Base Load Power Hub (BLPH)

TP - Joint Check Design Report

Document Number: 123A90-DDL-ST-RE-90012

Document Class: 1

02	06/11/2024	Accepted for Construction (AFC)	F. Biancheri	A. Balzarotti	D. LoSterzo
01	13/09/2024	Issue for Acceptance (IFA)	F. Biancheri	A. Balzarotti	D. LoSterzo
00	12/07/2024	Issue for Acceptance (IFA)	F. Biancheri	A. Balzarotti	D. LoSterzo
Rev.	Date	Description	Prepared	Reviewed	Approved
TECON					

This document undergoes an issuance cycle within EDMS Fusionlive". Its distribution is electronically controlled. Once printed shall be considered as "NOT CONTROLLED" copy

Revision / Change Log

Rev.	Description
00	Issue for Acceptance (IFA)
01	Issue for Acceptance (IFA)
02	Accepted for Construction (AFC)

HOLDS LIST

No.	Description
1	
2	
3	
4	

Table of contents

1 INTRODUCTION 8
1.1 PROJECT OVERVIEW 8
1.1.1 Project Location 8
1.1.2 Project Concept 9
2 DOCUMENT SCOPE 11
3 REFERENCES 12
3.1 CODES AND STANDARDS 12
3.2 COMPANY SPECIFICATIONS 12
3.3 PROJECT DOCUMENTS 13
3.4 SOFTWARE 13
4 DEFINITIONS AND ABBREVIATIONS 14
5 SUMMARY AND CONCLUSIONS 16
6 TRANSITION PIECE DESCRIPTION 17
7 MATERIALS 18
7.1 STRUCTURAL STEEL – LINEAR PROPERTIES 18
7.2 STRUCTURAL STEEL – NON-LINEAR PROPERTIES 19
8 DESIGN CRITERIA AND REQUIREMENTS 21
8.1 GENERAL 21
8.2 PARTIAL RESISTANCE FACTORS 22
8.2.1 Partial Resistance Factors for Tubular Sections: 22
8.2.2 Partial Resistance Factors for Non-tubular Sections 22
8.3 ACCEPTANCE CRITERIA 23
9 FE STRUCTURAL ANALYSIS 24
9.1 METHODOLOGY 25
9.1.1 ANSYS FE Model 26
9.1.2 Boundary Conditions 27

This document undergoes an issuance cycle within EDMS Fusionlive". Its distribution is electronically controlled. Once printed shall be considered as "NOT CONTROLLED" copy

9.1.3 Coordinate System 28

10 LIFTING PADEYE JOINT VERIFICATION 29

10.1 FE MODEL 29
10.1.1 Reference System 31
10.1.2 Material 32
10.1.3 Model Mesh 32
10.1.4 Constraints 34
10.1.5 Contacts 34
10.1.6 Loads 36

10.2 FEM ANALYSIS AND RESULTS 37

11 TP-STABBING GUIDE JOINT VERIFICATION 43

11.1 FE MODEL 44
11.1.1 Reference System 45
11.1.2 Material 45
11.1.3 Model Mesh 46
11.1.4 Constraints 48
11.1.5 Contacts 48
11.1.6 Loads 49

11.2 FEM ANALYSIS AND RESULTS 50
11.2.1 S460 Steel structural checks 52
11.2.2 S355 Steel structural checks 55
11.2.3 Deformations check 58

12 TP-TOPSIDE CONNECTION CHECK DURING SET DOWN 63

12.1 LOCAL CHECK 63

12.2 TP GLOBAL CHECK 64
12.2.1 Load combinations 64
12.2.2 Boundary Conditions 67

12.3 STRUCTURAL CHECKS 67
12.3.1 Member checks 67
12.3.2 Plate checks 69
12.3.3 ULS Joint checks 70

This document undergoes an issuance cycle within EDMS Fusionlive". Its distribution is electronically controlled. Once printed shall be considered as "NOT CONTROLLED" copy

