D_{11} \cdot U_{E0} \]  \hspace{1cm} (5.1)

\[ U_{E} = \frac{F_{y}}{D_{11}} \]  \hspace{1cm} (5.2)
\[ U_p = U - U_E \]  
\[ (5.3) \]

Where

- \( F_y \): The first yield force in the plasticity stage.
- \( F_y \): The force applied in the connector, as the following yield force.
- \( U_{E0} \): The displacement at the end of elasticity stage.
- \( D_{11} \): Curve gradient in the elasticity stage.
- \( U_p \): Plastic motion.
- \( U_E \): Elastic motion.

For the reason that all of the 12 ring springs were designed to have no compressive resistance capacity, and it is found from the first FEA model calculation trial that in several tests, the third or the fourth ring spring will have the compressive force at the beginning and then to their tensile resistance if linear elastic properties is applied in the connector. So the nonlinear elastic property and the plastic property should be worked together to get a same force-deformation relationship from the test results in the third or fourth ring spring. The difference working principles between linear elastic and nonlinear elastic is shown in Figure 5.10. The way of development of this kind of connectors will be introduced in the Appendix D, “Model building in ABAQUS” of this report.

A simpler linear elastic and plastic property of axial connector is applied for the rest of the ring springs. For the fifth and sixth ring spring which is stiffer and far from the fault movement central line, only the elastic part is appeared in the experiment results. That is to say, in the same time during the unloading process rings are still in their elastic stage; hence, just linear elastic property is applied for these ring springs. In principle, a simpler numerical model makes an easier calculation convergence. The flowchart for designing the connector from the first step to the last is shown in Figure 5.11.

Figure 5.10: Left: Connector linear elastic and plastic response. Right: Nonlinear elastic and plastic response. (Source: ABAQUS/Standard Analysis User’s Manual, 6.10)
Figure 5.11: Flowchart for building a connector.

A comparison of the F-U relationship from the measured test result and the finite element analysis output result for the second ring in test 9 is shown in Figure 5.12. “Target” is the pipe-soil interaction behaviour for designing the test; “T9R2L” and “T9R2R” represent the left and right ring spring behaviour measured during test; “Input data” is the designed data for numerical model; “L2” and “R2” is the left and right ring spring behaviour result from the numerical analysis. The results from numerical model both have the unloading and reloading stage at different ring deformations which depends on the moment when local buckling appeared.

Figure 5.12: Comparison results of ring force-axial deformation relationship from test and FEA model in Test 9 Ring2.
5.4 Internal pressure in the pipeline

5.4.1 The reaction of pipe internal pressure

Considering a straight section of pipe filled with a pressurized liquid or gas, internal pressure is another controlled aspect in GIPIPE project. In the global coordinate system, the definitions of the stress and strain coordinate system in the pipe are shown in Figure 5.11. The axial strain is output in the 22 direction and the circumferential strain is the output data in the 11 direction. Internal pressure existed in the internal surface of the pipeline will lead a strain in both the axial and circumferential directions, some extra strain gauges will be set for measuring the circumferential strain to investigate the internal pressure in the test. Axial strain along the pipe is arisen from the internal pressure, fault movement and the normal force in the two ends of the pipe.

![Internal pressure in the pipeline](image)

**Figure 5.13: Coordinate system in program**

The pipelines in Test 5, Test 6, Test 7 and Test 10 are applied the internal pressure with the value of 60bar, 120bar, 78bar and 39bar separately in the experiments. In the results from Gresnigt’s bending moment and internal pressure test (1986) the internal pressure has a positive effect on the magnitude of the curvature at which buckling of the pipe occurs. In the pipes considered there, for low values of P, buckling occurs fairly suddenly when the curvature increases. It is something of a “snap-through” effect. The buckled shape is sharp and angular. On the other hand, with high internal pressure in the pipe, buckling occurs much more gradually. Based on this curvature analysis, internal pressure could slow down the pipeline buckling shape during bending.

The inner portion of the pressurized pipe is exposed to same pressure in all the directions. The pressure force acts in the normal direction to the surface. Due to this the pipe wall is stretched in all directions. The wall of the pressurized pipe generally undergoes triaxial loading. From the symmetry of the circular cross section, there are two principal stresses, axial and circumferential, developed uniformly along the circumference of the pipe wall.
The internal pressure generates three principal stresses in the pipe wall, as illustrated in Figure 5.14: a hoop stress $\sigma_h$ (also referred to as circumferential or tangential stress, $S_{11}$ in ABAQUS), a longitudinal stress $\sigma_l$ (also referred to as axial stress, $S_{22}$ in ABAQUS), and a radial stress $\sigma_r$. The radial stress magnitude for a pipe is equally everywhere on the inside surface, and zero on the outside surface. The circumferential stress and the longitudinal stresses are usually much larger for pressure vessels, and so for thin walled instances the radial stress is usually neglected. When the ratio of the pipe diameter to its wall thickness $D/t$ is greater than 20 the pipe may be considered to be thin walled.

When analyzing the longitudinal stress of thin walled pressure pipe it is assumed that all stress act parallel to the surface of the pipe. In the pressurized cylinder in Figure 5.15, let a cut be made by slicing the cylinder in vertical direction. In order to satisfy equilibrium, for this free body diagram, there must be some force which counteracts the internal pressure. The only stress acting on the cut which counteracts the internal pressure is the normal stress $\sigma_1$.

The areas acted on by the longitudinal stress and the pressure are calculated in equation (5.4) (5.5). The Figure represents the fact that the force caused by the internal pressure must be equilibrated by the force caused by the longitudinal stress. As stress (and pressure) is expressed in units of force over area, to include stresses force (and pressure force) in force equilibrium equations, one must multiply the stress (and pressure) times the area on which it acts, equation(5.6) (5.7). In order to be cylinder stationary, stress forces balances the pressure forces, making equation (5.8) appear. This set of equations represents several variations of the formulas that might appear in different articles on this subject. The first expression is the most accurate one, whereas the last expression is the most conservative one yielding the highest stress. For simplification of the calculation in the whole model, last expression could help to conservatively get the longitudinal stress for pipes under internal pressure. Due to a high longitudinal stress, pipe may split up along its circumferences leading the damage.
\[ A_1 = \pi R^2 - \pi r^2 \]  
(5.4)

\[ A_2 = \pi r^2 \]  
(5.5)

\[ F = (\pi R^2 - \pi r^2) \cdot \sigma_l \]  
(5.6)

\[ F = \pi r^2 \cdot P \]  
(5.7)

\[ \sigma_l = \frac{\pi r^2 \cdot P}{\pi(R + r)(R - r)} = \frac{\pi r^2 \cdot P}{\pi(R + r) \cdot t} < \frac{(R + r)^2 \cdot P}{(R + r) \cdot t} = \frac{(R + r) \cdot P}{4t} < \frac{R \cdot P}{2t} \]  
(5.8)

Where

- \( A_1 \) Longitudinal stress applying area
- \( A_2 \) Internal pressure flow applying area
- \( R \) Outer radius
- \( r \) Inner radius
- \( P \) Internal pressure
- \( \sigma_l \) Longitudinal stress
- \( F \) Force acts in longitudinal direction
- \( t \) Pipe wall thickness

The same pressure also exerts the force in the diametrical direction, see Figure 5.16. The pressure acts on the element in the vertical direction. As drawn in figure the pressure tends to inflate the element focusing on forces which act in the vertical direction. The only stress acting in the vertical direction which can counteract the pressure shown left is the normal stress \( \sigma_h \). Calculating the forces on the elemental area of the cross section, force equilibrium is got from the inner portion of the pipe along the diameter and the area along pipe wall thickness with elemental length \( l \).

\[ A_3 = 2 \cdot l \cdot t \]  
(5.9)

\[ A_4 = 2r \cdot l \]  
(5.10)
\[ F = 2 \cdot l \cdot t \cdot \sigma_h \]  
(5.11)

\[ F = 2r \cdot P \]  
(5.12)

\[ \sigma_h = \frac{r \cdot P}{t} \]  
(5.13)

Where

- \( A_3 \) Vertical stress applying area
- \( A_4 \) Internal pressure flow applying area
- \( R \) Outer radius
- \( r \) Inner radius
- \( P \) Internal pressure
- \( \sigma_h \) Hoop stress
- \( F \) Force acts in vertical direction
- \( t \) Pipe wall thickness

On the inner portion of the pipe, along the diameter, with the elemental length \( l \) the pressure force projected area can be got. Along the pipe wall thickness \( t \) and elemental length \( l \) the force from the hoop stress can be got, see equation (5.9) - (5.13). These relations assume a uniform hoop stress across the thickness.

In reality, the stress is not uniform and is greater near the inside surface. In the diametrical direction, the stress is higher at the inner surface and lower at the outside surface. To compensate for this non-uniform stress distribution, the design equation normally uses the outside radius instead of the inside radius. In this case, the hoop stress is nearly constant through the wall thickness equal to

\[ \sigma_h = \frac{R \cdot P}{t} \]  
(5.14)

And the longitudinal stress is also constant through the wall and equal to half the hoop stress

\[ \sigma_l = \frac{R \cdot P}{2t} \]  
(5.15)

The radial stress which is caused by internal pressure varies between a stress equal to the internal pressure at the pipe’s inner surface and a stress equal to the atmospheric pressure at the pipe’s external surface. It varies from \( P \) at the inner surface of the pipe to zero on the outer surface where bending stresses are maximized. For this reason, this stress component has traditionally been ignored during the stress calculations.

With the regards to the internal pressure during the tests, a steel plate which is welded to each of the two ends makes the pipeline closed in test setup. This installation way will lead to an extra axial force for the steel pipe against the endcap in each of the two ends (see Figure 5.17), like the force in cut cross section mentioned in the longitudinal stress calculation method. The magnitude of this axial force could be calculated from the internal pressure and the area of the inner roundness shape using the equation:
Take Test 6 as an example, for other tests, axial force resulting from internal pressure against the end flanges of the pipeline is listed in Table 5.4. Internal pressure is 120bar (12 N/mm²), the outside radius of the pipe is 110mm and the thickness of the pipe is 5.5mm, so the extra axial force will be:

\[ F_{p,\text{end}} = P \times A = P \times \pi \times (R - t)^2 \]

(5.16)

\[ F_{p,\text{end}} = 12 \times \pi \times (110 - 5.5)^2 = 411684N \]

(5.17)

**Figure 5.17: Additional axial force in the two ends from internal pressure.**

### 5.4.2 Addition of the internal pressure in model

The internal pressure P and the extra axial force \( F_{p,\text{end}} \) are the constant force during the whole test procedure. In the numerical model, either force or displacement is applied to the element gradually with the iteration in increments during one step.

