



OULUN YLIOPISTO
UNIVERSITY of OULU

DEPARTMENT OF MECHANICAL ENGINEERING

FINITE ELEMENT MODELLING OF X-BAND RF FLANGES

Lauri Kortelainen

BACHELOR'S THESIS

2013

Supervisors: Dr. Hannu Koivurova
Dr. Germana Riddone



ABSTRACT

Finite element modelling of x-band RF flanges

Lauri Kortelainen

University of Oulu, Department of Mechanical Engineering

Bachelor's thesis 2013, 48 p. + 2 p. appendixes

Supervisor(s): Dr. Hannu Koivurova

Dr. Germana Riddone

A finite element model of different versions of RF flange used in Compact Linear Collider modules was created in ANSYS Workbench software. A 2D idealisation of the assembly was modelled using both plane stress and plane strain elements. Three of the versions were also modelled using 3D elements. A detailed description of finite element models and theoretical background accompanying the models are presented in this thesis.

Keywords: CERN, FEA, RF, FLANGE

TIIVISTELMÄ

Finite element modelling of x-band RF flanges

Lauri Kortelainen

Oulun yliopisto, Konetekniikan osasto

Kandidaatintyö 2013, 48 s. + 2 s. liitteitä

Työn ohjaaja(t): TkT Hannu Koivurova

Dr. Germana Riddone (CERN)

Compact Linear Collider –moduuleissa sijaitsevan laippakokoonpanon eri versiot mallinnettiin käyttäen elementtimenetelmää ANSYS Workbench –ohjelmistolla. Kokoonpano mallinnettiin käyttäen kaksiulotteisia tasojännitys ja –venymäelementtejä. Kolme versioista mallinnettiin myös käyttäen kolmiulotteisia solidielementtejä. Tämä työ esittelee mallinnuksen yksityiskohdat ja teoriaa liittyen aiheeseen.

Asiasanat: CERN, FEA, RF, FLANGE

ACKNOWLEDGEMENT

This thesis was carried out at CERN in a collaboration with Helsinki Institute of Physics (HIP) and University of Oulu. I would like to thank my supervisors Dr. Hannu Koivurova and Dr. Germana Riddone for their guidance and support. I also thank Dr. Fabrizio Rossi for giving me technical advice during the development of the simulation.

Geneva, 16th of April 2013

A handwritten signature in black ink, appearing to read 'Lauri' followed by a stylized flourish.

Lauri Kortelainen

TABLE OF CONTENTS

ABSTRACT

TIIVISTELMÄ

ACKNOWLEDGEMENT

TABLE OF CONTENTS

NOTATIONS

ABBREVIATIONS

1 INTRODUCTION.....	7
1.1 Background	8
1.2 CLIC study.....	9
2 THEORETICAL BACKGROUND	12
2.1 Tensile tests.....	12
2.1.1 Behaviour at larger strains	15
2.1.2 True stress and strain	16
2.2 Material models.....	18
2.2.1 Linear elastic.....	18
2.2.2 Nonlinear elastic-plastic material model	20
2.3 2D idealisation	22
2.4 Influence of heat treatment on mechanical properties of material: annealing	24
3 TECHNICAL DESCRIPTION OF THE X-BAND RF FLANGE.....	25
3.1 Geometry.....	25
4 2D FINITE ELEMENT MODEL DESCRIPTION	26
4.1 Model	26
4.2 Material	28
4.3 Mesh.....	29
4.4 Contact modelling	30
4.5 Boundary conditions	31
5 3D FINITE ELEMENT MODEL DESCRIPTION	33
5.1 Model	33
5.2 Material	34
5.3 Mesh.....	34
5.4 Contact modelling	34
5.5 Boundary conditions	35

6 RESULTS	37
6.1.1 2D plane stress and strain modelling	37
6.1.2 3D modelling	39
7 CONCLUSIONS AND FUTURE STEPS	46
8 REFERENCES.....	47
APPENDIX 1	

NOTATIONS

A cross-sectional area

E Young's Modulus

E_t tangent modulus

F force

ϵ engineering strain

ϵ_t true strain

ν Poisson's ratio

Δl elongation

σ engineering stress

σ_t true stress

ABBREVIATIONS

CLIC	Compact Linear Collider
ILC	International Linear Collider
LEP	Large Electron-Positron Collider
LHC	Large Hadron Collider
AS	Accelerating Structure
PETS	Power Extraction and Transfer Structure
RF	Radio Frequency
MB	Main Beam
DB	Drive Beam
FEA	Finite Element Analysis

1 INTRODUCTION

This thesis presents the finite element modelling of x-band RF flange used in CLIC modules. The aim of the finite element modelling is to examine the deformations of the copper gasket for different versions of the flange and to check if the specified requirements are met. In addition to the detailed description of the numerical simulations, the theoretical background is presented. The theoretical part of the thesis includes information about tensile tests and how the data obtained from them can be implemented in the finite element modelling. Elastic-plastic material models are described as well and the theory behind the two-dimensional idealisation of three-dimensional structure is presented.

Chapter 1 introduces CERN, CLIC project and CLIC two-beam modules. The theoretical background for the simulation is presented in chapter 2. Chapter 3 presents the flange assembly in question and its technical specifications. Chapter 4 describes the two-dimensional finite element model, whereas chapter 5 describes the three-dimensional one. Results of the simulations are presented in chapter 6. Chapter 7 includes conclusions and future steps.

1.1 Background

The European Organization for Nuclear Research (CERN), one of the world's largest and most respected particle physics laboratory, was founded in 1954. Today CERN is run by 20 Member States but also many non-European countries are involved in different research programs. (Ellis 2000.) Currently about 2400 people are employed by CERN but over 10 000 visiting scientists are related to the on-going projects through their research. They represent 608 universities and 113 nationalities. (About CERN 2013.)

CERN's main function is to provide accelerators for the high-energy physics research. During CERN's existence, several important scientific discoveries have been made in physics. Furthermore, the research work related to particle accelerator studies has revolutionized the technology used in many other fields. For example, CERN had an important role in the discovery of Positron Emission Tomography (PET) technology, which enables three-dimensional imaging of the functional processes of the brain. Recently, CERN has been a centre for the development of the grid network which enables the handling of large amounts of data produced by the current accelerator projects. (Physics for Health 2010.)

At the moment, LHC (Large Hadron Collider), which was completed in 2008, is CERN's most important accelerator. The circular LHC tunnel, which was occupied by LEP (Large Electron-Positron Collider) until year 2000, has a circumference of 27 km and is located 100 meters underground. On the 4th of July 2012 CERN held a press conference in which the finding of a new gauge boson was confirmed. As of February 2013 the operation of LHC was interrupted for LS1 (Long Shutdown 1). During shutdown wear parts will be replaced and upgrades will be made.

Many of the open questions in physics can be best addressed by a lepton-antilepton (electron-positron collider) instead of a hadron collider like LHC. A consensus has been reached among particle physicists that the results from the LHC have to be complemented by experiments at an electron-positron collider operating in the TeV

energy range. (Ellis 2000: 3.) Today, two approaches as a future linear collider are being developed in parallel: International Linear Collider (ILC) and Compact Linear Collider (CLIC). They have a different energy range, which leads to a different choice of technology. The main difference between the two is that ILC uses superconducting technology like LHC, which means components must be close to the absolute zero temperature, while CLIC uses normal-conducting technology at temperatures close to room temperature. (Geschonke 2010.)

1.2 CLIC study

The CLIC study has proven the feasibility of a multi TeV linear electron-positron collider with an accelerating gradient of 100 MV/m. The current CLIC layout is presented in Figure 1. (Stapnes et al. 2012)

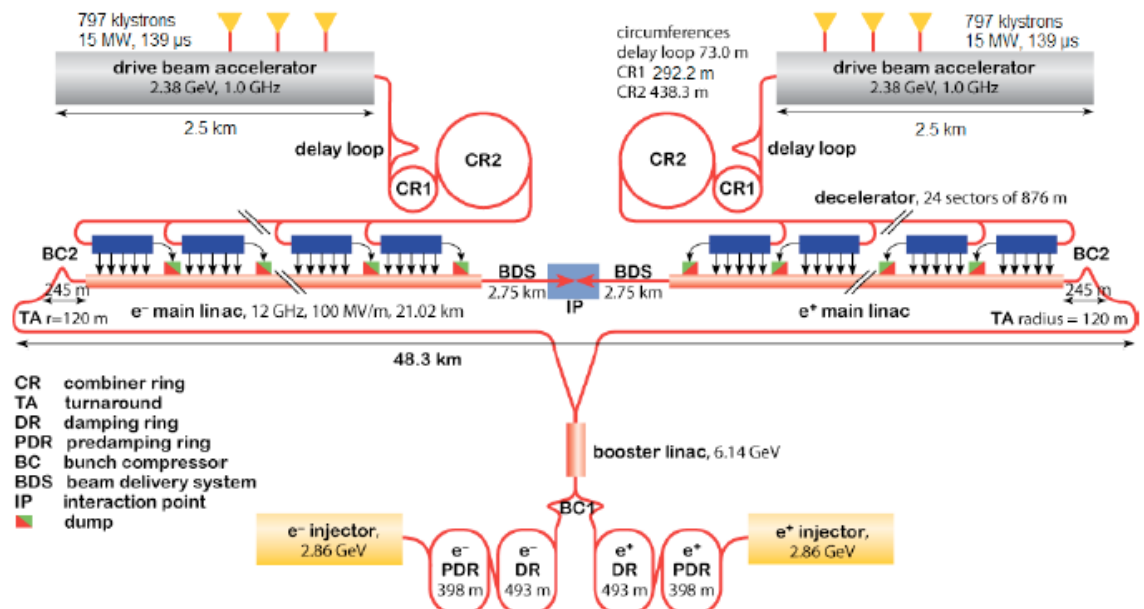


Figure 1. The proposed layout for CLIC.

The layout of CLIC relies on transforming the high current intensity and low energy from the Drive Beam (DB) to the low current and high energy Main Beam (MB). The RF power created in Power Extraction and Transfer Structures (PETS) is transferred through RF network to the Accelerating Structures (AS). The beam focusing is achieved by using quadrupole magnets which are situated along the whole length of the linac. The concept of the accelerator with the main components such as AS, PETS and quadrupoles can be seen in Figure 2. (Stapnes et al. 2012)

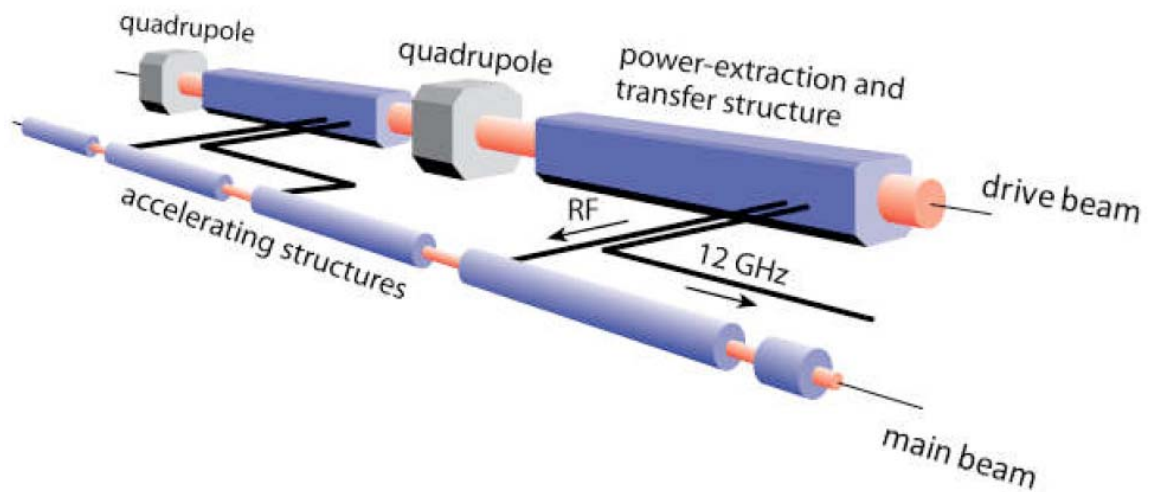


Figure 2. The CLIC Two-Beam scheme. The power provided by PETS is transferred to AS through RF network.