List of Figures

Figure 1.1 - HOLLANDSE KUST NOORD WIND PARK Location Map 8
Figure 1.2 - Wind Park Details 9
Figure 1.3 – BLPH Concept 9
Figure 6.1 – Transition Piece – 3D 17
Figure 7.1 – Steel constitutive law according to DNV-RP-C208 20
Figure 9.1 - TP FE structural model 24
Figure 9.2 – Analyzed critical joints 25
Figure 9.3 – Mesh elements – SOLID186 26
Figure 9.4 – Mesh elements – SOLID187 27
Figure 9.5 – Mesh elements – SHELL181 27
Figure 10.1 – TP Padeye joint – 3D View 29
Figure 10.2 – TP Padeye Joint FE Model 30
Figure 10.3 – TP Padeye Joint FE Model – Reference system 31
Figure 10.4 – TP Padeye Joint FE Model – converging beams Reference systems 31
Figure 10.5 –FE Model – assigned materials 32
Figure 10.6 – TP Padeye Joint FE Model mesh – 3D isometric view 33
Figure 10.7 – TP Padeye Joint FE Model mesh – joint detail view 33
Figure 10.8 – TP Padeye Joint FE Model mesh – trunnion detail view 34
Figure 10.9 – TP Padeye Joint FE Model – external constraints 34
Figure 10.10 – TP Padeye Joint FE Model – bonded contact between cheek plate and weld 35
Figure 10.11 – TP Padeye Joint FE Model – friction contact between cheek plate and padeye main lug 35
Figure 10.12 – TP Padeye Joint FE Model – joint contact between 3D joint and 2D beam 36
Figure 10.13 – TP Padeye Joint FE Model – applied loads 37
Figure 10.14 – TP Padeye Joint FE model – VM Stresses 38
Figure 10.15 – TP Padeye Joint FE model – VM Stresses > 320 MPa 39
Figure 10.16 – TP Padeye Joint FE model – VM Stresses > 400 MPa 39
Figure 10.17 – TP Padeye Joint FE model – Peak VM Stresses at hot spot areas 40
Figure 10.18 – TP Padeye Joint FE model – Plastic strains > 0.1% 41
Figure 10.19 – TP Padeye Joint FE model – Total strains > 0.2% 42
Figure 11.1 – Stabbing-guide Joint – 3D view 43

This document undergoes an issuance cycle within EDMS Fusionlive". Its distribution is electronically controlled. Once printed shall be considered as "NOT CONTROLLED" copy

Figure 11.2 – Stabbing-guide FE Model 44
Figure 11.3 – Stabbing guide FE model – local impact Reference system 45
Figure 11.4 – Stabbing guide FE Model mesh – 3D isometric view 46
Figure 11.5 – Stabbing guide FE Model mesh – lower connection mesh 47
Figure 11.6 – Stabbing guide FE Model mesh – top detail view 47
Figure 11.7 – TP Stabbing Guide FE Model – bonded contact between cross plate and tube weld 48
Figure 11.8 – Stubbing guide FE Model – applied loads – impact condition 49
Figure 11.9 – Stubbing guide FE model – VM Stresses (front view) 50
Figure 11.10 – Stubbing guide FE model – VM Stresses (lower connection detail view) 51
Figure 11.11 – Stubbing guide FE model – VM Stresses (upper detail view) 51
Figure 11.12 – Stabbing guide FE model – S460 VM Stresses (front view) 52
Figure 11.13 – Stabbing guide FE model – S460 VM Stresses > 320 MPa – upper cone 52
Figure 11.14 – Stabbing guide FE model – S460 VM Stresses > 320 MPa – lower connection 53
Figure 11.15 – Stabbing guide FE model – S460 VM Stresses > 400 MPa 53
Figure 11.16 – Stabbing guide FE model – S460 VM Stresses > 320 MPa – upper cone peak stress areas . 54
Figure 11.17 – Stabbing guide FE model – S460 VM Stresses > 320 MPa – lower connection peak stress areas 55
Figure 11.18 – Stabbing guide FE model – S355 VM Stresses 55
Figure 11.19 – Stabbing guide FE model – S355 VM Stresses – internal plates/welds 56
Figure 11.20 – Stabbing guide FE model – S355 VM Stresses > 276 MPa 56
Figure 11.21 – Stabbing guide FE model – S355 VM Stresses > 345 MPa 57
Figure 11.22 – Stabbing guide FE model – Plastic strains 58
Figure 11.23 – – Stuabbing guide FE model – Plastic strains > 0 59
Figure 11.24 – Stabbing guide FE model – total strains 60
Figure 11.25 – Stabbing guide FE model – total strains > 0.2% 60
Figure 11.26 – Stabbing guide FE model – total strains > 0.2% upper cone 61
Figure 11.27 – Stabbing guide FE model – total strains > 0.2% lower connection 61
Figure 11.28 – Stabbing guide FE model – total strains > 5% 62
Figure 12.1 – WCR - topside 63
Figure 12.2 – TP-topside connection, ref. 123A90-DDL-ST-DD-90054 63
Figure 12.3 – Beveled section check 64
Figure 12.4 – TP plan view – Stabbing guides position 65