Therefore, it is necessary to separate the constant and changed loads or other types of boundary conditions rather than apply the internal pressure together with the displacement of fault movement.

For the pipelines bear internal pressure, an additional step using general static loading force before the nonlinear analysis of the pipeline buckling is employed in the finite element model. During that step, approximately 10 increments set up the internal pressure to the inner face of the pipeline and the axial force to the ends of the pipeline. These formed load boundary conditions would be propagated and as constant parameters in the next nonlinear analysis step.

### 5.4.3 An intensify influence of the various thickness with internal pressure

An average diameter is applied for the outer layer along the whole pipeline in the finite element analysis model. Several homogeneous cross sections with various thicknesses for the shell elements are employed in the different segments. Without simulating the weld (neither the residual stress on girth welding nor the length of heat affect zone), the cross sectional area changes in the connection between the two pipe segments with two thickness magnitudes, resulting a small gap in the joint between the two inner surface
A sudden decreasing of stress and strain occurs when the cross section (or the diameter) becomes larger. A sudden increasing of stress and strain occurs when the cross section (or the diameter) becomes smaller. Especially with the increase of the fault displacement, the pipeline is experiencing its plastic property stage, the strain distinguish will be larger. In addition, this thickness distinguish could also result in an extra strains due to the wall bending.

Adding the internal pressure the strain in the longitudinal and circumferential directions will show this gap earlier and larger than the one without the internal pressure. However, in the experiment test, since welding part is always having its residual stress and the heat affect length which is not appear in the model, the distinguishes in the two sets of results will be little different in the range around the welding position.

To minimize this sudden change of strain, another way to build the model is the next trial which is using a constant diameter in the middle surface of the thickness along the pipeline, and making the different thickness cross section applied around the middle surface, see Figure 5.18(b). There is a smaller gap in the weld part at the outer and inner surfaces which is half of the previous inner surface gap at the joints. As well as the application of the internal pressure to the inner surface, distinguishes is smaller than the result of first method but still existed.

The first way for building the pipeline will result in high strains due to the thickness distinguish under wall bending, so the second way is a better solution to minimize this deviation in simulation. A comparison results from these two methods is shown in Figure 5.19 under fault movement \( u=1470 \text{mm} \) in Test 7, “FEA-top” means the same diameter in the outer layer of pipe; “FEA-middle” means the same diameter in the middle layer of pipe.

![Figure 5.18](image)

Figure 5.18: Overview of the joint in the part of diameter changing (a) with the same diameter in the outer layer of pipe; (b) with the same diameter in the middle layer of the pipe.
5.5 Axial forces in the two ends

Two hydraulic jacks at the ends of pipe apply the axial force during the experimental test. The design value of the axial normal force is a combination of the force due to pipeline deformation under large bending and the fault movement plane which is not perpendicular to the pipeline axis. In theoretical, compressive force occurs due to the angle of the fault movement plane is less than zero. However, it is hard to make hydraulic jack achieve in the compressive force in practice. Therefore, axial force keeps 0 kN until tensile force appears during the experimental design.

In the finite element static analysis, the magnitude of both displacement and load will rise with the increased increments in one analysis step. To control the axial load to increase in terms of experimental test, another nonlinear axial connector was applied in the model to achieve this type of loading. Through controlling the force-displacement relationship of the connector, the tensile force occurs and then increases at a specific moment. This moment is measured by displacement of the fault movement.

Take Test 7 for instance, before the fault movement is 360mm, there will be no axial force loaded on the pipeline. The axial connector in the simulation model should also make a constant 0 loading until the fault movement equals to 360 mm. An example graph is showed below in Figure 5.19, the “F1” and “F4” are a pair of axial force at the pipeline end. “FEA input” is the designed F-U relationship in numerical model according to F-U curve of F1 and F4.

The summary of how axial tensile force data is applied to pipelines in every test is shown in Table 5.4.
Table 5.4 Summary of the axial force applied in the two ends of the pipe

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Internal pressure [Bar]</th>
<th>$F_{p,end}$ [N]</th>
<th>$U_{fault}$</th>
<th>$F_{act}$</th>
<th>$F_{act}^2$ [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-</td>
<td>-</td>
<td>0</td>
<td>921000</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-</td>
<td>-</td>
<td>0</td>
<td>1650000</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-</td>
<td>-</td>
<td>1000</td>
<td>148000</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-</td>
<td>-</td>
<td>0</td>
<td>540000</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>60</td>
<td>205605</td>
<td>550</td>
<td>60000</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>120</td>
<td>411684</td>
<td>240</td>
<td>770000</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>78</td>
<td>940690</td>
<td>360</td>
<td>1020000</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-</td>
<td>-</td>
<td>0</td>
<td>1660000</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-</td>
<td>-</td>
<td>750</td>
<td>315000</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>39</td>
<td>4700057</td>
<td>720</td>
<td>450000</td>
<td></td>
</tr>
</tbody>
</table>

$U_{fault}$ is the displacement of fault movement when $F_{act}$ starts to apply to the pipe.

$F_{act}^2$ is the final tensile force applied at the two ends of the pipe in the model.

5.6 Linear buckling analysis

5.6.1 Overview of the initial imperfection

Linear buckling analysis can obtain the linear, elastic solutions of buckling shapes with respect to various buckling modes. This shape is result in a displacement distribution with 1 mm as the maximum value, see Figure 5.23. It detects the buckling of a structure when the structure’s stiffness matrix approaches a singular value.

In the test, two local buckles appear under the bending of pipeline. So it is necessary to create two imperfections in the simulation to make the local buckles occurs at the same positions as well. The initial imperfection is accomplished through incorporating a scaled elastic buckling mode got from linear perturbation to the later nonlinear buckling analysis model. The chosen elastic buckling mode depends on
the appearance of the shape in various modes.

There are two methods to obtain the initial imperfection, one is to build only one linear perturbation model, and the final pipeline involves two positions of imperfection from two linear buckling modes. The other method is to get these two initial imperfections using two linear buckling models.

The details of the test and FEA results are shown in Appendix C. The purple solid line is the test result local buckling position. The orange solid line is the result from the finite element analysis model. Some of them are the same as each other, but some may be apart from 1 or 2 elastic buckling waves. The spacing between the two peak values for the pipe type1 is approximately 100mm, and 150mm for the pipe type2. One wave deviation is about 1% tolerance.

5.6.2 Initial imperfections in one model

To achieve the two positions for tending the local buckling in one linear perturbation model, two particular lengths would be established to have the linear elastic buckling analysis. Through controlling the magnitude of UR2 of the pipe cross section centers in these particular lengths’ two ends, two pairs of moment boundary conditions are applied both on the left and right side of the fault.

After some trials, it is found that when the length range is too wide, the two inner points which is not the pair of designed moment will make a non-elastic linear buckling in between them. Usually, 800-1000mm for type 1 size pipe and 1000-1200 mm for the type 2 size pipe will be applied in the FEA linear elastic buckling analysis model, Figure 5.20 and Figure 5.21 are one of the two buckle parts for type 1 and pipe type 2 pipelines. From the first trial result, the final local buckling may be located one or two peak spacing length from the true local buckling position.

Figure 5.20: Linear elastic buckling shape under UR2=1 unit in Test 5 with deformation scale factor=200, right buckling part. Length range=1100mm.
5.6.2 Initial imperfections in two models

To make smaller differences between the peak values of the initial imperfection, a longer elastic linear buckling length is preferred in the FEA model, see Figure 5.22. The second method for using two linear perturbation models is applied. This method is aiming to get the pipeline with initial imperfections after the linear elastic buckling analysis from two separate models.

In case of the pipeline junction located in the elastic buckling analysis length, different thickness or Young’s modulus will end with a discontinuity imperfection displacement of the pipeline. To get an initial imperfection shaped in sinusoidal distribution or cosine distribution, same thickness and Young’s modulus are applied in the linear buckle model, this method is introduced in Appendix D.
Since there is no way for the two linear elastic buckling analysis lengths to influence each other in two models, longer length could be applied to get smaller difference between two adjacent peak values in this method. The magnitude of this length could be ranged from 5D to 6D, D is the average outer diameter of the pipeline, both for pipe type 1 and pipe type 2.

The comparison deformation results from the two length magnitudes are shown in Figure 5.23 for Test 2. The different from the adjacent peak values for the two models method is smaller than the one model method. Sometimes it is necessary to use the second method not only because to get a less difference between the peak values and also applying two pairs of moments together may lead a unstable shape in all the required modes.

Pipelines in test 2 and Test 8 are both type 2 size pipe. The real position at which local buckling occurred is almost symmetric to the central point of the pipe; the finite element model will be symmetric in mesh control, so does the coordinates of initial imperfections. Mesh size will decide the amount of the nodes; in the critical part whose length is no less than the linear elastic buckling distributing length, should be finer than the rest of the pipeline which may not have such deformations in the longitudinal or circumferential directions. The elastic buckling analysis length for other tests are all shown in the figures of Appendix C.

![Figure 5.23: Linear elastic buckling shape in the left pipeline part position in Test 2](image)

### 5.7 Boundary conditions and expected result

#### 5.7.1 Displacement boundary conditions

In the displacement control boundary condition, there are two kinds of displacement, one is in the transversal direction (U1) which is perpendicular to the pipeline axis and the other one is in the pipeline length direction (U3).

The fault movement in the test is achieved relying on the displacement of the moving box. In the numerical model, the far away reference points formed the profile of the moving box. U1 equals to the final displacement of fault movement and U2 U3 equal to zero. The other side on the fault plane is the fixing box,
making displacement of the reference points equal to zero in all the three directions, see Figure 5.24. On the other hand, for longitudinal direction boundary conditions, a reference point in the fault plane is fixed in the U3 direction. This reference point is located at central point of the pipe cross section coupled to the top and the bottom of pipe wall.

### 5.7.2 Load boundary conditions

For the axial force is increasing as a linear variation tendency with the growing displacement of the fault movement, a pair of interactive force will be applied in the two ends of the pipeline wall. If the axial force is not increasing as a linear variation tendency, then a connector interaction unit should be applied in numerical model to make the axial force change according to the designed value through controlling the axial displacement of the connector.

The nonlinear behaviour connectors which simulate the soil resistance acting as the ring springs between the boundary condition controlled reference points and pipeline are shown in Figure 5.24.

![Figure 5.24: Von Mises stress distribution in Test 9 finite element model under final positions.](image)

Furthermore, another situation is for the four internal pressurized pipelines. The internal pressure should be employed in the first step before the nonlinear static analysis step in FEA model.