The CLIC main linacs consist of a series of two-beam modules each having a length of about 2 metres. The main components of a module are Accelerating Structures (AS) and Power-Extraction and Transfer Structures (PETS). A 3D model of a two-beam module is shown in Figure 3.

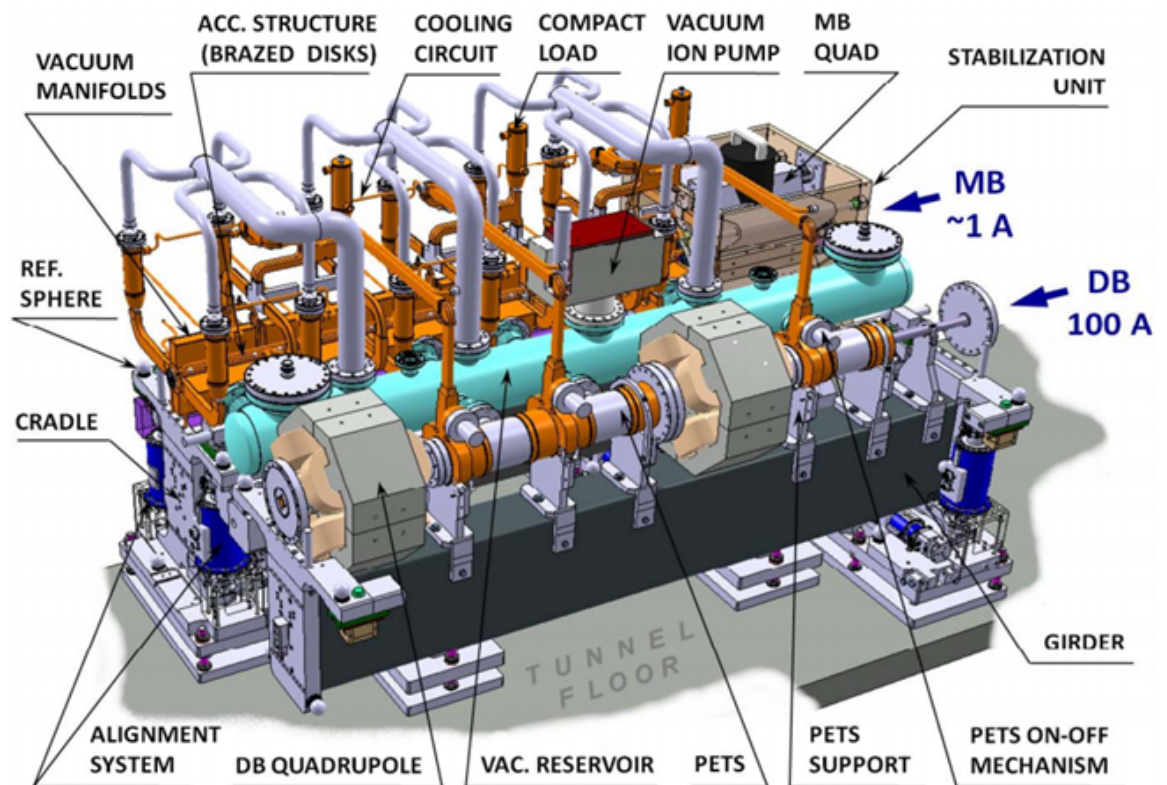


Figure 3. CLIC Two-Beam Module typical configuration.

2 THEORETICAL BACKGROUND

This chapter presents theoretical background for the finite element simulations.

2.1 Tensile tests

The aim of a tensile test is to evaluate material properties for the metal in question. The properties obtained can then be used for the design of components. For the tensile test first the specimen is placed in the testing machine (Figure 4) and then a force is applied. The force is increased very slowly and a strain gauge measures the change of length in the specimen. The results of a single test can then be applied to all sizes and cross-sections of specimens if the force is converted to stress and the distance between gauge marks to strain. Engineering stress and strain are defined in equations (1) and (2). (Askeland et al. 2010: 159-161.)

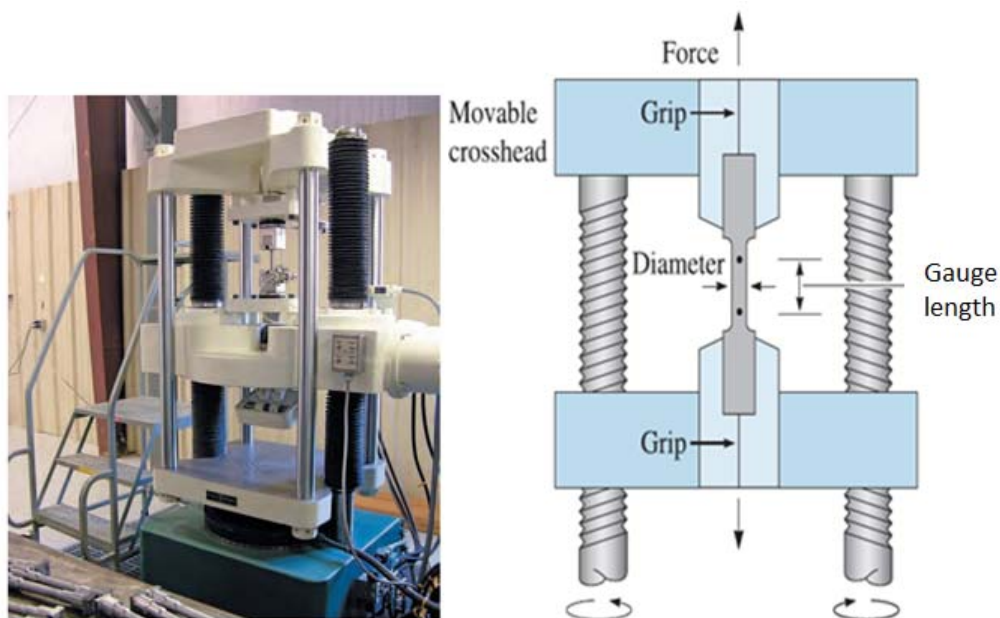


Figure 4. Uniaxial testing machine for tensile test.

$$\sigma = \frac{F}{A_0} \quad (1)$$

where, σ engineering stress [Pa]
 F force [N]
 A_0 initial cross-sectional area [m²]

$$\varepsilon = \frac{\Delta l}{l_0} \quad (2)$$

where, ε engineering strain
 Δl elongation [m]
 l_0 initial length of the test specimen [m]

Figure 5 presents a typical stress-strain curve at small strains obtained from a tensile test. The highest stress that can be applied in the linear region is called **the proportional limit**. The maximum stress that can be applied without causing permanent deformations is called **the elastic limit**. Usually the value found from reference books of material properties is neither of these. The departure from elasticity is defined by means of **proof stress**, value which is independent of the accuracy of the measurement device. This is done by choosing a strain value (normally 0.2%) and drawing a line parallel to Young's modulus at this point. The point of intersection between this line and the stress-strain curve marks the proof stress. This procedure also known as **the offset method** is shown in Figure 6. (Martin 2006: 37-38.)

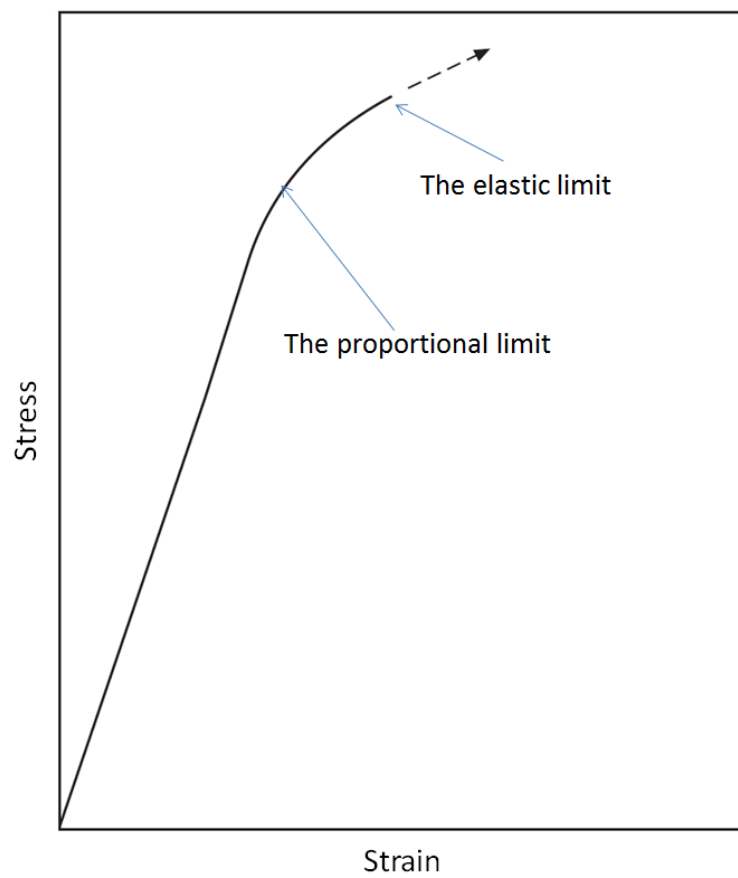


Figure 5. Behaviour of metal at small strains.

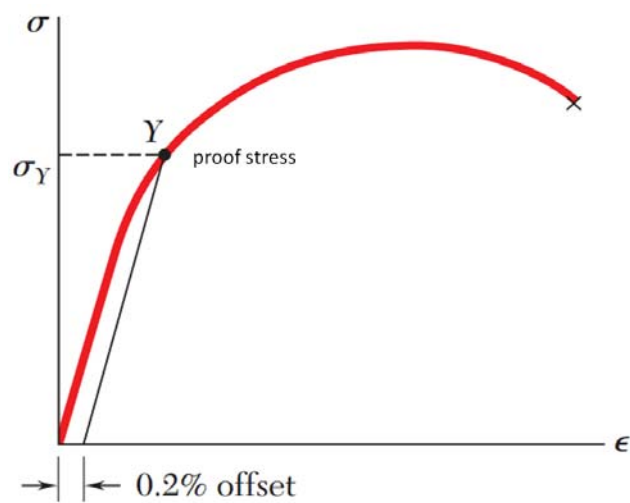


Figure 6. The offset method for evaluating the proof stress.

2.1.1 Behaviour at larger strains

Figure 7 shows the behaviour of a test specimen at larger strains. After the initial region the specimen becomes plastic. In this region the specimen will remain deformed if the load is removed and the load required to continue plastic deformation increases with increasing elongation, i.e. the material work hardens. The maximum stress on the stress-strain curve, **the tensile stress**, is the point at which **necking** starts. In necking the cross-sectional area of the specimen starts to decrease and with a reduced area less force is needed to continue the deformation. Since engineering stress is based on the initial cross-sectional area A_0 , the curve starts to decrease. Finally because of necking, the stress becomes rather localized and final rupture takes place. An illustration of necking and final rupture of a ductile test specimen is shown in Figure 8. (Martin 2006: 40.)