This document undergoes an issuance cycle within EDMS Fusionlive". Its distribution is electronically controlled. Once printed shall be considered as "NOT CONTROLLED" copy

Figure 12.5 – load condition DIA1 66
Figure 12.6 – load condition DIA2 67
Figure 12.7 – Member UC 68

List of tables

Table 4.1 - Definitions and Abbreviations 15
Table 7.1 - Material Selection, Ref /21/ 18
Table 7.2 – Mechanical properties of Plates 19
Table 7.3 – Mechanical properties of Structural Seamless Tubulars 19
Table 7.4 – Mechanical properties for Structural Sections 19
Table 7.5 – S420 steel constitutive law parameters 20
Table 7.6 – S355 steel constitutive law parameters 20
Table 8.1- Resistance Factors for Tubular Sections 22
Table 8.2- Resistance for Non Tubular Section 23
Table 10.1 – TP Padeye Joint FE model – mesh statistics 33
Table 10.2 – Summary Table - Slings internal actions: CF = 1.30 and ULS factor = 1.30 included. To be used for padeye design 36
Table 10.3 – TP Padeye Joint FE Model – applied loads 36
Table 11.1 – Stubbing guide FE Model – applied loads 49
Table 12.1 – Lifting Basic Combination TPLF 65

List of Attachments

This document undergoes an issuance cycle within EDMS Fusionlive". Its distribution is electronically controlled. Once printed shall be considered as "NOT CONTROLLED" copy

1 INTRODUCTION

1.1 PROJECT OVERVIEW

Crosswind, a Shell and Eneco joint venture, was awarded the tender to develop the offshore Hollandse Kust Noord (HKN) wind farm, located 18.5km off the coast of Netherlands near the town of Egmond aan Zee. The consortium plans to have HKN operational in 2023 with an installed capacity of 759MW generating at least 3.3TWh per year. This is sufficient renewable power to supply more than 1 million Dutch households with green electricity.

Offshore wind farms produce intermittent electricity and balancing these dips and peaks and integrating this electricity into the national energy system requires new technologies. For this reason, the offshore wind farm will include several new technologies that could be implemented at full-scale in the future. The new technologies being explored include a floating solar park, a short-duration battery storage as well as long-term storage via green hydrogen produced by water electrolysis and subsequently consumed in a fuel cell. The combination of these individual measures, herein referred to as the Base Load Power Hub (BLPH) will ensure a continuous power supply regardless of the wind situation.

1.1.1 Project Location

The Hollandse Kust Noord (HKN) wind farm is located 18.5km off the coast of the Netherlands near the town of Egmond aan Zee.