In the first step, all the reference points far away from pipeline connected to the connectors should be fixed in the three directions, so does the reference points at the two ends of the pipeline. It is doing so in order to make the pipeline loaded the internal pressure in situ. The additional axial force $F_P$ as well as the internal pressure should be added as the load boundary conditions in the first step to propagate as a constant load in the next step.

### 5.7.3 Expected results

From the results of the experimental test, pipeline with a high axial tensile force at the two ends (like Test 1)
or have the internal pressure and a high axial force (like Test 6 and Test 7), nonlinear buckling analysis will result in a large strain in the axial direction rather than a pipeline local buckling. However, if the axial force is too large and the internal pressure applying is large as well, a rupture failure may show up in the tensile part of the pipe earlier than the local buckling in the compressive part (like Test 5). The local buckling shape of pressurized pipeline is always a bulge in the compressive critical part. Under a symmetrical soil-pipeline interaction properties condition, the local buckling position should always be symmetrical on the left and right side of the fault plane. But some asymmetrical girth welds distribution especially in the critical part and the real ring spring interaction unequally in the two sides will lead an asymmetrical local buckling positions and a different appearance moment of them.

5.8 Element type and mesh size

5.8.1 Constant mesh size distributing

Mesh convergence studies is a control factor for the whole model and an important link for the accuracy of the entire simulation. Linear quadrilateral S4R, a 4-node doubly curved thin or thick shell, reduced integration was applied in the model, see Figure 5.25. Finer mesh size means the more element numbers and the more calculation memory storage and time, the amount of elements in the pipeline of Test 1 for different mesh sizes are shown in Figure 5.26.

There are two reasons to define the mesh size: in the construction of the FEA model, many partitions will be applied to locate the connections from ring spring. To get a uniform distributing element, the common divisors of the length of these partitions are employed. The other reason is: with respect to the proportion of the pipeline diameter around 5%-20%. Four mesh sizes which are 40, 20, 15 and 10 were chosen for the trials in Test 1.

For the first test, no local buckle formed so a constant mesh size distributing is applied to analyze the influence on the several types of mesh size. In the following models, to have a more accurate strain result and a particular linear elastic buckling shape in the Buckle model, a finer mesh size around 3% of pipeline diameter will be used in the critical part in which the linear elastic buckling located, and a coarse mesh size (defined after the mesh sensitive study) distributed in the rest of the pipeline on which the deformation is not so much.
From the result comparison in Figure 5.27 and 5.28, two specific points are picked in the significant deformation part to compare the axial strain with different pipeline mesh sizes. They are the points away from middle point -1750 mm and -2250 mm, corresponding to the maximum axial strain equals to 0.6% and 1.2%. The position with larger axial strain (-1750 mm) could represent the critical part for the pipelines deformed with local buckling. “U” is the displacement of fault movement $U_{\text{fault}}$.

For the pipeline of which no local buckling occurred, a constant distributing with mesh size equaling to approximately 5% of the diameter is applied. Mesh size of 12 for pipe type 1 and 15 for pipe type 2 in the finite element analysis model of Test 1, Test 6 and Test 7. For the rest of tests, a local buckling would occur at the compressive part of the pipe cross section. An unequally mesh size distribution provides a more accurate calculation and analysis.

Figure 5.26: The quantity of elements with different mesh sizes.

Figure 5.27: Overview of the axial strain along the pipe length under different mesh sizes with fault movement equals to 1200mm in Test 1.
5.8.2 Unequally mesh size distribution

In the model including the initial imperfection trending to the local buckling, a finer mesh size is necessary to analyze the nonlinear behaviour of pipeline. Thus, in the finite element model, an unequally distributing mesh size for the pipeline could be applied. Fine mesh size should be used in the critical part with the linear elastic buckling range included.

It can be concluded that the difference is not small for local buckling part when the mesh size is larger than 10, so mesh size less than 10 for pipe type 1 (5% of diameter) would be used for the critical part with the large axial deformation. For the rest of pipeline with small deformation, a coarse mesh size (10% of diameter) can be applied depending on the comparison result figure (Figure 5.28). The mesh size ranging from 3% to 6% of pipeline diameter is a good choice for the first trial, and the mesh chosen for the pipe segments far from the fault location is coarser from 10% to 12% of diameter. An even-distributed transition section should be employed in between the two mesh size sections. Table 5.5 shows the mesh size under different proportion with regards to two kinds of pipe diameters.

Table 5.5: Overview of the mesh size based on proportion of diameter.

<table>
<thead>
<tr>
<th>Pipe Type</th>
<th>Pipe Diameter</th>
<th>Mesh size</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>2%</td>
</tr>
<tr>
<td>1</td>
<td>220 mm</td>
<td>4.4</td>
</tr>
<tr>
<td>2</td>
<td>406 mm</td>
<td>8.1</td>
</tr>
</tbody>
</table>

In the ten FEA models, in consideration of the partition dimension, mesh size equals to 8 applying in the length of critical part for pipe type1, mesh size 12 applying in the length of critical part for pipe type2. The mesh size for the rest of the pipeline could be applied the mesh size around 10% of diameter. The length of the critical part must no less than the length between the two ends of the elastic buckling load boundary conditions, and longer in the side close to middle point than the end sides close to the two ends of the pipeline.
5.9 Thickness layers control in program

5.9.1 Settings for the thickness layer

To find a proper number of shell section integration points which divide the S4R element into several layers along the shell thickness, different numbers of points is defined in Simpson’s rules for homogeneous shell section to pick up a proper and reliable integration point applying in the next ten FEA test models. It is said in the aspect of defining the shell section integration in ABAQUS Analysis User’s Manual that Simpson’s rule and Gauss quadrature are provided to calculate the cross-sectional behaviour of a shell. The number of section points through the thickness of each layer can be specified. The default number of section points should be sufficient for routine thermal-stress calculations and nonlinear applications (such as predicting the response of an elastic-plastic shell up to limit load). For more severe thermal shock cases or for more complex nonlinear calculations involving strain reversals, more section points may be required. By default, Simpson’s rule will be used for the shell section integration. The default number of section points is five for a homogeneous section and three in each layer for a composite section. Simpson’s integration rule should be used if results output on the shell surfaces or transverse shear stress at the interface between two layers of a composite shell is required and must be used for heat transfer and coupled temperature-displacement shell elements.

![Figure 5.29: 9, 11, 13 shell section integration points along the shell thickness.](image)

From the requirement of the test simulation, Simpson’s integration rule was chosen for defining section properties. 7, 9, 11, 13, 15 and 17 shell integration points are applied separately in 6 trial models for Test 2 to see the difference values of their finite element analysis axial strain result after final pipeline deformation. 7 shell section integration points included in the thin wall means there are 7 layers through the thickness of the pipe, 9 points means 9 layers, and so on, see Figure 5.29. The large number of integration points makes that more layers verge to the true stress slope distribution line through the thickness, one side is tension and the other one is compression. The axial strain or the circumferential strain obtained from the numerical model is the value of the middle one of these shell integration points to avoid the local compressive or tensile influence from the shell bending itself.

55
5.9.2 Influence of the amount of thickness layers

From above part we can draw attention to the thickness layers that the more integration points there will be more layers and the calculation will be more accurate but it will get more time and more storage space as well. In Test 2, with the situation under mesh size equals 8 for the critical part (with local buckling) and 20 for the rest of pipe, second coupling method and using the modified Riks static analysis, different integration points number range from 7 to 17 are chosen to compare the axial strain result.

As can be seen in the Figure 5.30, with different shell integration points, the difference values compared to the shell element with 17 points is getting larger under the decreasing number of integration points. With the displacement of fault movement $U_{\text{fault}}$ getting larger, the pipeline deformed more. The differences are not small any longer due to the occurrence of large axial strain which could be induced by local buckling in the critical part.

A proper layer number is necessary to be decided for the first trial in other test models and adding the points when a high accuracy needs to achieve. From the graphs below we can make the conclusion that for the great deformed pipeline, more integration points is necessary: at least 9 for pipe type 1 and 11 for pipe type 2 integration points will be the first choice in the numerical models.

![Figure 5.30: Differentials of axial strain with different integration points in Test 2. Couple method: C2, Mesh size: 20, 8 at critical part, Static analysis: Modified Riks method. Fault movement displacement: 400 mm and 1000 mm.](image-url)
5.10 Chapter conclusion

This chapter discussed the construction of the finite element analysis model.

The geometric and material properties of the pipe segments are built according to the data in Appendix A and Appendix B. The ring springs are simulated using an interaction tool named “connector” in finite element analysis program ABAQUS. The elastic and plastic property of the connector is defined by the force-deformation relationship measured during the test. To avoid the compression for the connector, a nonlinear elasticity and plasticity is applied in the full scale test numerical model. “Continuum Distributing Coupling” constraint in ABAQUS of reference points coupled each other in the degree of freedom U1 U2 and U3 is applied for all the joints put between the connectors and pipeline.

Axial normal force and internal pressure are the imposed load in the model. Displacement for the virtual moving box in the transverse direction and the fixed pipeline centre in the direction of pipe length are the restrained boundary conditions in the model.

The mesh size and the number of thickness layers would influence the accuracy of the numerical result. Linear quadrilateral S4R element type with finer meshed (3% of the diameter) in critical part, coarse meshed (10% of the diameter) in rest of pipeline is employed. At least 9 layers are applied through the shell element thickness.

This developed numerical model will be validated in case of the strain results agree well with the test results.
6 Results of Full Scale Tests model

6.1 Summary of imperfection amplitudes

Linear elastic buckling shape as the initial imperfection in pipe is a tendency for making the buckling just occur in the local buckling position measured in final deformed pipes in tests. The amplitude of the initial imperfection is a crucial factor that decides local buckling to occur in the finite element model. The value of the amplitude is decided by a scaled superposition of the selected linear elastic buckling modes. This part can be accomplished with a parameter called "scale factor" in the FEA program. For instance, a linear elastic buckling mode results in a peak value $U=1$ mm, then the amplitude of the initial imperfection will be $0.05$ mm when scale factor is $0.05$.

The scaled amplitude is the initial imperfection applied in the nonlinear analysis from deformation result of the elastic buckling analysis model. The local buckling will occur in an earlier fault movement with a bigger scale factor, and in a later fault movement with a smaller scale factor. As can be seen in Figure 6.1, "SF" is the scale factor, the local buckling occurs at first at a smallest fault movement when SF=0.2. With the decreasing of the scale factor, the local buckling deformed slowly. Finding the proper scale factor will make the local buckling happen at the same time as the experimental test. The time in this model means the same displacement of the fault movement.