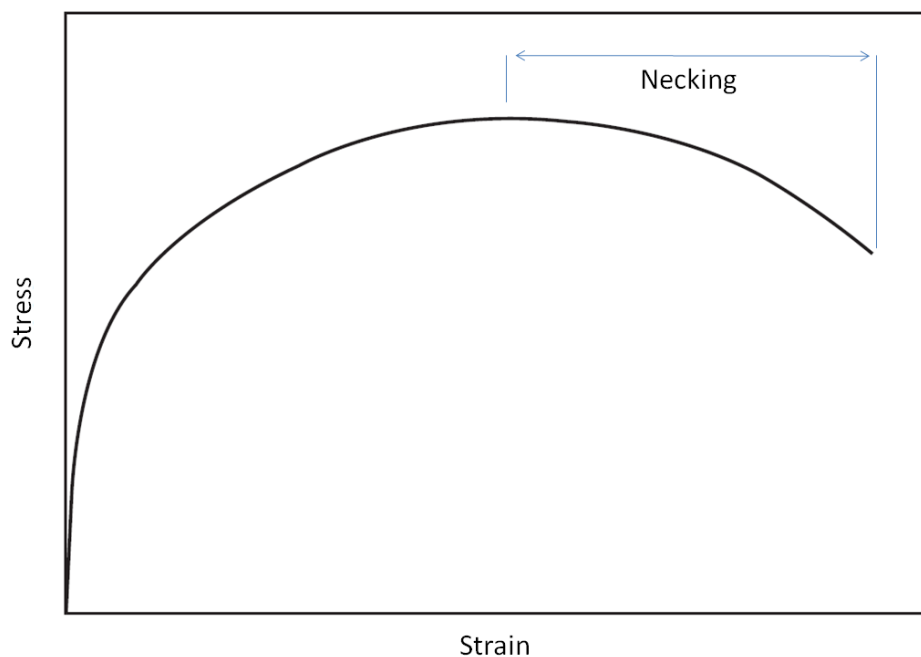


Figure 7. Behaviour of metal at larger strains



Figure 8. Necking (a) and final rupture (b) of a ductile test specimen.

2.1.2 True stress and strain

Engineering stress is defined by using the initial cross-sectional area A_0 , which leads to the decrease in engineering stress at the onset of necking. In reality the cross-sectional area changes continually. True stress is defined by using the deformed cross-section instead of the initial one. True strain, also known as the logarithmic strain, provides the correct measure of the strain when deformation occurs incrementally, taking into account the influence of the strain path. Equations (3) and (4) define true stress and true strain while equations (5) and (6) show the relationships between engineering and true stress, and engineering and true strain respectively. Equations (5) and (6) are only valid until the onset of necking. Beyond this point true stress and strain should be computed from actual load, cross-sectional area and gauge length parameters. (Callister 2001: 167-168; Smallman 1999: 197-199.)

$$\sigma_t = \frac{F}{A} \quad (3)$$

where, σ_t true stress [Pa]
 F force [N]
 A cross-sectional area [m²]

$$\varepsilon_t = \int_{l_0}^l \frac{dl}{l} = \ln\left(\frac{l}{l_0}\right) = \ln\left(\frac{A}{A_0}\right) \quad (4)$$

where, ε_t true strain

$$\sigma_t = \sigma(1 + \varepsilon) \quad (5)$$

$$\varepsilon_t = \ln(1 + \varepsilon) \quad (6)$$

The true stress-strain curve is compared with the engineering stress-strain curve in Figure 9. The curves are identical until the yield point. After this, true stress continues to increase. Although the required load decreases, the cross-sectional area decreases even more leading to increase in stress. (Beer et al 2012: 61-62.)

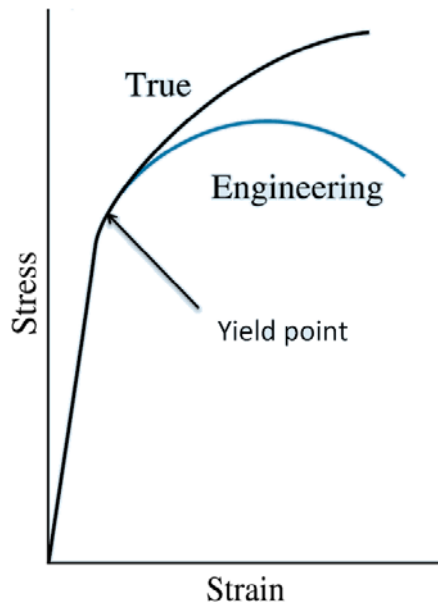


Figure 9. True stress-strain and engineering stress-strain curves.

2.2 Material models

2.2.1 Linear elastic

Figure 10 presents the stress-strain curve of a linear elastic material under uniaxial load and the stress components in 3D. Most engineering structures are designed to undergo only small deformations, involving only the linear part of the stress-strain curve, in which the slope of the curve is called Young's Modulus E . The relationship between stress and strain components applied in three dimensions, shown in equation (7), is called the generalized Hooke's law. (Beer et al 2012: 62.)

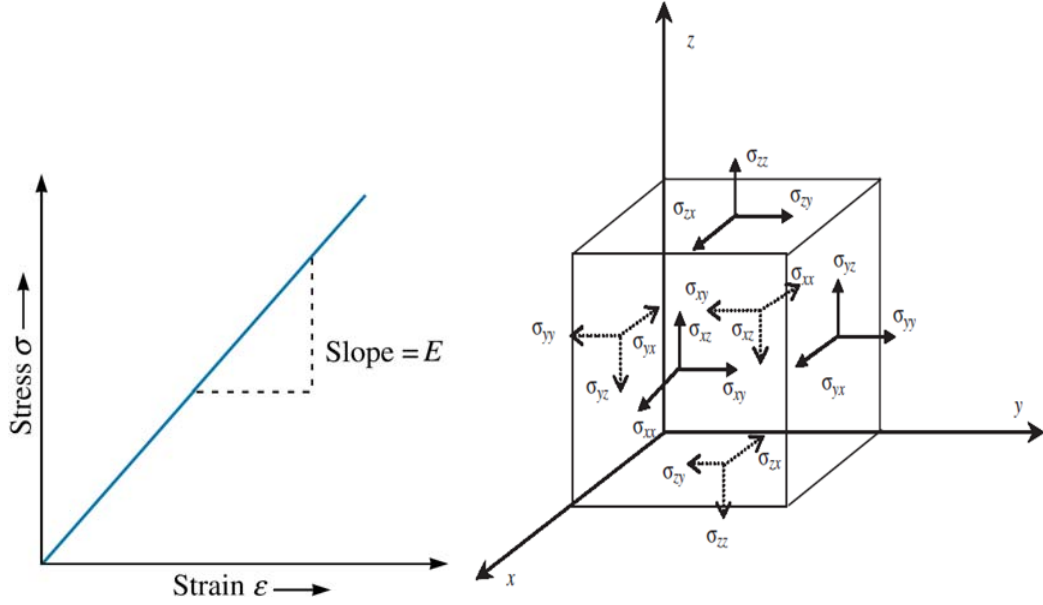


Figure 10. Young's Modulus E is defined as the slope of the linear-elastic stress-strain curve (left). Stress components in 3D (right).

$$\begin{aligned}
 & \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \\
 & = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \varepsilon_{xy} \\ \varepsilon_{yz} \\ \varepsilon_{zx} \end{Bmatrix}
 \end{aligned} \tag{7}$$

where, σ stress component

ε strain component

E Young's Modulus

ν Poisson's Ratio

2.2.2 Nonlinear elastic-plastic material model

In nonlinear material model the slope of the tangent, which replaces E , changes with the level of stress (Figure 11). Usually a simplified version of the nonlinear stress-strain curve is used when conducting an analysis.

One possible method is to use a stress-strain curve consisting of two line segments: one for the linear part and one for the plastic part. This is called **an idealized elastic-plastic material model**. If the stress-strain curve consists of two line segments but the plastic part has a slope, the material model is called **bilinear isotropic hardening**. Using this model the work hardening of the material during plastic deformation is taken into account which means that an increase in load is required for the deformation to continue. E_t , tangent modulus, is the slope in the plastic part of the curve. The two material models are presented in Figure 12. (Beer et al. 2012: 64-65.)

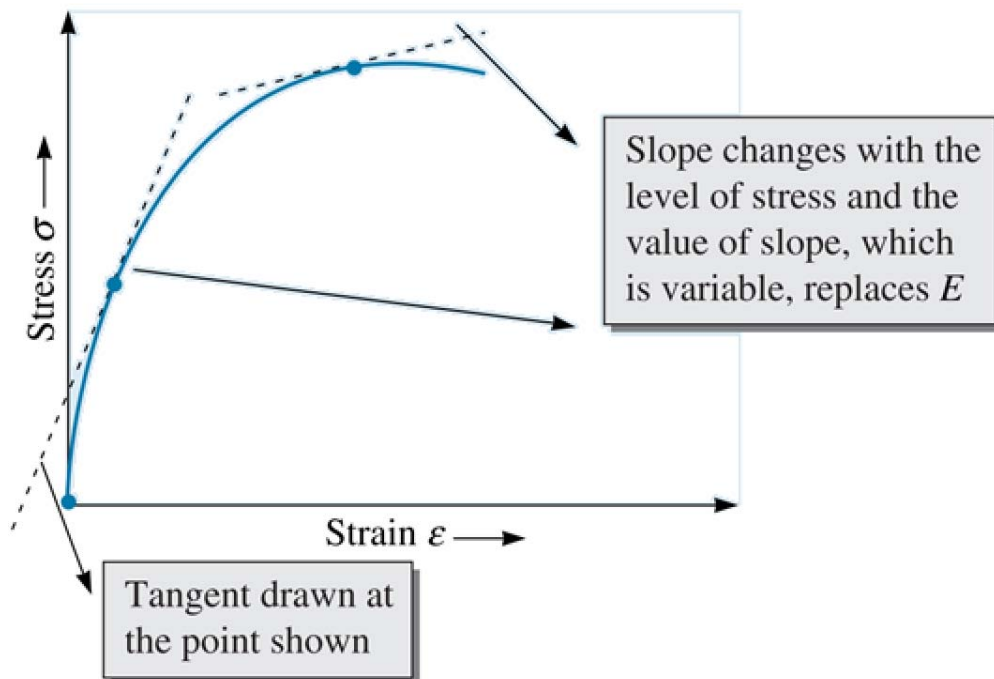


Figure 11. Nonlinear stress-strain curve.

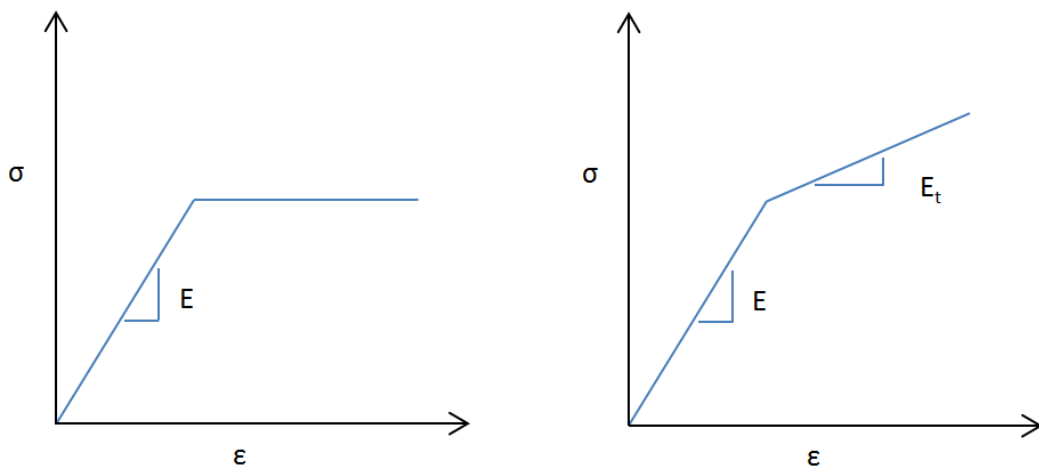


Figure 12. An idealized elastic-plastic material model (left) and bilinear hardening material model (right). E is Young's Modulus and E_t tangent modulus.