![Figure 6.1: Scale Factor sensitivity analysis in RK54 of Test 9.](image)

Deviation between the FEA model and test result is got from the following equation:

$$\frac{U_{\text{fault from FEA}} - U_{\text{fault from test}}}{1500} \times 100\%$$

Where the “Ufault” means the fault movement at the occurrence of the local buckling, since the Ufault from FEA is smaller than test result, the deviation will be negative when scale factor is larger than $0.05$, see in Figure 6.2. Scale factor equals $0.05$ provides the smallest deviation from the test result, therefore, for Test 9, the scale factor for the initial imperfection is $0.05$. 

58
In some pipes, girth welds are arranged in the critical part where the maximum axial strain may occur to make the comparison with the undivided pipe segment. The ductility property of welding part is weaker than the seamless steel pipe segment. It can be proved in the result from the test that the side with girth weld will occur the local buckling earlier than the other side. In order to add this attribute as the initial imperfection, a bigger scale factor for welding side than the normal side should be applied in the pipe which satisfies the requirements mentioned above. The summary of the scale factor of the ten tests are shown in Table 6.1.

From the combined data as shown, and the sensitivity analysis result in the Test 9, scale factor equals to 0.03 is a good choice for pipe type 1 and 0.05 is a good choice for pipe type 2. The value is concluded as around 0.5% of the thickness of pipe wall. The differences for changing this value around 0.01 to 0.1 are always very small. For the girth weld part, a scale factor is about 2 to 4 times larger than the ordinary one could be chosen to get the same strain result from tests in this simplified simulation model.

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Type</th>
<th>Left Part</th>
<th>Right Part</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-</td>
<td>-</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0.03</td>
<td>0.03</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.03</td>
<td>0.05</td>
<td>Girth weld in right local buckling</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.03</td>
<td>0.03</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.15</td>
<td>0.03</td>
<td>Girth weld in left local buckling</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>-</td>
<td>-</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>2</td>
<td>-</td>
<td>-</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>2</td>
<td>0.05</td>
<td>0.10</td>
<td>Girth weld in right local buckling</td>
</tr>
<tr>
<td>9</td>
<td>2</td>
<td>0.05</td>
<td>0.05</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>2</td>
<td>0.03</td>
<td>0.03</td>
<td></td>
</tr>
</tbody>
</table>
6.2 Comparison results from test and FEA model

6.2.1 Test 1

In Test 1, no local buckling occurred, final deformed pipe are shown with von Mises stress distributing in Figure 6.4. The maximum tensile axial strain along the pipe length is approximately +2.8% and the maximum compressive strain is approximately -1.3% under final pipe deformation. There are three failure modes mentioned in the experimental test, they are medium strain (1%), high strain (3%) as a trending of tensile strain failure, and local buckling, see Table 6.2. The displacement of the fault movement is the measurement to mark the moment when these modes occurs, and in the test, the strain gauge which is the nearest one to the most deformed position is picked up to label the extreme values. To get a analysis with all the influence aspects are consistent, for the FEA model, the displacement when these failure modes occur are also picked up from the specific strain gauge.
For a more clear and sharp viewing of the strains, some strain gauges were picked up to show the result under different condition control (integration points, mesh size, coupling position, static method, etc.). The coordinates of the strain gauge in the most critical part in Test 1 are shown in the following schematics, Figure 6.6. The comparison results from the test and FEA model on these specific strain gauge positions are shown in Appendix E for all the ten tests.

Table 6.2: Overview of reached failure modes during Test 1.

<table>
<thead>
<tr>
<th>Failure mode</th>
<th>Ufault in test [mm]</th>
<th>Ufault in FEM [mm]</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strain</td>
<td>Medium (1%)</td>
<td>422</td>
<td>425</td>
</tr>
<tr>
<td></td>
<td>High (3%)</td>
<td></td>
<td>-</td>
</tr>
<tr>
<td>Local buckling</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

The axial strain along the pipe length in the longitudinal direction is shown in Figure 6.5. The finite element model result coincides with the test result, symmetrically to the middle point in the two sides of the pipeline. Some wave appears because of the coupling method in the program. These waves also appeared in the test.

Figure 6.5: Comparison of results: Axial strain in the front and back central line along the length under different displacement of fault movement in Test 1.
Figure 6.6: Coordinates of the strain gauge in two sides of the pipe in Test 1

6.2.2 Test 2

Figure 6.7: Left: Local buckling on left side of fault. Right: Local buckling on right side of fault.

Figure 6.8: Axial strain after local buckling for left pipeline part in Test 2.
In Test 2, local buckling occurred on both left and right sides of the pipeline shaping in bulge lobe in the middle and two side lobes, see Figure 6.7. Final deformed pipe are shown with axial strain distributing in Figure 6.8 and Figure 6.9. The maximum tensile axial strain along the pipe length is approximately +3.2% and the maximum compressive strain is approximately -10% at left local buckling position under final pipe deformation.

Three failure modes are set same to Test 1. From the strain curve obtained from test and FEA model, local buckle occurred when there is a peak value or an abrupt change of compressive strain. The fault movement displacement of local buckling failure mode is considered by the specific compressive strain gauge. The strain gauge is the nearest one (RK12 on the left and RK45 on the right) to the local buckle, see Table 6.3. The same way of finding the fault movement is applied to the other tests local buckling occurred. The coordinates of the strain gauge in the most critical part in Test 2 are shown in the following schematics, Figure 6.11. The comparison results from the test and FEA model on these specific strain gauge positions are shown in Appendix E.

**Table 6.3: Overview of reached failure modes during Test 2.**

<table>
<thead>
<tr>
<th>Failure mode</th>
<th>Ufault in test [mm]</th>
<th>Ufault in FEM [mm]</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strain</td>
<td>Medium (1%)</td>
<td>660</td>
<td>625</td>
</tr>
<tr>
<td></td>
<td>High (3%)</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Local buckling</td>
<td>L:910 R:840</td>
<td>L:870 R:780</td>
<td>RK12, RK45</td>
</tr>
</tbody>
</table>
Figure 6.10: Comparison of results: Axial strain in the front and back central line along the length under different displacement of fault movement in Test 2.

The axial strain along the pipe length in the longitudinal direction is shown in Figure 6.10. The two sides of pipeline are asymmetrical to the middle point, but the two series of results are almost identical. Some axial strains of the points in the compression local buckling positions are larger in the left side part. A sudden increment of axial strain in the thickness changing part is occurred in left side of pipe; and in the Young’s modulus changing an increment of axial strain is occurred in right side of pipe.
6.2.3 Test 3

Figure 6.11: Coordinates of the strain gauge in two sides of the pipe in Test 2

Figure 6.12: Up: Local buckling on left side of fault. Down: Local buckling on right side of fault.
In Test 3, local buckling occurred on left side of the pipeline shaping in a very small bulge lobe; local buckling occurred on right side of the pipeline at the girth weld, see Figure 6.12. Final deformed pipe are shown with axial strain distributing in Figure 6.13 and Figure 6.14. The maximum tensile axial strain along the pipe length is approximately +2.8% and the maximum compressive strain is approximately -16.5% at right local buckling position under final pipe deformation. The coordinates of the strain gauge in the most critical part in Test 3 are shown in the following schematics, Figure 6.16.

In Test 3, there is a distance (approximate 80 mm) between the result local buckling positions from numerical model and the test. It will lead a huge difference in strain when take the same coordinate system. The solution to get a more precise comparison, a same distance between the strain gauge and buckle position from the test is applied for the numerical model. Take RK12 (-2225 mm from the centre of pipeline) for instance, the axial distance between the left buckle position (-2240) and the strain gauge is 15 mm in experimental test. However, the left buckle occurred at -2310 during the numerical simulation. So the compared axial strain from numerical model will be obtained at -2295, with the same distance (15 mm) between the measurement and buckle position. This way is only for the two strain measurement positions next to the buckle, and the rest of the measurements from numerical model share a same global coordinate to the test strain gauge. The comparison results from the test and FEA model on these specific strain gauge
positions are shown in Appendix E. The same comparison method is also applied in Test 4, Test 5, Test 8, Test 9 and Test 10.

Table 6.4: Overview of reached failure modes during Test 3.

<table>
<thead>
<tr>
<th>Failure mode</th>
<th>Ufault in test [mm]</th>
<th>Ufault in FEM [mm]</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strain</td>
<td>Medium (1%)</td>
<td>960</td>
<td>980</td>
</tr>
<tr>
<td></td>
<td>High (3%)</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Local buckling</td>
<td>L:1250 R:810</td>
<td>L:1200 R:870</td>
<td>RK12, RK45</td>
</tr>
</tbody>
</table>

The axial strain along the pipe length in the longitudinal direction is shown in Figure 6.15. The finite element model result coincides with the test result in the left side of pipeline, asymmetrically for most of points in the critical part to the middle point in the two sides because of the existing of right girth weld. It is noteworthy that the formation of a buckle on the right side of the specimen apparently leads to an increased strain rate of the rest of the pipe. The experiment axial strain results show that the strains in the highly strained cross section on the left side of the fault start increasing more rapidly due to the formation of the local buckle on the right side of the fault. However, this phenomenon didn’t show up in the FEA model which is without the welding residual stress or heat affect zone.

The axial strain or the local buckling position results from FEA model are almost symmetric in both tensile and compressive sides of the pipeline. The right part occurred local buckling earlier than left side since the weld existed. If adding the residual stress or the influence welding produced in the right side girth weld part, results might be more identical for these two situations. There is a sudden change in the right side for the change of the pipe wall thickness on both front and back side of pipe. In the compression side of the right critical part, some tensile strains are shown under large Ufault in the test results, which may be procured by the large deformation of pipeline in the weak welding part.
Figure 6.15: Comparison of results: Axial strain in the front and back central line along the length under different displacement of fault movement in Test 3.

Figure 6.16: Coordinates of the strain gauge in two sides of the pipe in Test 3

6.2.4 Test 4
In Test 4, local buckling occurred on both left and right sides of the pipeline shaping in dent lobe, see Figure 6.17. Final deformed pipe are shown with axial strain distributing in Figure 6.18 and Figure 6.19. The maximum tensile axial strain along the pipe length is approximately +4% and the maximum compressive strain is approximately -17% at left and right local buckling positions under final pipe deformation. The coordinates of the strain gauge in the most critical part in Test 4 are shown in the following schematics, Figure 6.21. The comparison results from the test and FEA model on these specific strain gauge positions are shown in Appendix E.
Table 6.5: Overview of reached failure modes during Test 4.

<table>
<thead>
<tr>
<th>Failure mode</th>
<th>Ufault in test [mm]</th>
<th>Ufault in FEM [mm]</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strain</td>
<td>Medium (1%)</td>
<td>470</td>
<td>480</td>
</tr>
<tr>
<td></td>
<td>High (3%)</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Local buckling</td>
<td>L:690 R:800</td>
<td>L:670 R:840</td>
<td>RK12, RK44</td>
</tr>
</tbody>
</table>

The axial strain along the pipe length in the longitudinal direction is shown in Figure 6.20. The two sides of pipeline are almost symmetrical to the middle point and the axial strain in both front and back sides of the pipeline from test results and FEA mode results are almost identical.

![Figure 6.20: Comparison of results: Axial strain in the front and back central line along the length under different displacement of fault movement in Test 4.](image-url)
Figure 6.21: Coordinates of the strain gauge in two sides of the pipe in Test 4

6.2.5 Test 5

Figure 6.22: Left: Local buckling and rupture on left side of fault. Right: Local buckling on right side.

Figure 6.23: Axial strain after local buckling for left pipeline part in Test 5.
In Test 5, local buckling occurred on both left and right sides of the pipeline shaping in bulge lobe, see Figure 6.22. Final deformed pipe are shown with axial strain distributing in Figure 6.23 and Figure 6.24. In the experimental test, rupture failure occurred when fault movement equals to 710mm. So the final displacement of fault is applied 700mm in the Test 5. The maximum tensile axial strain along the pipe length is approximately +3.5% and the maximum compressive strain is approximately -20% at right local buckling position under final pipe deformation. The coordinates of the strain gauge in the most critical part in test 5 are shown in the following schematics, Figure 6.26. The comparison results from the test and FEA model on these specific strain gauge positions are shown in Appendix E.

**Table 6.6: Overview of reached failure modes during Test 5.**

<table>
<thead>
<tr>
<th>Failure mode</th>
<th>Ufault in test [mm]</th>
<th>Ufault in FEM [mm]</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strain</td>
<td>Medium (1%)</td>
<td>410</td>
<td>430</td>
</tr>
<tr>
<td></td>
<td>High (3%)</td>
<td>550</td>
<td>670</td>
</tr>
<tr>
<td>Local buckling</td>
<td>L:455 R:580</td>
<td>L:490 R:580</td>
<td>RK10, RK43</td>
</tr>
<tr>
<td>Rupture</td>
<td>710</td>
<td>-</td>
<td>Opposite of left buckle</td>
</tr>
</tbody>
</table>

The axial strain along the pipe length in the longitudinal direction is shown in Figure 6.25. The two sides of pipeline are almost symmetrical to the middle point except there is a sudden gap in left critical part due to the change of the wall thickness in FEA model. The axial strain in both front and back sides of the pipeline from test results and FEA mode results are almost identical.
Figure 6.25: Comparison of results: Axial strain in the front and back central line along the length under different displacement of fault movement in Test 5.

<table>
<thead>
<tr>
<th>Front</th>
<th>Back</th>
</tr>
</thead>
<tbody>
<tr>
<td>RK8  -1650</td>
<td>RK59 -1650</td>
</tr>
<tr>
<td>RK12 -1250</td>
<td>RK55 -1250</td>
</tr>
<tr>
<td>RK15 -900</td>
<td>RK52 -900</td>
</tr>
<tr>
<td>RK19 900</td>
<td>RK49 900</td>
</tr>
<tr>
<td>RK22 1250</td>
<td>RK45 1250</td>
</tr>
<tr>
<td>RK26 1650</td>
<td>RK41 1650</td>
</tr>
<tr>
<td>RK7  -1850</td>
<td>RK61 -1850</td>
</tr>
<tr>
<td>RK10 -1450</td>
<td>RK57 -1450</td>
</tr>
<tr>
<td>RK14 -1100</td>
<td>RK54 -1100</td>
</tr>
<tr>
<td>RK20 1100</td>
<td>RK47 1100</td>
</tr>
<tr>
<td>RK24 1450</td>
<td>RK43 1450</td>
</tr>
<tr>
<td>RK28 1850</td>
<td>RK40 1850</td>
</tr>
</tbody>
</table>

Figure 6.26: Coordinates of the strain gauge in two sides of the pipe in Test 5
6.2.6 Test 6

In Test 6, no local buckling occurred, final deformed pipe are shown with von Mises stress distributing in Figure 6.27. The maximum tensile axial strain along the pipe length is approximately +3.1% and the maximum compressive strain is approximately -1.5% under final pipe deformation. The coordinates of the strain gauge in the most critical part in Test 6 are shown in the following schematics, Figure 6.30. The comparison results from the test and FEA model on these specific strain gauge positions are shown in Appendix E.

For the pipes will the internal pressure, the circumferential strain is output as well from the FEA model to see how the internal pressure worked, see Figure 6.29. The axial strain along the pipe length in the longitudinal direction is shown in Figure 6.28. The two sides of pipeline are almost symmetrical to the middle point. The axial strain in both front and back sides of the pipeline from test results and FEA mode results are almost identical, so are the circumferential strain.

Table 6.7: Overview of reached failure modes during Test 6.

<table>
<thead>
<tr>
<th>Failure mode</th>
<th>Utfault in test [mm]</th>
<th>Utfault in FEM [mm]</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strain</td>
<td>Medium (1%)</td>
<td>450</td>
<td>440</td>
</tr>
<tr>
<td></td>
<td>High (3%)</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Local buckling</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>
Figure 6.28: Comparison of results: Axial strain in the front and back central line along the length under different displacement of fault movement in Test 6.
Figure 6.29: Comparison of results: Circumferential strain in the front and back central line along the length under different displacement of fault movement in Test 6.

Then is the axial strains from the strains gauge will show a more clear strain-U fault displacement relationship and for the internal force. These 4 tests applied the strain gauge in the circumferential direction to see the strain transformation from the internal pressure.

![Diagram showing strain distribution](image)

Figure 6.30: Coordinates of the strain gauge in two sides of the pipe in Test 6
In Test 7, no local buckling occurred, final deformed pipe are shown with von Mises stress distributing in Figure 6.31. The maximum tensile axial strain along the pipe length is approximately +1.55% and the maximum compressive strain is approximately -1.2% under final pipe deformation. The coordinates of the strain gauge in the most critical part in Test 7 are shown in the following schematics, Figure 6.34. The comparison results from the test and FEA model on these specific strain gauge positions are shown in Appendix E.

For the pipes will the internal pressure, the circumferential strain is output as well from the FEA model to see how the internal pressure worked, see Figure 6.33. The axial strain along the pipe length in the longitudinal direction is shown in Figure 6.32. The result of Test 7 from finite element analysis is lower than the experiment result in the range of part 2, the reason for this phenomenon may be the material property accuracy, welding residual stress and the force relationship with the ring springs, or the influences from the combination of them. With the various shell wall thickness, there is a sudden change in the left side of pipe. The avoidance way has been introduced in chapter 4.3.

<table>
<thead>
<tr>
<th>Failure mode</th>
<th>Utfault in test [mm]</th>
<th>Utfault in FEM [mm]</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strain</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Medium (1%)</td>
<td>971</td>
<td>1330</td>
<td>RK55</td>
</tr>
<tr>
<td>High (3%)</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Local buckling</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 6.8: Overview of reached failure modes during Test 7.
Figure 6.32: Comparison of results: Axial strain in the front and back central line along the length under different displacement of fault movement in Test 7.
Figure 6.33: Comparison of results: Circumferential strain in the front central line along the length under different displacement of fault movement in Test 7.

Figure 6.34: coordinates of the strain gauge in two sides of the pipe in Test 7

6.2.8 Test 8
Figure 6.35: Up: Local buckling on left side of fault. Down: Local buckling on right side of fault.

Figure 6.36: Axial strain after local buckling for left pipeline part in Test 8.

Figure 6.37: Axial strain after local buckling for right pipeline part in Test 8.
In Test 8, local buckling occurred on left side of the pipeline shaping in a very small bulge lobe; local buckling occurred on right side of the pipeline at the girth weld, see Figure 6.35. Final deformed pipe are shown with axial strain distributing in Figure 6.36 and Figure 6.37. The maximum tensile axial strain along the pipe length is approximately +10% and the maximum compressive strain is approximately -20% both of them occurred at the right local buckling position under final pipe deformation. The coordinates of the strain gauge in the most critical part in Test 8 are shown in the following schematics, Figure 6.39. The comparison results from the test and FEA model on these specific strain gauge positions are shown in Appendix E. The limitation of the strain gauge is 3.5%, so as the nearest one RK10 has exceeded the maximum measuring range, which would be replaced by RK11 to recognize the occurrence of local buckle on the left of the pipeline.

**Table 6.9: Overview of reached failure modes during Test 8.**

<table>
<thead>
<tr>
<th>Failure mode</th>
<th>UFault in test [mm]</th>
<th>UFault in FEM [mm]</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strain</td>
<td>Medium (1%)</td>
<td>646</td>
<td>550</td>
</tr>
<tr>
<td></td>
<td>High (3%)</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Local buckling</td>
<td>L:895 R:700</td>
<td>L:900 R:750</td>
<td>RK11, RK44</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Right buckle at girth weld</td>
</tr>
</tbody>
</table>

The axial strain along the pipe length in the longitudinal direction is shown in Figure 6.38. The right local buckling occurred at the girth weld which is much earlier than the other side local buckling by using a larger scale facto in FEA model. As can be seen from the results comparison, some differences take place in the right tensile part of the pipe. This gap is formed by the sudden changes in the wall thickness and material properties.
Figure 6.38: Comparison of results: Axial strain in the front and back central line along the length under different displacement of fault movement in Test 8.

Figure 6.39: Coordinates of the strain gauge in two sides of the pipe in Test 8
6.2.9 Test 9

Figure 6.40: Up: Local buckling on left side of fault. Down: Local buckling on right side of fault.

Figure 6.41: Axial strain after local buckling for left pipeline part in Test 9.
In Test 9, local buckling occurred on both left and right sides of the pipeline shaping in bulge lobe with two small side dent lobes, see Figure 6.40. Final deformed pipe are shown with axial strain distributing in Figure 6.41 and Figure 6.42. The maximum tensile axial strain along the pipe length is approximately +10% and the maximum compressive strain is approximately -40% both of them occurred at the left and right local buckling position under final pipe deformation. The coordinates of the strain gauge in the most critical part in Test 9 are shown in the following schematics, Figure 6.44. The comparison results from the test and FEA model on these specific strain gauge positions are shown in Appendix E. The axial strain along the pipe length in the longitudinal direction is shown in Figure 6.43.

Table 6.10: Overview of reached failure modes during Test 9.