2.3 2D idealisation

Three-dimensional (3D) problems can be simplified if they can be treated as two-dimensional (2D). In 2D models, one coordinate, usually the z-axis, is removed. It is then assumed that all the dependent variables are independent of the z-axis. Also all the external loads should be independent of the z-coordinate and applied only in x-y plane. Figure 13 presents the 3D stress components in a cubic block and corresponding 2D stress components when z-coordinate is omitted. (Liu 2003: 19-21)

There are two types of 2D elements: plane stress and plane strain. 2D plane stress element has a thickness that is small compared to the other dimensions. Plane strain element on the other hand has a thickness in z direction which is very large compared to the other dimensions, hence the strain in the z direction is assumed to be zero. Figure 14 illustrates both plane stress and plane strain elements. Hooke's law for 2D plane stress and 2D plane strain is presented in equations (8) and (9). (Liu 2003: 19-21)

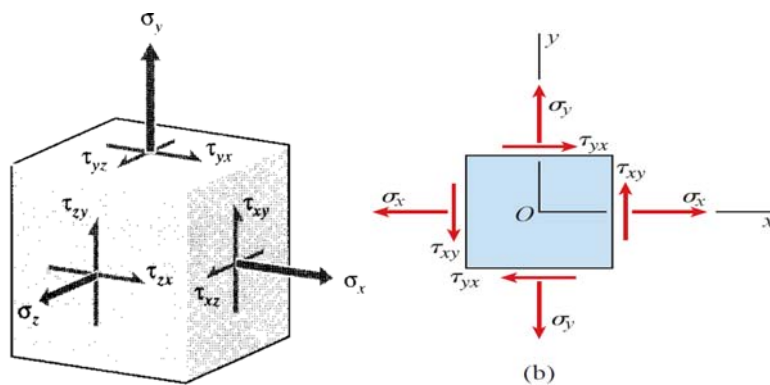


Figure 13. Stress components in a 3D cubic block and a 2D idealisation where components in the z direction are omitted.

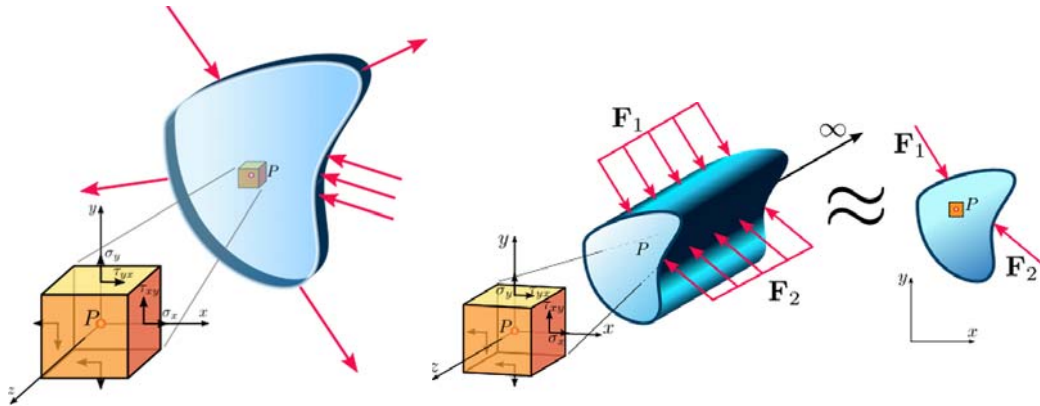


Figure 14. Illustration of plane stress (thin body) and plane strain (thick body) solids.

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} \quad (8)$$

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & 0 \\ \frac{\nu}{1-\nu} & 1 & 0 \\ 0 & 0 & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} \quad (9)$$

2.4 Influence of heat treatment on mechanical properties of material: annealing

Annealing is a heat treatment in which a material is exposed to an elevated temperature and then cooled. This process leads to the elimination of the residual stresses produced during cold working. Annealing at low temperatures will eliminate the residual stresses that were produced during cold working without affecting the mechanical properties of the part. Cold working annealing cycle can be repeated many times to produce the required material properties. Normally a fine-grained microstructure is desired so the heat treatment is stopped before appreciable grain growth has occurred. Temperatures commonly used for annealing cold-worked oxygen free copper are between 250 and 650 °C. (Callister 2001: 124.)

3 TECHNICAL DESCRIPTION OF THE X-BAND RF FLANGE

3.1 Geometry

Figure 15 presents the geometry of the flange assembly with the main dimensions. The assembly consists of two stainless steel flanges and an annealed copper gasket between them. In order to assure the leak tightness, the two flanges are pushed against each other by threading the eight screws connecting them. As a result, the copper gasket is compressed by the axial forces produced by the tightening torque (~ 8 Nm).

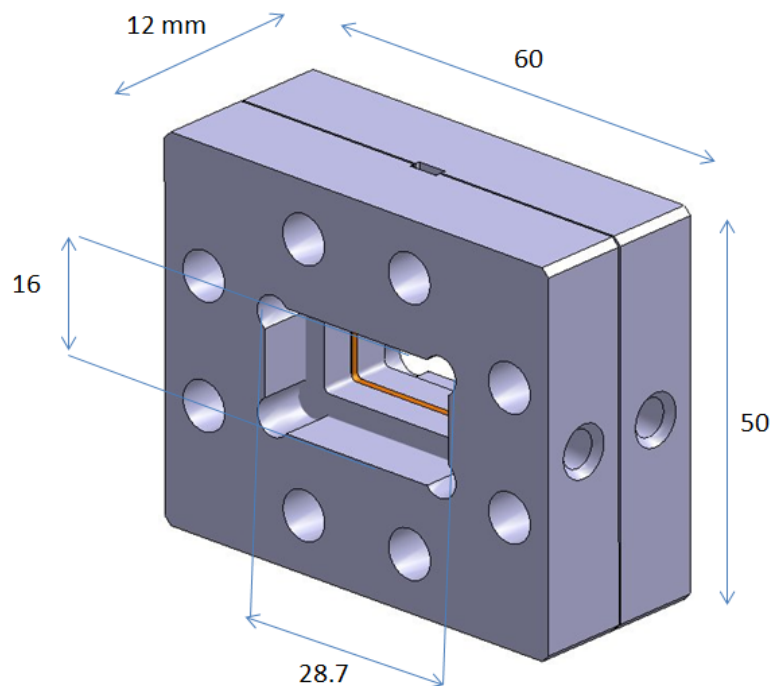


Figure 15. Assembly of RF flanges and the copper gasket (all dimensions are in millimetres).

4 2D FINITE ELEMENT MODEL DESCRIPTION

2D finite element modelling is described in detail in this chapter.

4.1 Model

2D analyses using plane stress and plane strain elements were created in ANSYS Workbench FEA software to evaluate the deformations of the gasket when compressed by the two flanges. Figure 16 presents the dimensions of the geometry, the flange knife being on the left. Twelve different geometries summarized in Table 1 were simulated. Detailed drawings of the flange and the gasket can be found in Appendix 1. Large deformation option was enabled to take into account large strains occurring in the gasket.

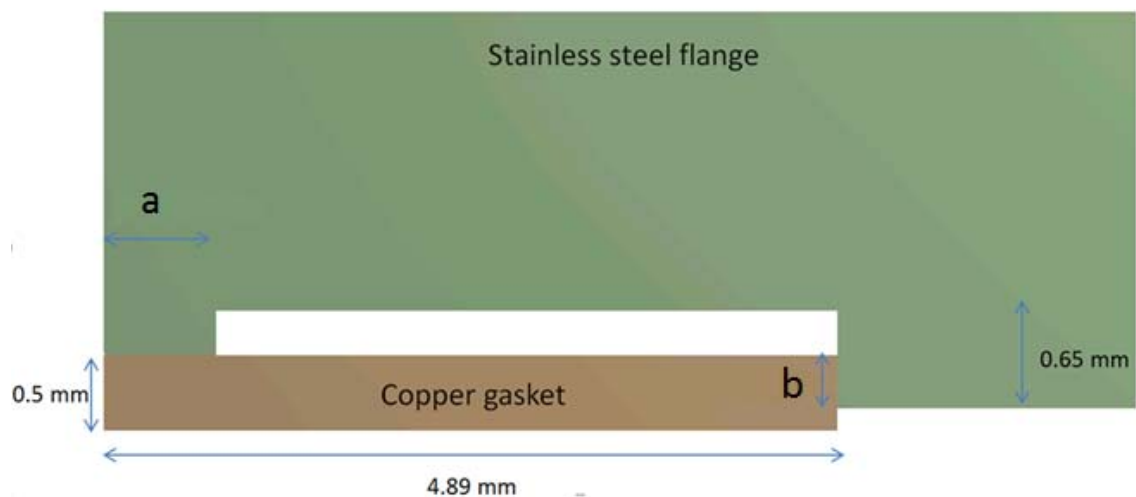


Figure 16. 2D model of the flange with gasket.

Table 1. Flange geometry parameters for versions 1 to 12.

Version	a (mm)	b (mm)
1	0.75	0.35
2	0.75	0.3
3	0.5	0.35
4	0.5	0.3
5	1	0.35
6	1	0.3
7	0.5	0.4
8	0.5	0.25
9	0.75	0.4
10	0.75	0.25
11	1	0.4
12	1	0.25

4.2 Material

Stainless steel flange was modelled as a linear elastic material. Table 2 shows the material parameters used for stainless steel.

Table 2. Material properties of stainless steel.

Property	Value
Modulus of Elasticity	206 GPa
Poisson's Ratio	0.3

Annealed copper gasket was modelled with a linear-plastic material model. The data from the material database *MatWeb*, shown in Table 3, were used to create a bilinear elastic-plastic engineering stress-strain curve which was further converted to equivalent true stress-strain curve to model the real behaviour of the material (Figure 17).

Table 3. Material properties of annealed copper.

Property	Value
Hardness	50 HV
Tensile Strength, Ultimate	210 MPa
Tensile Strength, Yield	33.3 MPa
Elongation at Break	60 %
Modulus of Elasticity	110 GPa
Bulk Modulus	140 GPa
Poisson's Ratio	0.343
Shear Modulus	46 GPa

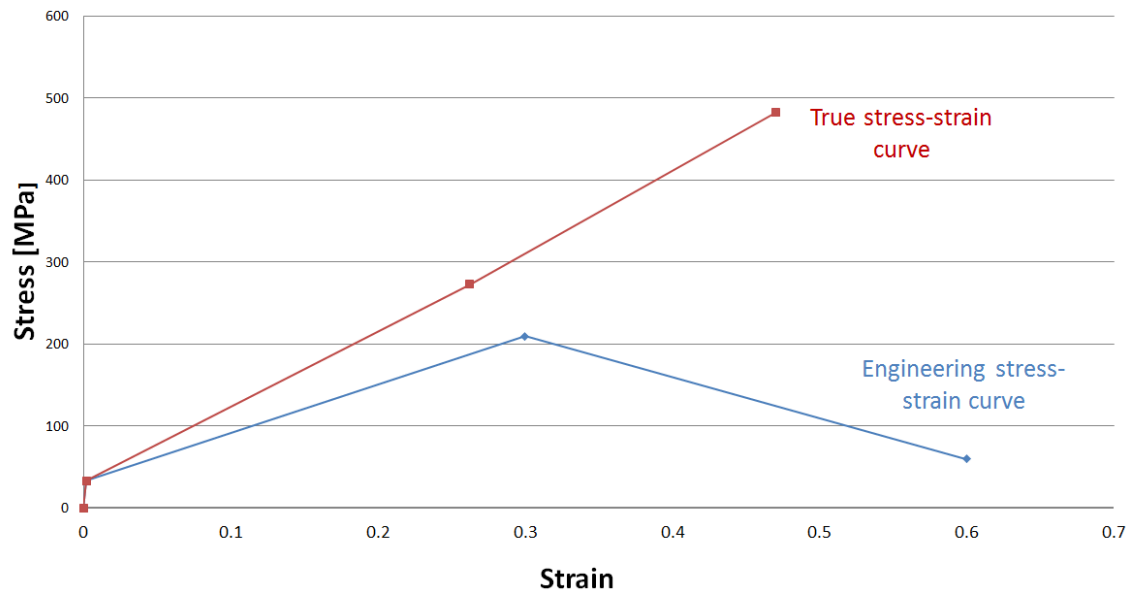


Figure 17. Engineering and true stress-strain curves for annealed copper.