<table>
<thead>
<tr>
<th>Failure mode</th>
<th>Ufault in test [mm]</th>
<th>Ufault in FEM [mm]</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strain</td>
<td>Medium (1%)</td>
<td>1050</td>
<td>980</td>
</tr>
<tr>
<td></td>
<td>High (3%)</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Local buckling</td>
<td>L:1165 R:1030</td>
<td>L:1150 R:1000</td>
<td>RK9, RK42</td>
</tr>
</tbody>
</table>
Figure 6.43: Comparison of results: Axial strain in the front and back central line along the length under different displacement of fault movement in Test 9.

Figure 6.44: Coordinates of the strain gauge in two sides of the pipe in Test 9

6.2.10 Test 10

Figure 6.45: Left: Local buckling on left side of fault. Right: Local buckling on right side of fault.
In Test 10, local buckling occurred on both left and right sides of the pipeline shaping only in bulge lobe, see Figure 6.45. Final deformed pipes are shown with axial strain distributing in Figure 6.46 and Figure 6.47. The maximum tensile axial strain along the pipe length is approximately +4.5% and the maximum compressive strain is approximately -24% at both left and right local buckling position under final pipe deformation. The coordinates of the strain gauge in the most critical part in Test 10 are shown in the following schematics, Figure 6.50. The comparison results from the test and FEA model on these specific strain gauge positions are shown in Appendix E.

**Table 6.11: Overview of reached failure modes during Test 10.**

<table>
<thead>
<tr>
<th>Failure mode</th>
<th>Ufault in test [mm]</th>
<th>Ufault in FEM [mm]</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile strain</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Medium (1%)</td>
<td>560</td>
<td>520</td>
<td>RK60</td>
</tr>
<tr>
<td>High (3%)</td>
<td>1070</td>
<td>1050</td>
<td>RK60</td>
</tr>
<tr>
<td>Local buckling</td>
<td>L:680 R:640</td>
<td>L:700 R:630</td>
<td>RK7, RK44</td>
</tr>
</tbody>
</table>

The axial strain along the pipe length in the longitudinal direction is shown in Figure 6.48. The circumferential strain is shown in Figure 6.49. The two sides of pipeline are almost symmetrical to the middle point except both of the two local buckling are apart one linear elastic buckling wave from the test local buckling result.
Figure 6.48: Comparison of results: Axial strain in the front and back central line along the length under different displacement of fault movement in Test 10.
Figure 6.49: Comparison of results: Circumferential strain in the front and back central line along the length under different displacement of fault movement in Test 10.

Figure 6.50: Coordinates of the strain gauge in two sides of the pipe in Test 10.

6.3 Chapter conclusion

This chapter is mainly about the result comparison from the tests and numerical models. Scale factor is an important parameter to determine the deformation of the local buckling. Through the axial strain of the neutral line in the front and back sides along the pipe length when fault movement is 200 mm, 250 mm, 400 mm, 500 mm etc., the result from FEA model coincides with the test result. More precisely, the axial strain for every strain gauge from the test and FEA model are shown in the figures of Appendix E. These comparison results are the two validation methods: strain-distance curve and strain-fault movement curve. After the local buckling, ring spring next to the local buckling position will unload for the moment and then
with the increasing displacement of fault movement, these ring springs will reload again. Strain will decrease after the appearance of the local buckling; the last point before this decreasing is the local buckling appeared time. That is the way to recognize the local buckling failure mode in the strain-fault movement relationship curve.

No buckling occurred in Test 1, Test 6 and Test 7 due to the large axial force at the two ends of pipeline and the applied internal pressure. Maximum tensile strains of all of the pipelines have exceeded 1%, Test 5 and Test 10 of them exceeded 3%; and local buckling appeared with large tensile and compressive strain on both left and right sides from the fault movement within different deformation. The local buckling location is not always symmetric in the test because some reasons, such as the residual stress of the girth weld part and its heat affect zone; the straps contact surface or the asymmetrical rotation of the straps from the ring spring devices on the two sides of the fault, or the various material properties from two adjacent pipe segments. Local buckling might locate from simulated model a little apart from the test result; these deviations are very little according to the entire pipeline length, see axial comparison figures in this chapter and figures in Appendix C.

The shape of local buckling is different for different pipelines. Buckle in bulge shape is always the final deformation of the pressurized pipe. Buckles for the rest pipes are formed in one long dent and two side dents in trail of pipe wall beside the long one.

From both the strain-distance relationship and the strain-fault movement relationship, the results agree with each other well. So the numerical model could be validated through investigating the tests results.
7 Conclusions and Recommendations

7.1 Conclusions

7.1.1 Nonlinear geometrical and material properties

This report analyzed buried pipeline under induced-ground deformation which might lead to the pipeline formed in local buckling under large bending. To simulate the pipeline under large deformation in the finite elements analysis, nonlinear geometrical and nonlinear material properties are applied.

The geometric dimensions for the pipe segments are sketched in Appendix A. Pipe segments behave in the plastic properties when local buckling occurs, as well as a period of time before buckle. Calculated nonlinear material properties of the pipe segments are obtained from the engineering stress strain relationship in tensile specimen tests, shown in Appendix B.

7.1.2 Pipe-soil interaction

In the experimental test, nonlinear behaviour property collapsible rings are used to represent the pipe-soil interaction. To achieve the same properties of devices the elastic and plastic behaviour “connector” is applied in the finite element model in order to obtain the energy release during pipeline deformation. If no compressive force is required for the ring springs then the “connector” with nonlinear elasticity and plasticity is applied. The properties of element “connector” in the numerical model are the force-displacement of the ring springs measured during the test.

For the connections between the ring spring devices and pipeline, reference points coupled each other in the degree of freedom U1 U2 and U3. “Continuum Distributing Coupling” is applied for all the joints between controlled reference points.

7.1.3 Initial imperfections

Without any geometrical imperfection measurements before the tests, amplitude of the imperfection which is decided by a scaled deformed shape from the linear elastic buckling analysis determines the deformation of the local buckling.

The amplitude is around 0.5% of the thickness of the pipeline wall which is established by a parameter named “scale factor” imported before the nonlinear buckling analysis. The adjustment is needed after analyzing the relationship between axial strain and fault movement; a larger scale factor leads to the local buckling occurs earlier and a smaller scale factor postpones the local buckling occurrence. Sometime it should be 2-4 times larger at the girth weld side to get an early local buckling without simulating the girth weld in the welding pipeline part.
7.1.4 Advanced numerical model for investigating full scale tests

During the pipeline deformation, there might be two local buckling parts on the different side of the pipe, so the length range for the linear elastic buckling analysis and the linear buckling loading application should be considered with regards to two aspects: the distance local buckling from fault and the diameter of the pipes. Usually, 800-1000 mm for type 1 size pipe and 1000-1200 mm for the type 2 size pipe. If it is impossible or the elastic buckling shape is not the ideal shape, applied the two initial imperfection positions in one model should be replaced by two models. The linear elastic buckling analysis length for the second method is 5D to 6D, applied in Test 3, Test 5 and Test 8.

Boundary conditions of numerical model are the imposed load and displacement. Axial normal force and internal pressure are the imposed load in the model. Displacement around 1500 mm for the virtual moving box in the transverse direction and the fixed pipeline centre in the direction of pipe length are the restrained boundary conditions.

The mesh size and the number of thickness layers would influence the accuracy of the numerical result. Linear quadrilateral S4R element type with finer meshed (3% of the diameter) in critical part, coarse meshed (10% of the diameter) in rest of pipeline is employed. At least 9 layers are applied through the shell element thickness of pipeline.

7.1.5 Conclusions from result comparison

As the result from numerical analysis: no buckling occurred in Test 1, Test 6 and Test 7. Maximum axial tensile strains of all of the pipelines have exceeded 1%, Test 5 and Test 10 of them have exceeded 3%; and local buckling appeared with large tensile and compressive strain on both left and right sides from the fault movement within different deformation. The local buckling location is not always symmetric in the test. For the pipeline length is approximately 20 m, local buckling always happens at the distance around ±2000 mm to ±3000 mm from fault movement central line. In generally, the ring spring close to the centre will support the largest lateral soil-pipeline interaction force under the fault movement. The local buckling is always occurred at the peak imperfection value. For pipe type 1, spacing distance between two peak values of linear elastic analysis result is 100 mm which is smaller than 150 mm resulted from pipe type 2.

The shape of local buckling is different for different pipelines. Buckle in bulge shape is always the final deformation of the pressurized pipe. Buckles for the rest pipes are formed in one long dent and two side dents in trail of pipe wall beside the long one, or only in one dent or one bulge shape.

A finite element analysis model with the correct properties, boundary conditions and parameter settings in every single part of it will give the same reaction and result to the experimental test. The step of adding the initial imperfection is necessary for the pipes that have local buckling. From both the strain-distance relationship and the strain-fault movement relationship, the results from tests and numerical models agree with each other well. Through investigating the full scale tests results, the developed numerical model is validated.
7.2 Recommendations for further studies

- The measurement of the geometrical imperfection of the pipeline is necessary to decide the scale factor in the FEA model. Otherwise, many trials should be taken to find a proper scale factor.

- For getting a more accurate result, residual stress along the whole pipeline during the fabrication period and the residual stress on the welding part should be calculated and applied in the FEA model.

- An improved model could be built after considering putting the girth weld at the crucial positions. However, it is still need some model designing method to get much closer to practice, especially these girth welds will lead a various results of the axial strain and the circumferential strain.

- Considering the two methods for the addition of initial imperfections mentioned in section 5.6, the way to use two models is a better solution in the numerical analysis rather than the one model way. Improvements for the linear elastic analysis could be taken in Test 2, Test 4, Test 9 and Test 10.

- Result comparison of the ovalisation between test and FEA models.

- Apply the geometrical imperfection to the pipeline in which no local buckling occurred. Make sure that no local buckling occurred as well during the nonlinear buckling analysis of an imperfect pipeline under fault movement.

- The full scale test is designed to investigate the buried pipeline deformation using 12 ring springs as the pipe-soil interaction. And the established numerical model has shown the same result to the tests. For a simulation more close to actual, pipeline could be surrounded by infinite nonlinear behaviour springs on both the transverse sides (almost like the continuous soil model). If the simulated result well agrees with the existed 12 springs test result, it is proven that this designed test model with discrete springs could represent the buried pipeline in practice.
8 Reference


Federal Emergency Management Agency (FEMA) and the American Society of Civil Engineers (ASCE) (2001), Guidelines for the Design of Buried Steel Pipe. American Lifelines Alliance.


Gresnigt, A. M. (1987), Plastic Design of Buried Steel Pipelines in Settlement Areas. HERON, Vol. 31 no.4


O’Rourke, M. J., & Liu, X. (1999), Response of Buried Pipelines Subject to Earthquake Effects. *Multidisciplinary Center for Earthquake Engineering Research*.