4.3 Mesh

The flange and the gasket were meshed with 2D quadrilateral elements with mid-side nodes activated. The average value of the mesh size is 0.02 mm, resulting in 44 787 elements and 135 893 nodes. Global mesh of the flange assembly and a detail of the flange knife area are presented in Figure 18.

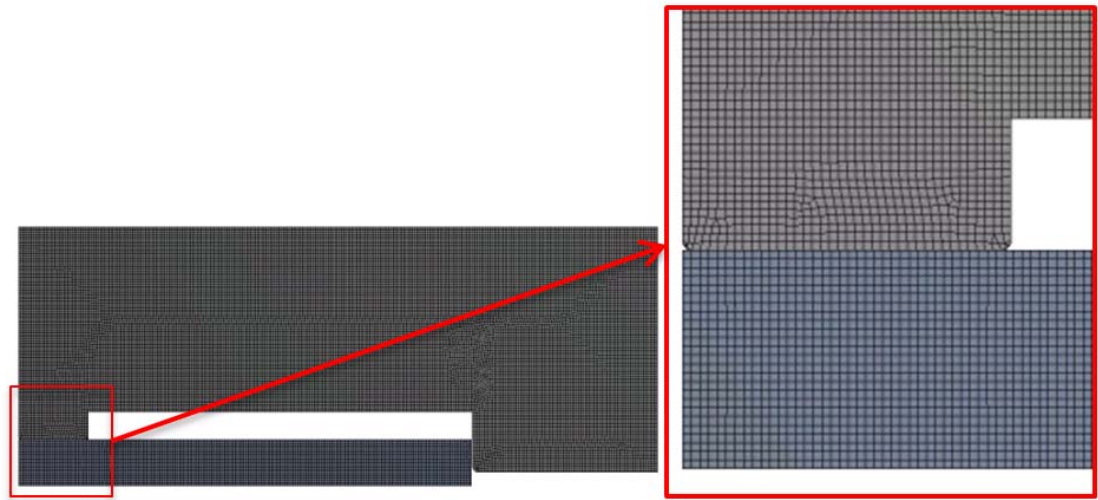


Figure 18. 2D global mesh (left) and detail of the flange knife contact area (right).

4.4 Contact modelling

The model includes contacts A and B between the flange and the gasket. Both of the contacts were modelled as frictional, the friction coefficient being $\mu = 0.5$. Contact A is defined between the flange knife and the gasket. Contact B, which is defined between the vertical edges constraints the deformation of the gasket rightwards. Contact B includes an initial gap of 0.01 mm due to the tolerances of the gasket and the flanges. The contact areas are shown in Figure 19.



Figure 19. Contact definitions for 2D analysis.

4.5 Boundary conditions

Three boundary conditions, shown in Figure 20, were applied to the model. Boundary condition C is an imposed displacement up to the symmetry line. Because only half (one flange and half of the gasket) of the geometry was modelled, boundary condition D was added to represent the symmetry, while boundary condition E constraints the left side of the flange.

C: $y =$ imposed displacement to the symmetry line

D: $y = 0$ (symmetry)

E: $x = 0$

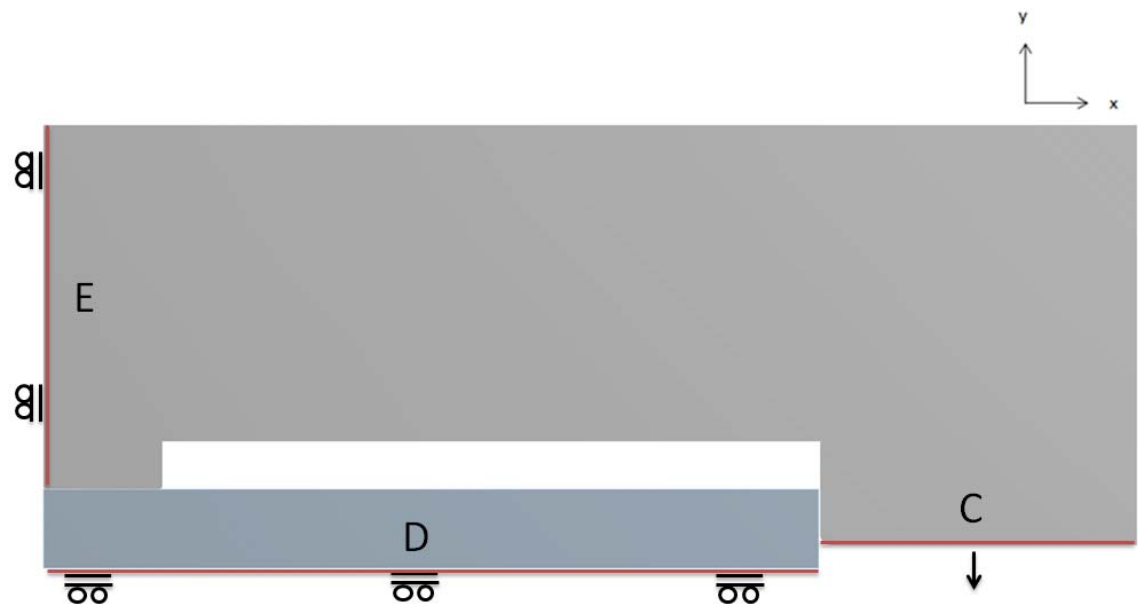


Figure 20. Boundary conditions for 2D analysis.

5 3D FINITE ELEMENT MODEL DESCRIPTION

This chapter describes in detail the 3D finite element model.

5.1 Model

3D model was created in ANSYS Workbench FEA software to evaluate the deformations of the gasket for versions 1, 7 and 12. In particular, on the basis of 2D results, version 7 had the minimum deformations, while version 12 had the maximum deformations. Because of symmetry only 1/8 of the geometry was used. In Figure 21, L refers to the long face of the gasket, while S refers to the short face of it. As in 2D simulation, large deformation option was enabled to take into account large strains occurring in the gasket.

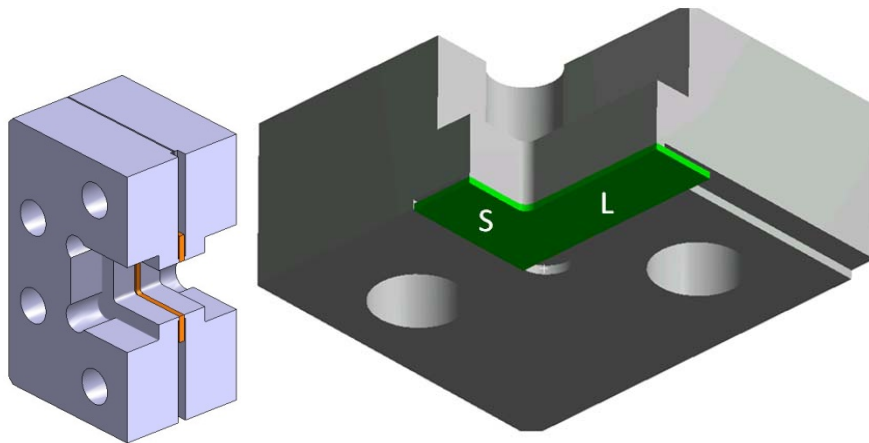


Figure 21. Left: Assembly of the flanges and the gasket (half). Right: A view of the finite element geometry (L: long face of the gasket, S: short face of the gasket).

5.2 Material

The same material models as in the 2D simulation (chapter 4.2) were used.

5.3 Mesh

The gasket was meshed with hexagonal cube elements and the flange with tetrahedrons, mid-side nodes being activated in both parts; in order to improve the accuracy of the simulation and to reduce the computational time, the mesh was refined in the contact area. The average value of the mesh size is 0.1 mm, resulting in 189 378 elements and 478 009 nodes (Figure 22).

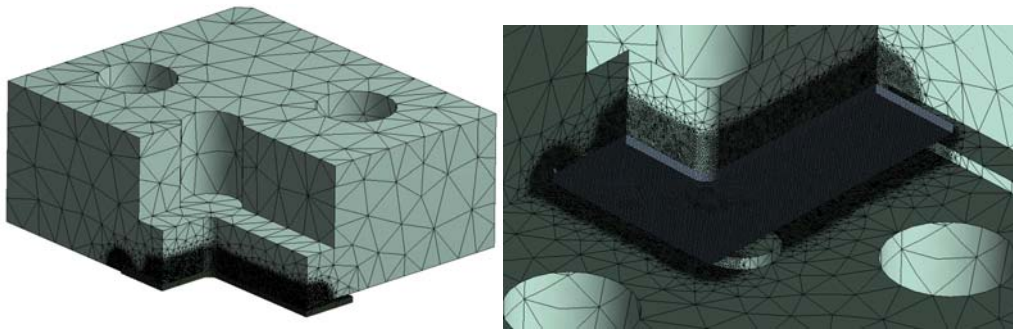


Figure 22. Left: Global mesh. Right: Detailed view of the contact area.

5.4 Contact modelling

Three contacts were defined for the analysis (Figure 23). Contacts A and B are defined between the vertical surfaces of the flange and the external surfaces of the gasket, so that the outward expansion of the gasket is constrained by the internal surfaces of the flange. Contact C, defined between the flange knife and the gasket, is transferring the load from the flange to the gasket. Friction coefficient $\mu = 0.5$ was used for all the contacts.