Zhao, L., Cui, C., & Li, X. (2010), Response analysis of buried pipelines crossing fault due to overlying soil

## Appendix A: Specimen layout and properties

Table 2.3: The summary of specimen material and geometric properties of each part of the pipelines. [Length dimension: mm, Stress dimension: MPa]

<table>
<thead>
<tr>
<th>No.</th>
<th>Part1</th>
<th>Part2</th>
<th>Part3</th>
<th>Part4</th>
<th>Part5</th>
<th>Part6</th>
<th>Overall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test 1</td>
<td>D</td>
<td>220.06</td>
<td>219.88</td>
<td>219.96</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>t</td>
<td>5.6</td>
<td>5.6</td>
<td>5.5</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_{0.2}%$</td>
<td>567MPa</td>
<td>571MPa</td>
<td>591MPa</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_t$</td>
<td>607MPa</td>
<td>612MPa</td>
<td>646MPa</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Test 2</td>
<td>D</td>
<td>406.94</td>
<td>407.15</td>
<td>406.90</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>t</td>
<td>7.59</td>
<td>7.53</td>
<td>7.53</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_{0.2}%$</td>
<td>552MPa</td>
<td>562MPa</td>
<td>535</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_t$</td>
<td>597MPa</td>
<td>630MPa</td>
<td>592MPa</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Test 3</td>
<td>D</td>
<td>220.10</td>
<td>220.01</td>
<td>219.98</td>
<td>219.89</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>t</td>
<td>5.51</td>
<td>5.56</td>
<td>5.56</td>
<td>5.49</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_{0.2}%$</td>
<td>567MPa</td>
<td>594MPa</td>
<td>571MPa</td>
<td>591MPa</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_t$</td>
<td>607MPa</td>
<td>653MPa</td>
<td>612MPa</td>
<td>646MPa</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Test 4</td>
<td>D</td>
<td>220.04</td>
<td>220.19</td>
<td>220.62</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>t</td>
<td>5.51</td>
<td>5.54</td>
<td>5.53</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_{0.2}%$</td>
<td>567MPa</td>
<td>598MPa</td>
<td>591MPa</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_t$</td>
<td>607MPa</td>
<td>650MPa</td>
<td>646MPa</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Test 5</td>
<td>D</td>
<td>220.04</td>
<td>219.95</td>
<td>219.81</td>
<td>219.67</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>t</td>
<td>5.58</td>
<td>5.60</td>
<td>5.56</td>
<td>5.50</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_{0.2}%$</td>
<td>591MPa</td>
<td>581MPa</td>
<td>581MPa</td>
<td>567MPa</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_t$</td>
<td>646MPa</td>
<td>636MPa</td>
<td>636MPa</td>
<td>607MPa</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Test 6</td>
<td>D</td>
<td>220.25</td>
<td>219.74</td>
<td>220.13</td>
<td>219.62</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>t</td>
<td>5.55</td>
<td>5.50</td>
<td>5.42</td>
<td>5.54</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_{0.2}%$</td>
<td>591MPa</td>
<td>598MPa</td>
<td>587MPa</td>
<td>567MPa</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_t$</td>
<td>646MPa</td>
<td>650MPa</td>
<td>644MPa</td>
<td>607MPa</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Test 7</td>
<td>D</td>
<td>407.22</td>
<td>407.47</td>
<td>406.79</td>
<td>406.80</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>t</td>
<td>7.56</td>
<td>7.60</td>
<td>7.56</td>
<td>7.57</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_{0.2}%$</td>
<td>535MPa</td>
<td>552MPa</td>
<td>545MPa</td>
<td>552MPa</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$f_t$</td>
<td>592MPa</td>
<td>597MPa</td>
<td>609MPa</td>
<td>597MPa</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Test 8</td>
<td>D</td>
<td>407.22</td>
<td>406.42</td>
<td>406.62</td>
<td>407.16</td>
<td>407.02</td>
<td>406.75</td>
</tr>
<tr>
<td></td>
<td>t</td>
<td>7.55</td>
<td>7.65</td>
<td>7.64</td>
<td>7.60</td>
<td>7.60</td>
<td>7.62</td>
</tr>
<tr>
<td></td>
<td>$f_{0.2}%$</td>
<td>535MPa</td>
<td>485MPa</td>
<td>506MPa</td>
<td>545MPa</td>
<td>502MPa</td>
<td>552MPa</td>
</tr>
<tr>
<td></td>
<td>$f_t$</td>
<td>592MPa</td>
<td>563MPa</td>
<td>595MPa</td>
<td>609MPa</td>
<td>577MPa</td>
<td>597MPa</td>
</tr>
<tr>
<td>Test 9</td>
<td>D</td>
<td>406.96</td>
<td>407.25</td>
<td>406.93</td>
<td>406.75</td>
<td>407.10</td>
<td>406.38</td>
</tr>
<tr>
<td></td>
<td>t</td>
<td>7.58</td>
<td>7.70</td>
<td>7.60</td>
<td>7.63</td>
<td>7.70</td>
<td>7.60</td>
</tr>
<tr>
<td>Test 10</td>
<td>f₀₂₉</td>
<td>535MPa</td>
<td>485MPa</td>
<td>559MPa</td>
<td>485MPa</td>
<td>502MPa</td>
<td>552MPa</td>
</tr>
<tr>
<td>---------</td>
<td>------</td>
<td>--------</td>
<td>--------</td>
<td>--------</td>
<td>--------</td>
<td>--------</td>
<td>--------</td>
</tr>
<tr>
<td>D</td>
<td>407.13</td>
<td>406.97</td>
<td>407.00</td>
<td>406.44</td>
<td>-</td>
<td>-</td>
<td>406.9</td>
</tr>
<tr>
<td>t</td>
<td>7.57</td>
<td>7.59</td>
<td>7.60</td>
<td>7.58</td>
<td>-</td>
<td>-</td>
<td>7.59</td>
</tr>
<tr>
<td>f₀₂₉</td>
<td>535MPa</td>
<td>485MPa</td>
<td>502MPa</td>
<td>552MPa</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>fₜ</td>
<td>592MPa</td>
<td>563MPa</td>
<td>577MPa</td>
<td>597MPa</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Test 1</th>
<th>H86916400B</th>
<th>H86916024</th>
<th>H86916331B</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-9998</td>
<td>-3580</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3670</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>9986</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Test 2</th>
<th>H60415859</th>
<th>H60410114</th>
<th>H6023730</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-10040</td>
<td>-4030</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1860</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>9950</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Test 3</th>
<th>H86916400B</th>
<th>H86916270B</th>
<th>H86916024</th>
<th>H86916331B</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-10010</td>
<td>-4012</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2300</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>10005</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Test 4</th>
<th>H86916400B</th>
<th>H86916288B</th>
<th>H86916331B</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-10030</td>
<td>-4880</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3755</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>9989</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Test 5</th>
<th>H86916331B</th>
<th>H86916314</th>
<th>H86916314</th>
<th>H86916400B</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-10025</td>
<td>-3995</td>
<td>-1445</td>
<td>5145</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>9985</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Test 6</th>
<th>H86916331B</th>
<th>H86916288</th>
<th>H86916400B</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-9994</td>
<td>-3908</td>
<td>-3513</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>5015</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>9992</td>
</tr>
</tbody>
</table>
Figure 2.6: Test 1-10, Positions of connections for one pipeline, the girth welding layout. [mm]
Appendix B: Material Tensile Test Data

Figure A-1: Stress strain property for material type H86916400B

Figure A-2: Stress strain property for material type H86916331B

Figure A-3: Stress strain property for material type H86916024
Figure A-4: Stress strain property for material type H86916270B

Figure A-5: Stress strain property for material type H86916288B

Figure A-6: Stress strain property for material type H86916288
Figure A-7: Stress strain property for material type H86916314

Figure A-8: Stress strain property for material type H60410114

Figure A-9: Stress strain property for material type H60413730
Figure A-10: Stress strain property for material type H60415859

Figure A-11: Stress strain property for material type H60423640

Figure A-12: Stress strain property for material type H60415422
Figure A-13: Stress strain property for material type H60416313

Figure A-14: Stress strain property for material type H60416153

Figure A-15: Stress strain property for material type H60423631
Stress-strain relationship graphs for all modeled pipes are given below.

(a) Material properties comparison for Test 1

(b) Material properties comparison for Test 2

(c) Material properties comparison for Test 3
(d) Material properties comparison for Test 4

(e) Material properties comparison for Test 5

(f) Material properties comparison for Test 6
(g) Material properties comparison for Test 7

(h) Material properties comparison for Test 8

(i) Material properties comparison for Test 9
(j) Material properties comparison for Test 10

Figure A-16: (a)-(j), the sum up of material properties for each entire pipe
Appendix C: Detailed Initial Imperfection Shape

The following figures show the initial profiles of the compression side of the pipes used in the non-linear analysis models resulting from the linear perturbation buckle models.

The horizontal coordinate is the distance from midpoint (fault); the vertical coordinate is the deformation of the pipeline under different modes. “Test” is the local buckling position result from experimental test taken in lab; “FEA” is the one in numerical finite element result. The unit in the figures is mm.

Figure B-1: Linear elastic buckling shape selected as the initial imperfection for Test 2.

Figure B-2: Linear elastic buckling shape selected as the initial imperfection for Test 3.
Figure B-3: Linear elastic buckling shape selected as the initial imperfection for Test 4.

Figure B-4: Linear elastic buckling shape selected as the initial imperfection for Test 5.
Figure B-5: Linear elastic buckling shape selected as the initial imperfection for Test 8.

Figure B-6: Linear elastic buckling shape selected as the initial imperfection for Test 9.
Figure B-7: Linear elastic buckling shape selected as the initial imperfection for Test 10.
Appendix D: Model building in ABAQUS

For creating a thin wall shell element, several steps should be done firstly:

- Create a shell part using extrusion the base shape in length direction; the sketched geometry of the base shape is a circle with an average outside diameter of all the pipe segments in one pipeline. Depth of the end condition of this base shape is the length (around 20 meters) of the pipe.

- Create the all kinds of the material properties for each pipe segment in one pipeline. For a nonlinear analysis, not only the elastic modulus but also the full stress-strain curve should be defined, in terms of true stress-true plastic strain.

- Define the shell homogeneous sections for each segment with selecting their material properties respectively. The value of shell thickness, number of integration points should be assigned appropriately as well. Since there are at least three segments in one pipeline, the one built already should be partitioned in several sections at the specific coordinate. Then select the different regions to assign the defined section property to each pipe segment, the top surface should be chosen for the shell offset definition.