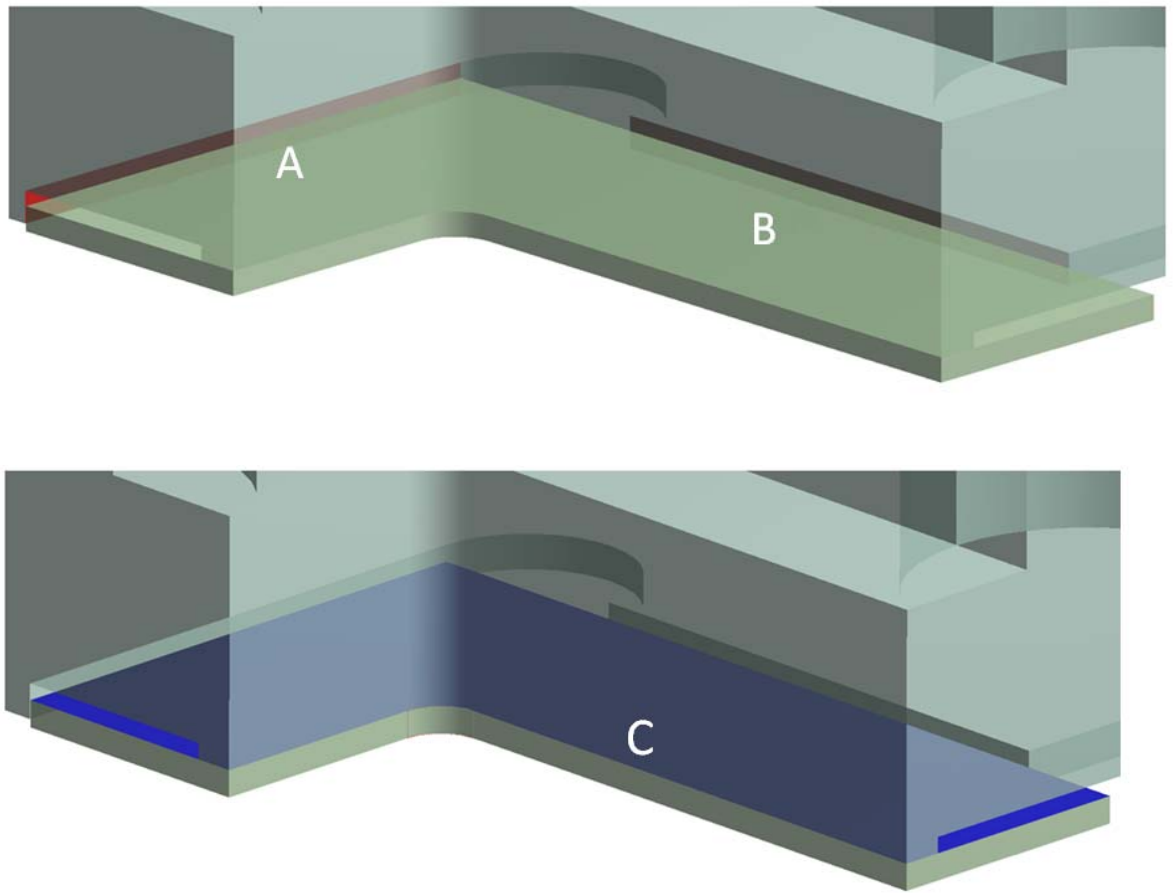


Figure 23. Contact definition for the 3D modelling.

5.5 Boundary conditions

Boundary conditions are presented in Figure 24. The symmetry boundary D condition was reproduced using frictionless supports. Displacement boundary condition E was applied to the bottom surface of the flange.

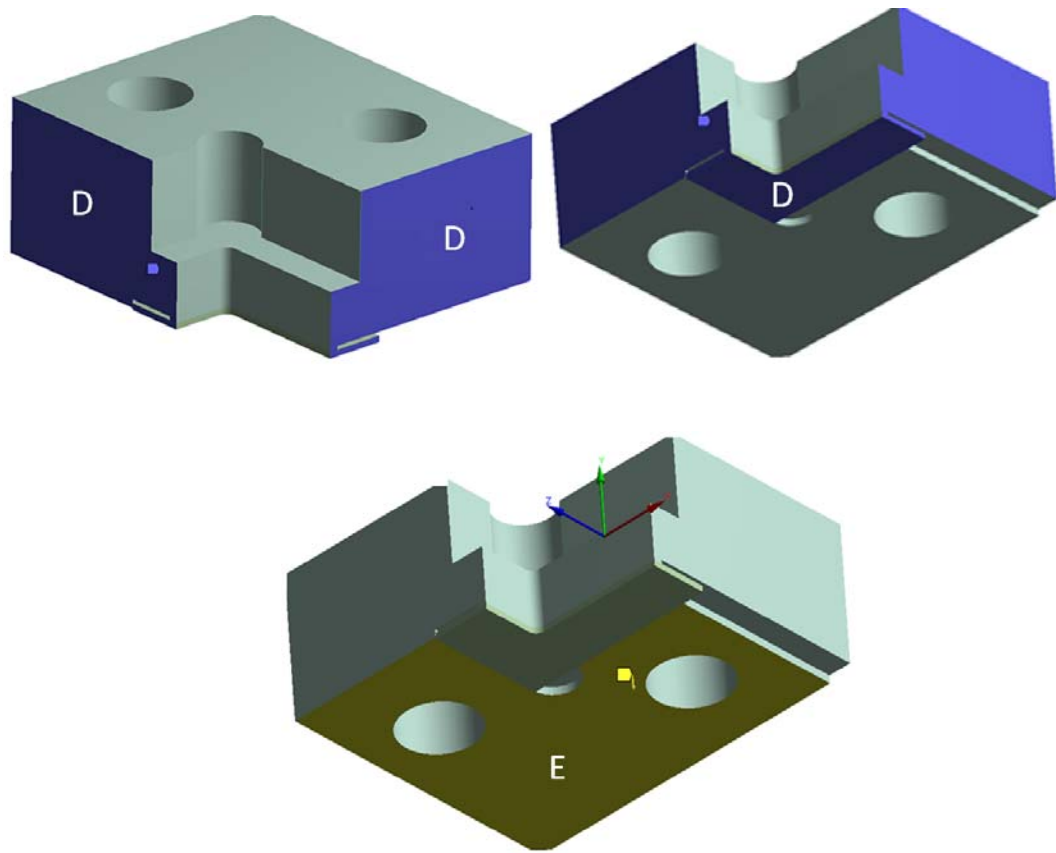


Figure 24. Symmetry boundary condition (D) and displacement boundary condition (E).

6 RESULTS

In this chapter the results of the finite element simulations using both 2D and 3D elements are presented and compared.

6.1.1 2D plane stress and strain modelling

Twelve versions of different flange knife dimensions were simulated.

Figure 25 shows deformations in the x direction for flange version 1 using plane strain elements. The elements near the flange knife were the most distorted during the simulation, thus leading to convergence problems, especially for versions having the highest final deformations. Table 4 presents the maximum deformation values of the gasket in the x direction for all the case studies, the maximum being at point A. From the results it can be observed that as the width and the length of the knife increase, the gasket deforms more. Version 12, having the largest flange knife dimensions, yields the maximum deformations (maximum 234 μm) of the gasket.

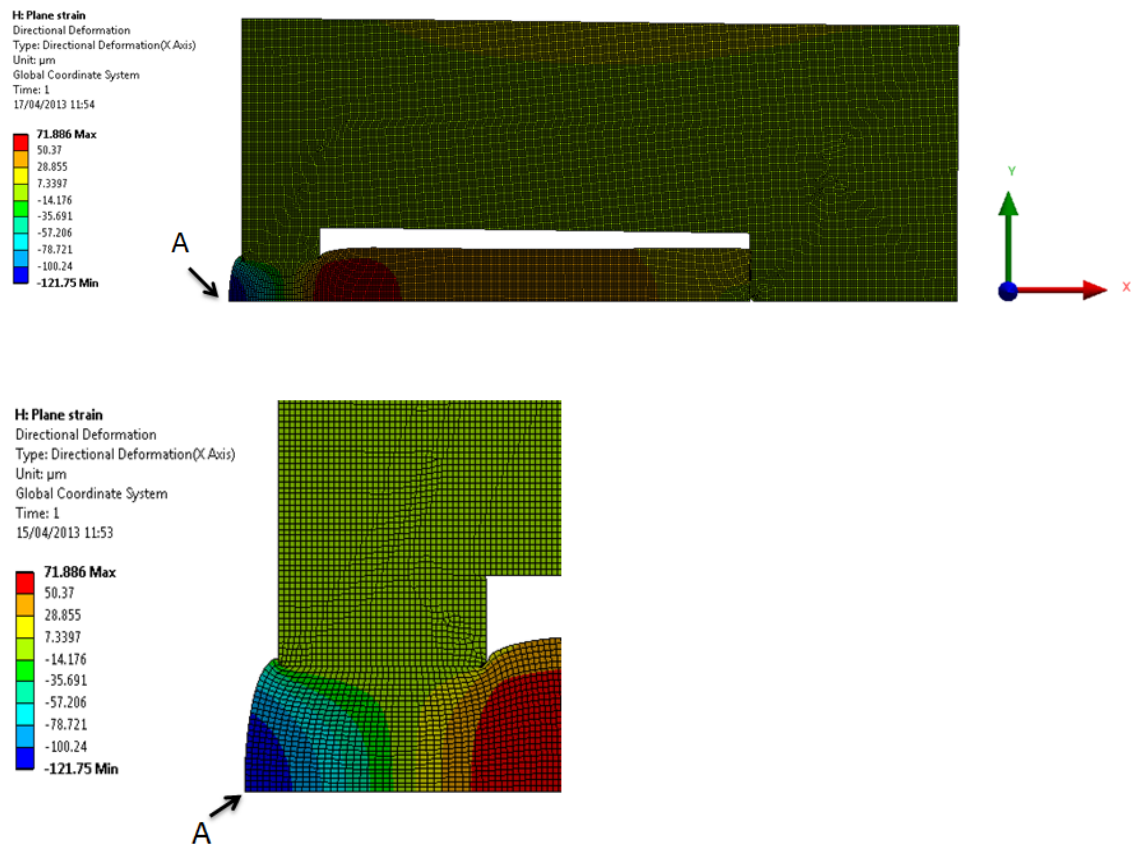


Figure 25. Top: Deformations in the x direction. Bottom: Detailed view of deformations in the flange knife area (version 1).

Table 4. Maximum deformation of the gasket in 2D plane strain model.

Version	a (mm)	b (mm)	Plane strain: directional deformation at point A (μm)
1	0.75	0.35	-122
2	0.75	0.3	-171
3	0.5	0.35	-101
4	0.5	0.3	-139
5	1	0.35	-140
6	1	0.3	-187
7	0.5	0.4	-59
8	0.5	0.25	-179
9	0.75	0.4	-78
10	0.75	0.25	-212
11	1	0.4	-89
12	1	0.25	-234

6.1.2 3D modelling

On the basis of the 2D results, three versions (1, 7 and 12) were selected to be modelled using 3D elements. The resulting deformations of the three versions are shown from Figure 26 to Figure 31. Table 5 presents the maximum deformation values of the gasket for the 3D analysis; these values are compared to the corresponding ones coming from the 2D analysis using plane strain elements. Because of the distorted elements generated during the simulations in the flange knife area, version 12 flange faced convergence problems causing the solver to stop at 64 % of the last step. The maximum deformation value presented for version 12 (440 μm) is an estimation and is predicted on the basis of the last step of the simulation. The deformations obtained in the 3D analysis are larger than the 2D counterparts. When the two faces S and L of the gasket are compressed, the resulting stresses generated by the constrained displacements are increasing due to higher deformation of the gasket

Table 5. Maximum deformation values for 2D plane strain and 3D models.

Version	2D plane strain model (μm)	3D model (μm)
1	126	210
7	59	98
12	234	440

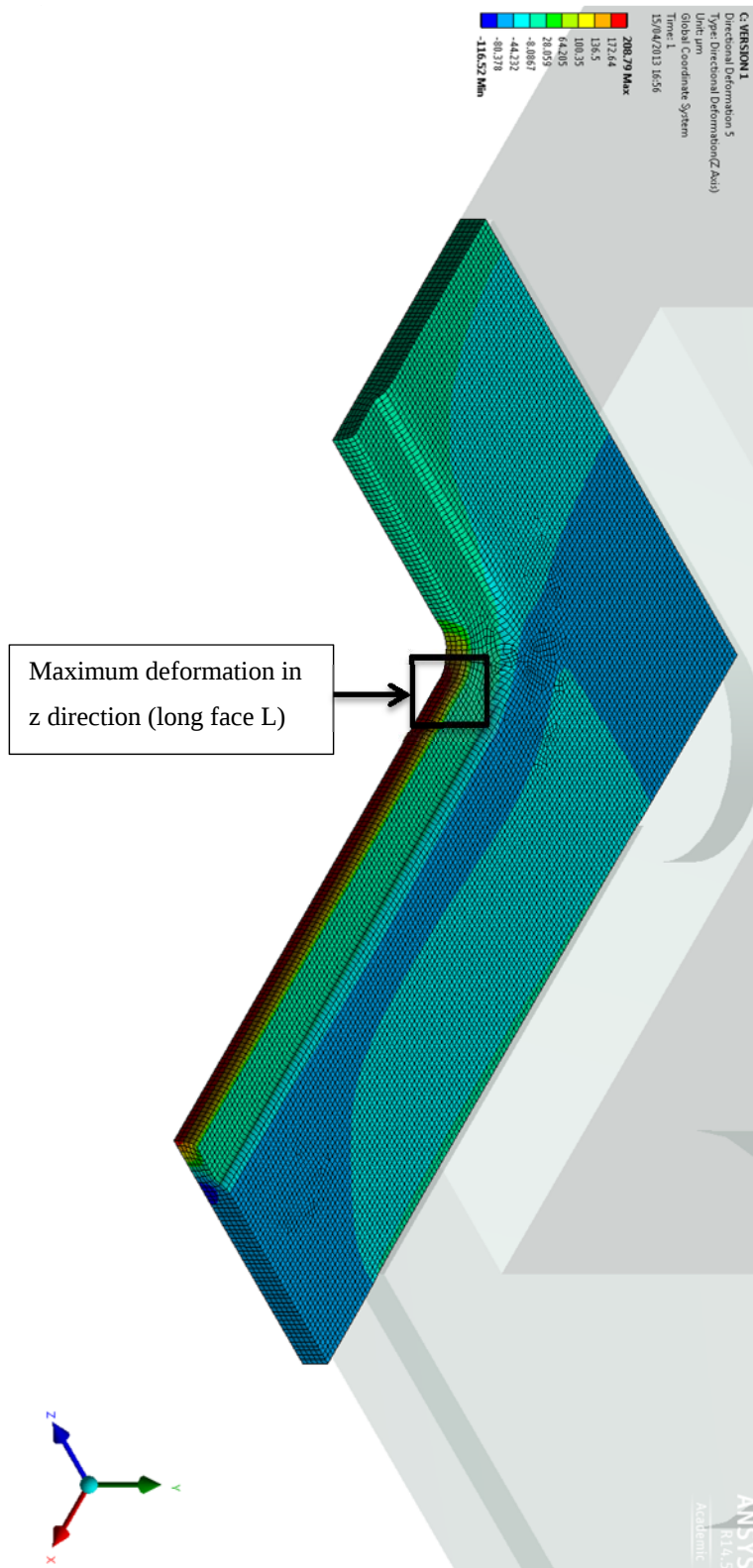


Figure 26. Version 1, directional deformations z-axis.

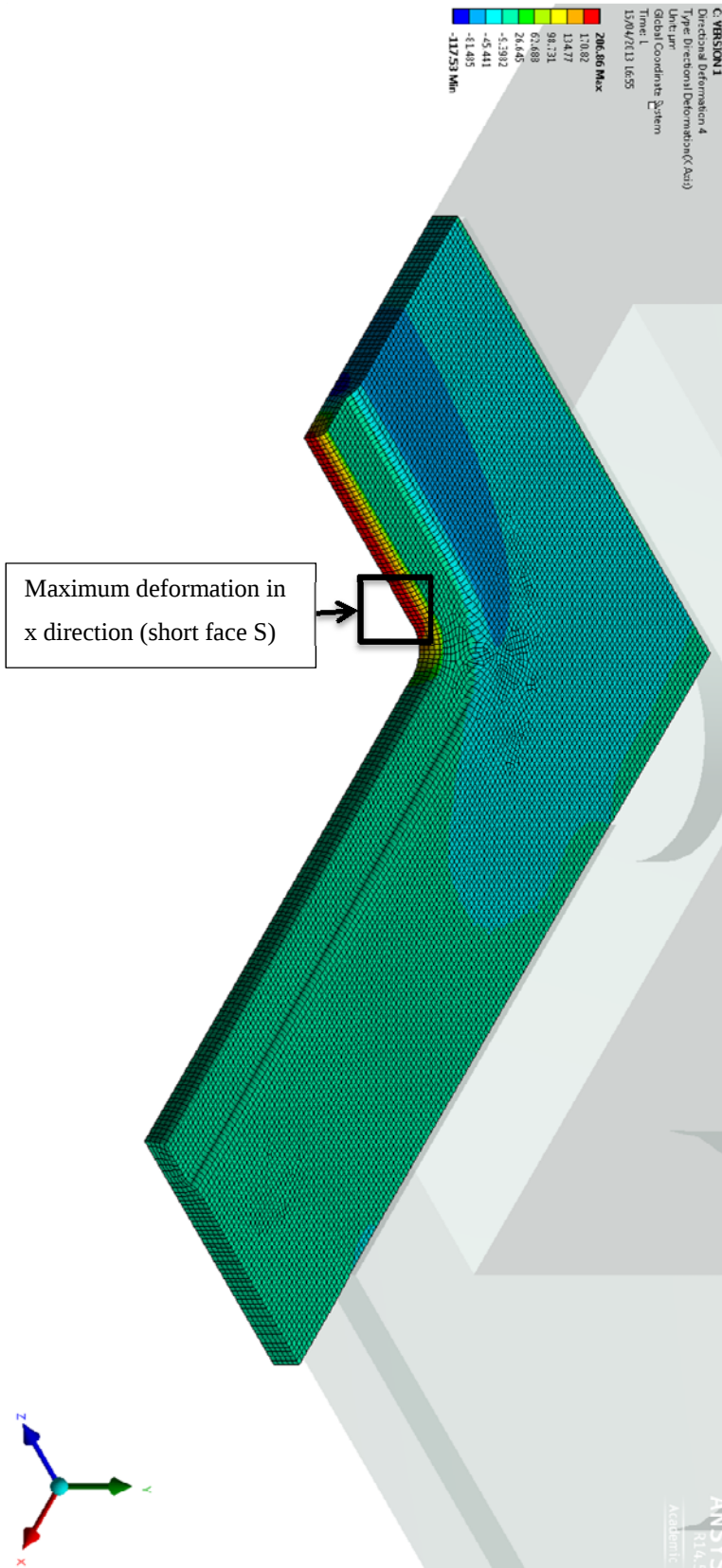


Figure 27. Version 1, directional deformations x-axis.

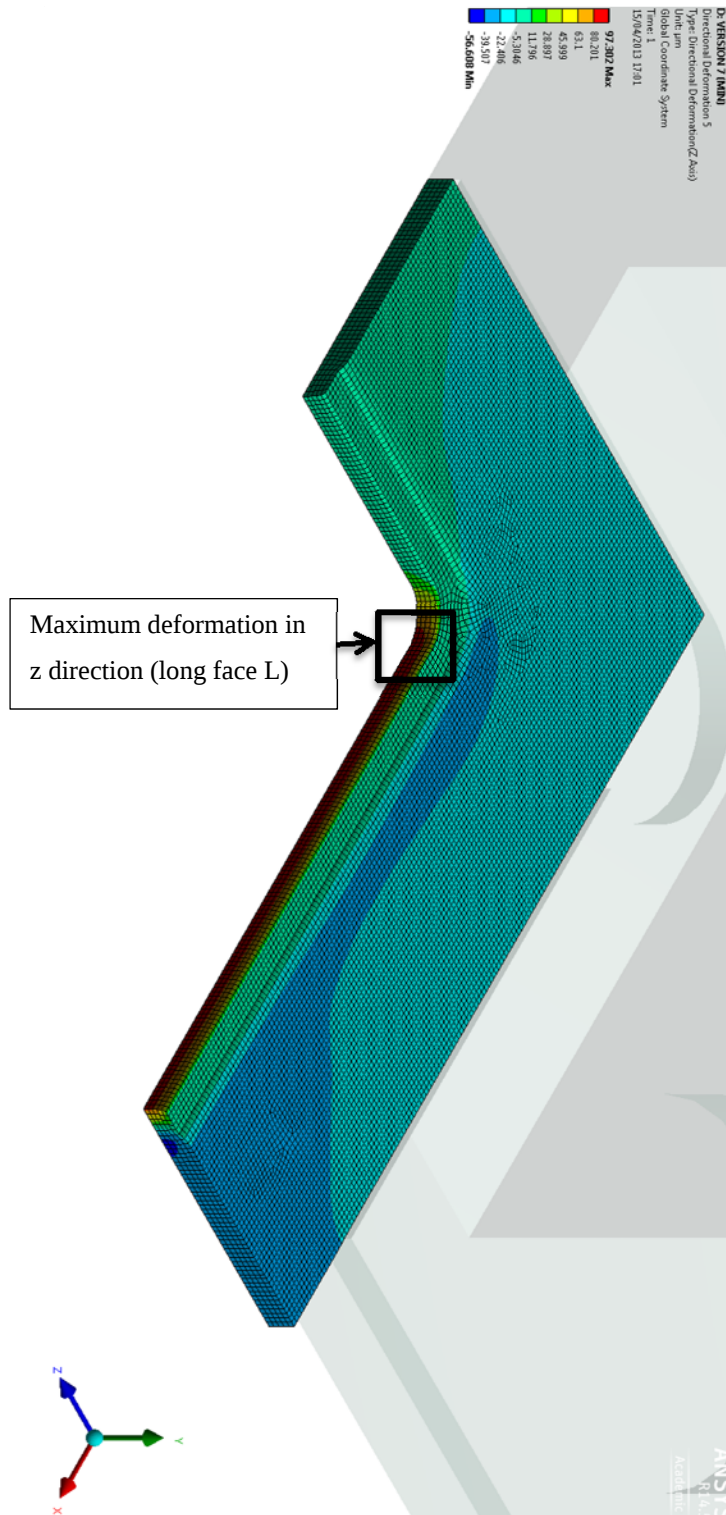


Figure 28. Version 7, directional deformations z-axis.

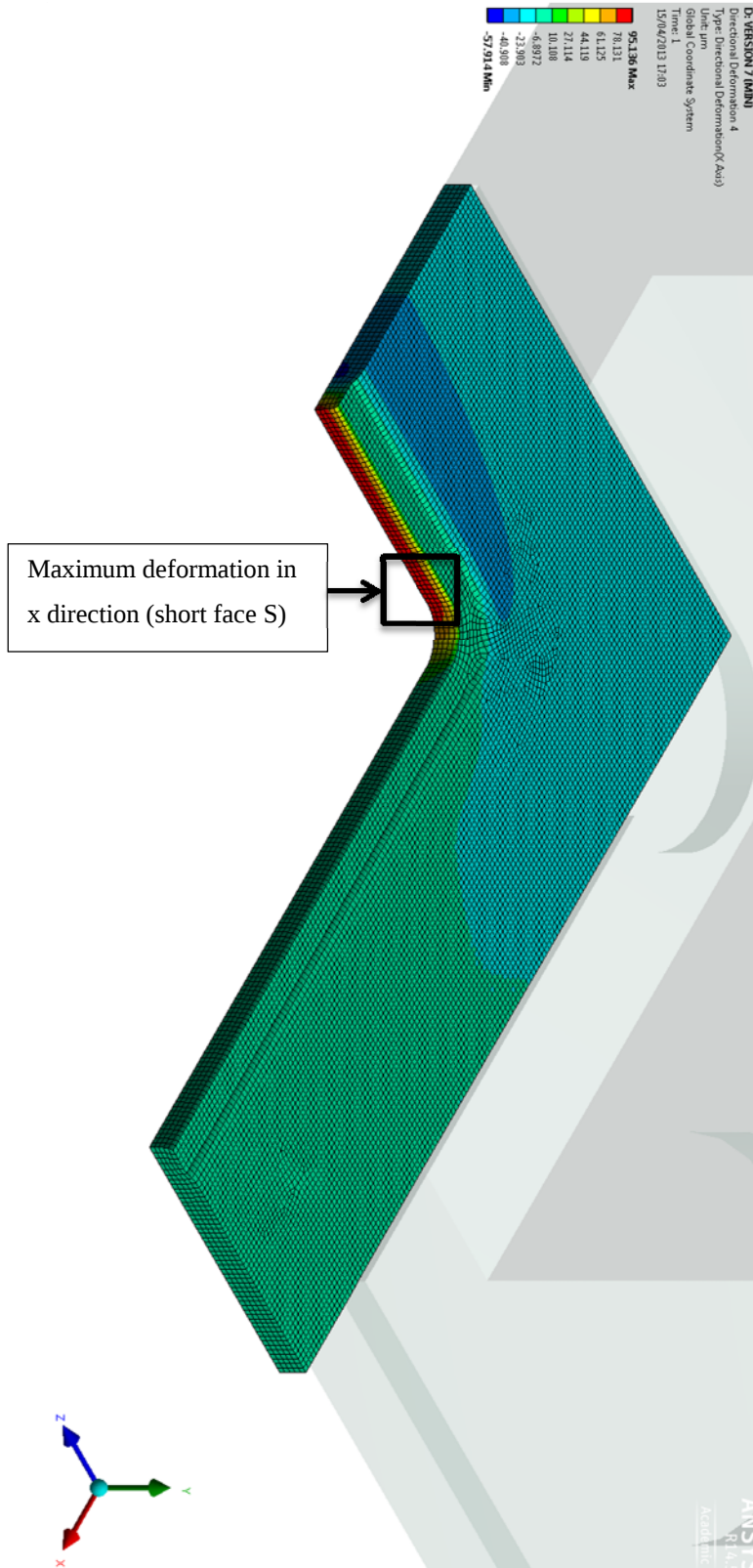


Figure 29. Version 7, directional deformations x-axis.