- Create the instance with assembling an independent pipeline part. After the assembly, datum should be positioned in the reference coordinate from the XY Datum plane. Using these datum planes, the partitions could be placed which will help put the reference point mentioned as the slave and control points in the report. The partitions in the top and bottom central line in the strap width should be applied to transform the soil-pipeline interaction bearing load coupled by the slave points. Partition in the front and back central line is applied as facility to output the database in such a line. In the length where the boundary conditions for linear elastic buckling are applied, several partitions in parallel with the XY plane is positioned as well.

In order to apply the boundary conditions, in addition to the points for straps, more reference points should be created in the assembly. They are: (a) A series of points far away from the pipelines with the same coordinate to the control point for each pair of straps in one ring spring; (b) Two ends’ reference
points, one is assigned at 0,0,0, and the other one is assigned at 0,0,x(x=pipe length). (c) If the axial force is not monotonic increasing in the two ends of pipeline, the points as the other end of the axial connector in the longitudinal direction should be applied.

- The element type can be chosen by clicking on ‘Select Element Type’. Linear finite-strain elements should be selected. Reduced integration can be used to control the possibility of shear locking, but the stiffness of the element will also be reduced slightly (S4R).

For the critical part where the linear elastic buckling analysis will be employed, a finer mesh size is needed, so there should be a transition section around 200-400 between the two size sections, using the Structured technique mesh control under Quad-dominated element shape and for the other sections the Free or Sweep technique mesh control are both allowed. Now, the part can be meshed by selecting ‘Mesh Part Instance’.

Some Sets have to be assigned now in order to export the output and in order to indicate the displacement of the specific point or the deformation of all the elements and nodes on the pipeline. A geometry set of a moving referent point far away from the pipeline is created to which the fault movement is assigned; and two node sets are created at both the left and right side in the critical part of the pipeline, the length of node set must cover the distance between the boundary conditions of the linear elastic buckling analysis.

- Connectors and the Continuum distributing Coupling are defined in the Interaction. Many constraints are applied to couple the reference points as the two methods mentioned in the report for simulating the load.
transmission of straps. The reference points in the middle part of pipe and their coupled nodes are coupled by Continuum distributing without the degrees of freedom UR1, UR2 and UR3. The reference points in the centre at the pipeline two ends and their coupled circumferential nodes are coupled by Continuum distributing with all the six degrees of freedom.

Connectors are applied for the 12 ring springs, one of the end points should be the control point positioned in the pipeline with the existed CSYS for pipeline, and the other of the points is the reference point far away from the pipeline with a specify CSYS. The connection type is the basic axial and the section should be the various input linear elastic plastic or nonlinear elastic plastic properties. When using the nonlinear elastic plastic connector properties, some tips should be notified: (1) The Extrapolation option should be ‘Constant’ under ‘Edit Connector Section→Table Options’ menu bar; (2) A force-displacement relationship on the compressive stage (the first two lines in the Elasticity page) of the connector property could have a very gentle slope compared with tensile stage (the last two lines in the Elasticity page) for avoiding the instability of the whole model during operation.
The Boundary conditions should be added to control the displacement of the reference points, as the way said in Chapter 4.6 of the report.

To define the analysis Steps. If the internal pressure is applied, a static general step has to be applied in order to make a constant force and pressure induced by the internal pressure in the Riks step. Then a static Riks step has to be applied. A stopping criteria can be specified by editing the maximum load proportionality factor equals to 1. Nonlinear geometry flag should be activated for both steps.

The incrementation is an important aspect that it controls the speed of the boundary conditions showing the smoothness of the resulting solution, and can influence whether or not the solution converges close to buckling. A maximum and initial step size of 0.01 is a good initial guess which is very close to the experimental value and won’t take a long time; and the minimum step size can also be decrease if the solution does not converge (1E-10 is a good guess):

Some Output Requests should be defined in the Step module as well. For History Output: (1) the displacement of the geometric node set in U1 direction; (2) the force-displacement relationship in all the axial connectors through the wires in U1 direction which is the axial direction of the local coordinate.
system. For Fields Output: choose the output at the layered section points into a Specify value which is
the middle one of the integration points (6 for 11 points, 5 for 9 points, etc.).

-The final and important step is the initial imperfections. Adding the buckle model in the nonlinear
analysis model, both of them are applied to add several sentences in editing the keywords. Within the
graphical user interface, this can be accomplished by right-clicking the name of the model and clicking
‘Edit Keywords’. Select the appropriate scale factor for the imperfection mode, then run the original
model. The sentences are entered before the first step as shown below:

The syntax is as follows:

For the first method which is to add all the deformed nodes in instance part to the nodes in nonlinear
analysis model:

*IMPERFECTION, FILE=FILE NAME, STEP=1

**MODE NUMBER, SCALE FACTOR

Before this analysis works, a buckling analysis has to be performed. The pipeline model is simply copied
and the following changes are made:

-The static steps are suppressed and a Linear Perturbation→Buckle step is added.

-Delete all the extra elements including the connectors and the useless reference points in this model.
Make Kinematic Coupling Constraint between the loading point and the circumferential nodes in
pipeline in all the 6 DOFs.

-A load should be applied. For this situation, both in the left and right side of the pipeline, a pair of
rotation UR2=1 and UR2=2 in the two ends of the defined elastic loading length mentioned in
Chapter4.5 are applied to the coupled reference points for getting the ideal elastic buckling shapes.
Besides, \( U_1 = U_2 = U_{R1} = U_{R3} = 0 \) and \( U_3 = 0 \) in one of the two ends.

-The following keywords should be added to the input file right before the keyword *End Step:

*NODE FILE

U

For the second method which is to add the selected deformed nodes to the nodes of the instance part in nonlinear static analysis model:

*IMPERFECTION, SYSTEM=R

PART NAME, NODE NUMBER, node displacement in \( U_1, U_2, U_3 \)

Before this analysis works, a buckling analysis has to be performed. The pipeline model is simply copied twice and the following changes are made:

-The static steps are suppressed and a Linear Perturbation 
\( \rightarrow \) Buckle step is added.

-Delete all the extra elements including the connectors and the useless reference points in this model.

Make Kinematic Coupling Constraint between the loading point and the circumferential nodes in pipeline in all the 6 DOFs.

-A load should be applied. For this situation, only one buckling part the left or right side of the pipeline will be deformed in one model. A pair of rotation \( UR_2 = 1 \) and \( UR_2 = 2 \) in the two ends of the defined elastic loading length mentioned in Chapter 4.5 are applied to the coupled reference points.
for getting the ideal elastic buckling shapes. Besides, U1=U2=UR1=UR3=0 and U3=0 in one of the
two ends.

-The following keywords should be added to the input file right before the keyword*End Step:

*NODE PRINT, NSET=SET NAME
U

-The node displacement in U1, U2 and U3 could be got from ‘Data File’ by clicking the ‘Monitor’ in
the Job Manager of the buckling model under different eigenvalue number. It should be noticed that
the node number is the globe number which should be converted into the local number on the pipeline
part.

-For the first situation, in buckling model, the name of the job must match the name of the job from which
the initial imperfection will be taken for the first buckling method. For the second situation, the part name
should be the same. After submitting and running, the buckling analysis in complete, the nonlinear static
analysis can be submitted in the same way.

-The way to export the output is in the Visualization stage from ODB file. Make a Path along the central
line in the front and back side of the pipeline first and use these lines to output the strain data depending on
the Ufault. The strain-distance from fault relationship would be exhibited. The axial strain and the
circumferential strain at the specific gauge could be drawn through pickling up the points and output their
strain data with Ufault as the horizontal coordinate. Ufault is the output displacement of the geometric set
obtained from ODB history output. The force-deformation relationship for all the connectors can be
obtained from history output as well.

-This original data would be further manipulated in another data processing program to draw the required
relationship graphs.
Appendix E: Comparison of results at strain gauges

Test 1

Comparison of results: Axial strain at the specific gauge point:
- **Blue** lines are the left part from the middle point, and **Red** lines are the right part.
- Solid lines are the results from test, and dash lines are the results from FEA model.
Test 2

Comparison of results: Axial strain at the specific gauge point:

- **Red** lines are the left part from the middle point, and **Blue** lines are the right part.
- Solid lines are the results from test, and dash lines are the results from FEA model.
Test 3

Comparison of results: Axial strain at the specific gauge point:

- **Red** lines are the left part from the middle point, and **Blue** lines are the right part.

- Solid lines are the results from test, and dash lines are the results from FEA model.
Test 4

Comparison of results: Axial strain at the specific gauge point:

- **Red** lines are the left part from the middle point, and **Blue** lines are the right part.

- Solid lines are the results from test, and dash lines are the results from FEA model.
Test 5

Comparison of results: Axial strain at the specific gauge point:

- **Blue** lines are the left part from the middle point, and **Red** lines are the right part.

- Solid lines are the results from test, and dash lines are the results from FEA model.
Test 6

Comparison of results: Axial strain at the specific gauge point:

- Red lines are the left part from the middle point, and Blue lines are the right part.

- Solid lines are the results from test, and dash lines are the results from FEA model.
Test 6

Comparison of results: Circumferential strain at the specific gauge point:

- **Blue** lines are the left part from the middle point, and **Red** lines are the right part.

- Solid lines are the results from test, and dash lines are the results from FEA model.
Test 7

Comparison of results: Axial strain at the specific gauge point:

- **Red** lines are the left part from the middle point, and **Blue** lines are the right part.

- Solid lines are the results from test, and dash lines are the results from FEA model.
Test 7

Comparison of results: Circumferential strain at the specific gauge point:

- **Blue** lines are the left part from the middle point, and **red** lines are the right part.

- Solid lines are the results from test, and dash lines are the results from FEA model.
Test 8

Comparison of results: Axial strain at the specific gauge point:

- **Blue** lines are the left part from the middle point, and **Red** lines are the right part.
- Solid lines are the results from test, and dash lines are the results from FEA model.
Test 9

Comparison of results: Axial strain at the specific gauge point:

- **Blue** lines are the left part from the middle point, and **Red** lines are the right part.

- Solid lines are the results from test, and dash lines are the results from FEA model.

- Results shown for RK13 and RK21, there is an **Orange** solid line representing the axial strain under the scale factor equals to 0.075, local buckling will occur a little bit earlier than the 0.05 one.
Test 10

Comparison of results: Axial strain at the specific gauge point:

- **Blue** lines are the left part from the middle point, and **Red** lines are the right part.

- Solid lines are the results from test, and dash lines are the results from FEA model.
Comparison of results: Circumferential strain at the specific gauge point:

- **Blue** lines are the left part from the middle point, and **Red** lines are the right part.

- **Solid lines** are the results from test, and **dash lines** are the results from FEA model.

*Some strain gauges were broken in the experimental process so that further strain data were missing.*