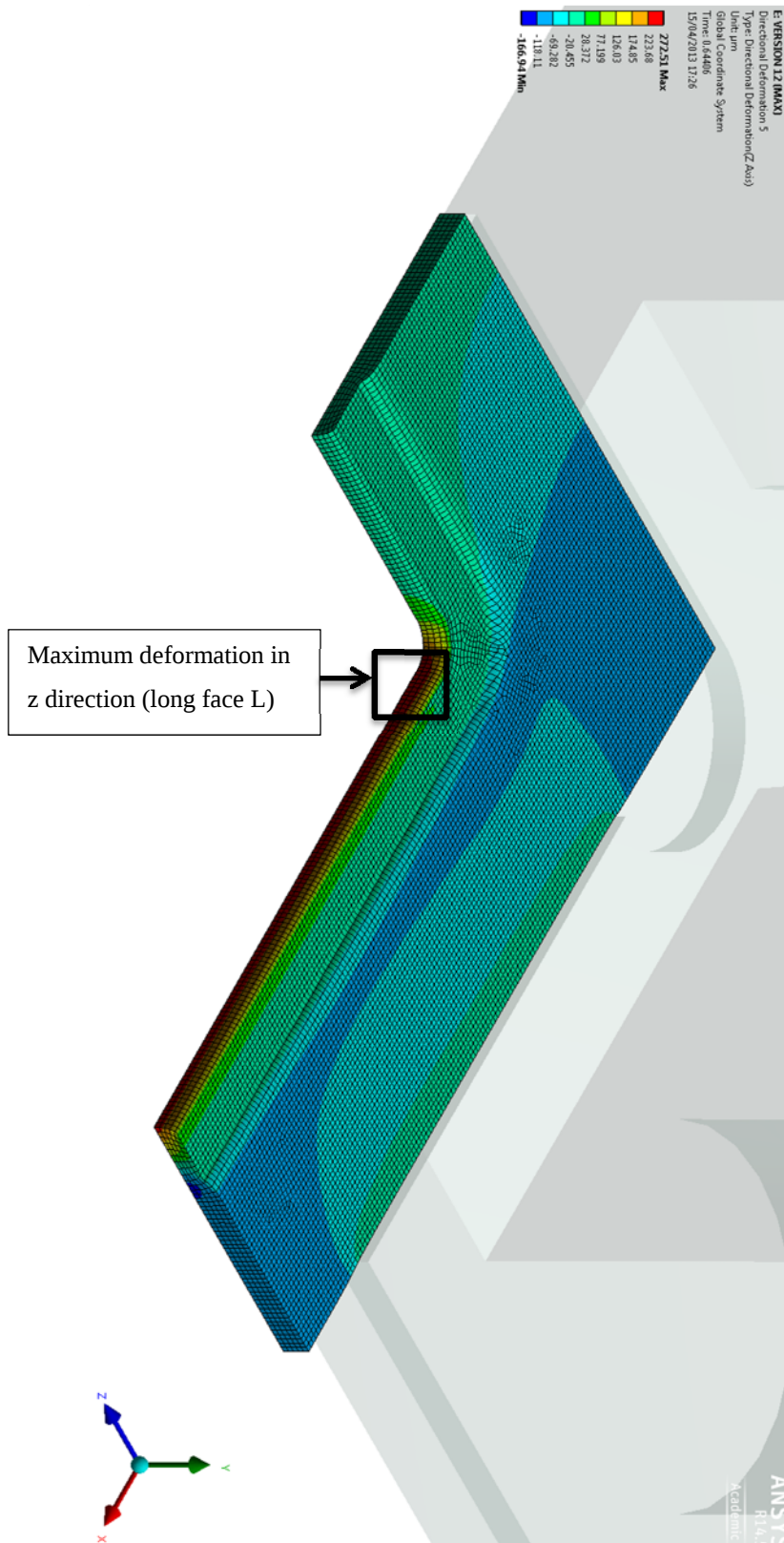


Figure 30. Version 12, directional deformations z-axis at time 0.64s.

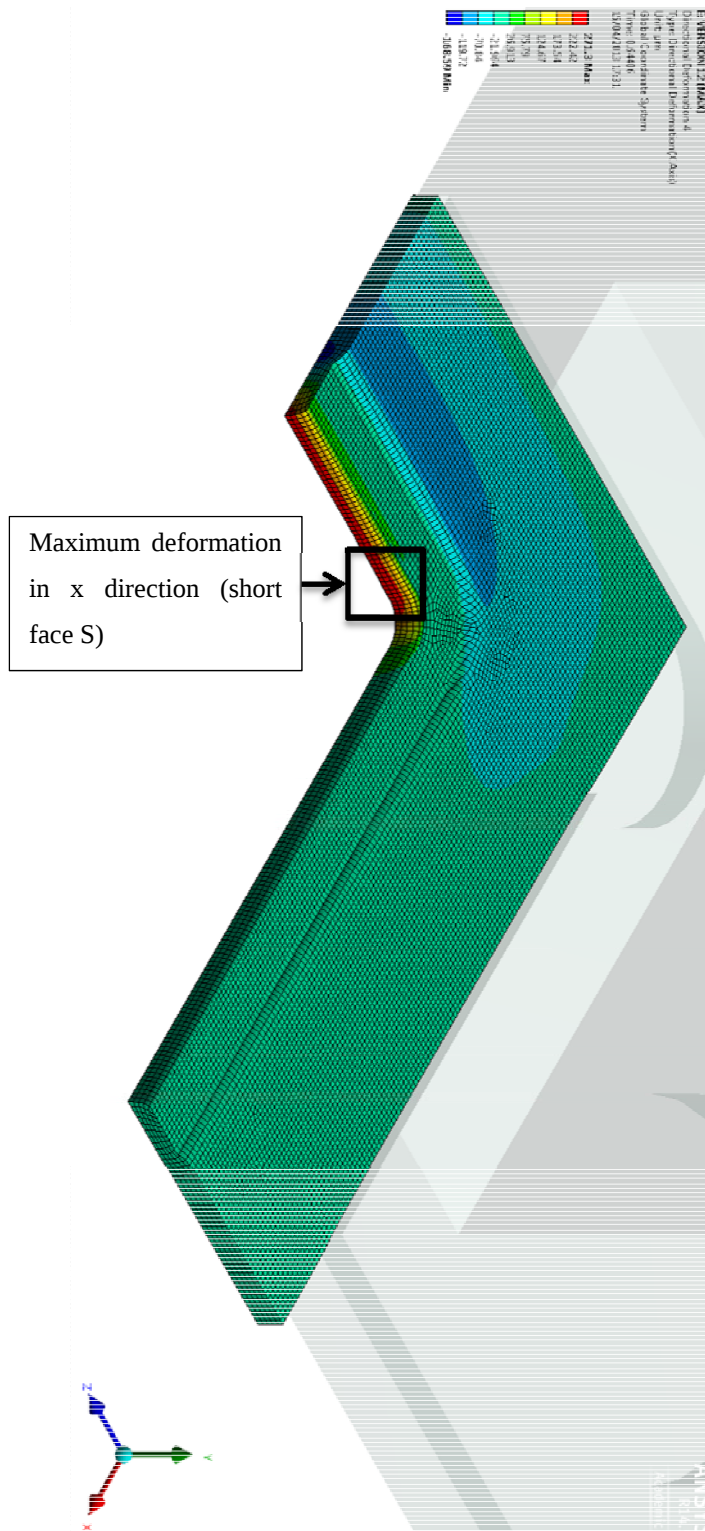


Figure 31. Version 12, directional deformations x-axis at time 0.64s.

7 CONCLUSIONS AND FUTURE STEPS

The newly designed x-band RF flange assembly was simulated with ANSYS FEA software. Twelve different versions of flanges were simulated using 2D plane stress and plane strain elements. Three of these versions were also modelled using 3D elements.

The resulting deformations obtained using plane stress elements were smaller than the ones obtained with plane strain elements; in particular in the first case, deformations perpendicular to the plane were allowed and no additional stresses were contributing to the total deformation of the gasket. By comparing these results with the ones obtained using solid elements, it can be seen that the deformations in the 3D analysis are even bigger, since the lateral faces of the flange taken into account in this modelling are constraining further the expansion of the gasket in the other directions, thus resulting in additional stresses. Overall, the deformations obtained for all the versions are meeting the specified requirements.

In the next future, experimental tests will be conducted at CERN and the theoretical results presented in this thesis will be validated.

8 REFERENCES

- Stapnes et al. (2012) A Multi-TeV linear collider based on CLIC technology: CLIC Conceptual Design Report. CERN 2012-007.
- Askeland et al. (2010) Essentials of Materials Science and Engineering. Stamford: Cengage Learning.
- Beer et al. (2012) Mechanics of Materials. New York: McGraw-Hill.
- Callister WD Jr (2001) Fundamentals of Materials Science and Engineering. New York: John Wiley & Sons.
- CERN (2013) About CERN. <http://home.web.cern.ch/about>. [1.4.2013]
- CERN (2013) Member states. <http://home.web.cern.ch/about/member-states>. [1.4.2013]
- CERN (2010) Physics for health. <http://physics-for-health.web.cern.ch/physics-for-health>. [1.4.2013]
- Ellis JR (2000) Physics Goals of the Next Century at CERN. CERN-TH-2000-050.
- Ellis JR & Wilson I (2000) New Physics with the Compact Linear Collider. CLIC-Note-466.
- Geschonke G (2010) The Next Energy-Frontier Accelerator a Linear e^+e^- Collider?. CERN-OPEN-2010-007. CLIC-Note-807.
- Martin J (2006) Materials for engineering. Cambridge: Woodhead Publishing Limited and Maney Publishing Limited.

Roylance D (2000) Introduction to Elasticity. Cambridge: Department of Materials Science and Engineering.

Smallman RE (1999) Modern Physical Metallurgy & Materials Engineering. Oxford: A division of Reed Educational and Professional Publishing Ltd.

ANSYS Workbench TM, ANSYS, Inc.

MatWeb. Material property data.

<http://www.matweb.com/Search/MaterialGroupSearch.aspx?GroupID=179>

[1.4.2013]