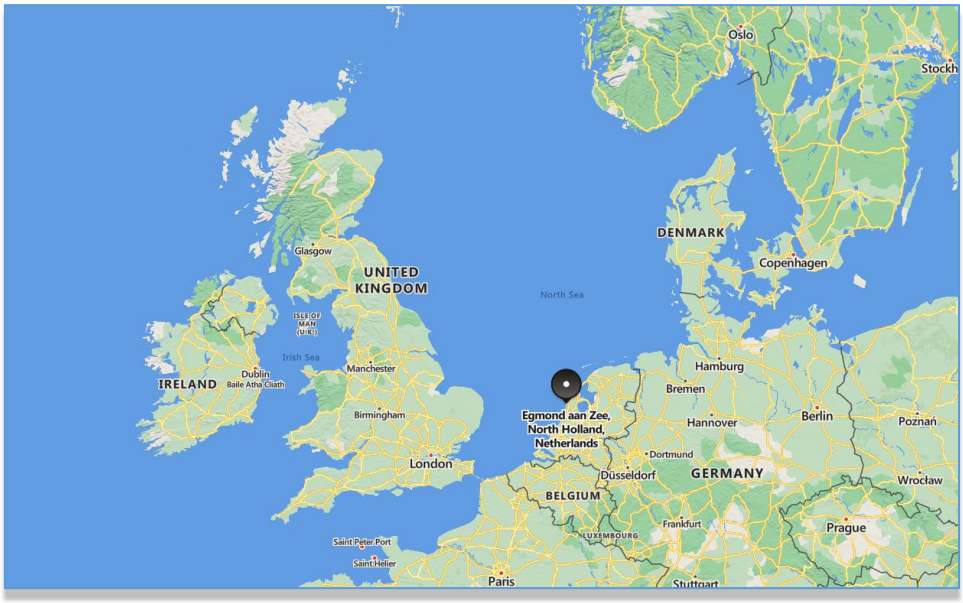


Figure 1.1 - HOLLANDSE KUST NOORD WIND PARK Location Map

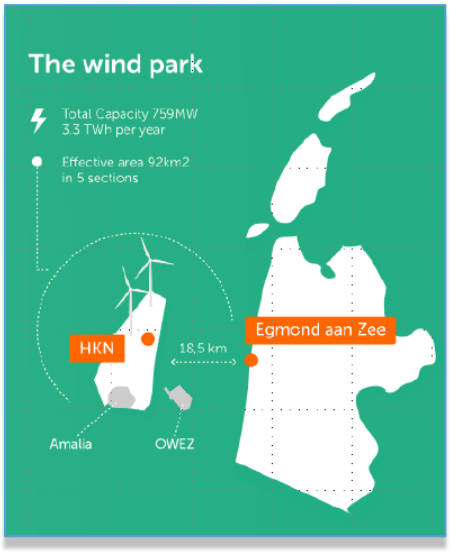


Figure 1.2 - Wind Park Details

The wind farm consist of No. 69 x 11.5MW wind turbine generators each to be installed on a dedicated monopile structure. A 70th monopile located on coordinates E 590719.1, N 5841602.3 is reserved for installing the platform accommodating the BLPH. The monopile structure was designed and installed by Van Oord Offshore Wind (VooW) and therefore not part of the BLPH scope.

1.1.2 Project Concept

All components included in the BLPH will be integrated onshore on a topsides and installed in an offshore location within the boundaries of the HKN wind farm. At its simplest, the BLPH will store energy during periods of high electricity generation from renewable sources (a wind turbine generator and the floating solar PV farm) and release it during periods of low generation. In practice. It is expected that the energy stored will supplement generation from renewable sources to ensure that at least 1.0MW of power is exported to the electrical grid regardless of wind/solar availability.

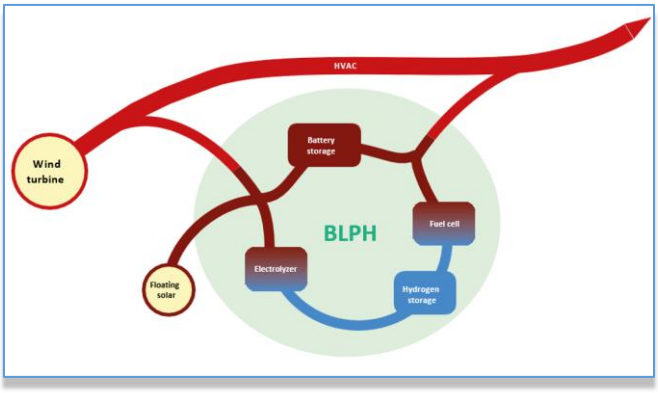


Figure 1.3 – BLPH Concept

Electricity in the BLPH will be stored in the form of electrons in a battery energy storage system (BESS) as well as converted into green hydrogen via water electrolysis. When the power output from the connecting wind turbine drops to below 1 MWe, the BLPH will return its stored energy to supplement the remaining power output. This will be by discharging the BESS, while stored hydrogen will be converted back into electricity using a fuel cell to extend the discharge duration.

The purpose of the BLPH s to function as a testbed for the integration of energy storage technologies with renewable generation. It is expected that equipment installed on the BLPH will operate as Normally Unmanned Installation for at least two (2) years with no or minimal maintenance and it will have its own independent control room with the ability to communicate with Crosswind's control room.

This document undergoes an issuance cycle within EDMS Fusionlive". Its distribution is electronically controlled. Once printed shall be considered as "NOT CONTROLLED" copy

2 DOCUMENT SCOPE

The objective of this report is to present the results of the detailed checks performed on the principal and most stressed joints of BLPH Transition piece structure for installation phase, e.g. lifting, installation and set down.

TP principal joints check for in-service and transportation conditions are detailed in report /26/ and /23/.

The analysis has been performed in compliance with the requirements, criteria and parameters reported in the Structural Design Premises, Ref./18/.

Structural Code checks are performed according to EN ISO 19902 for tubular members and tubular joints, Ref./6/, and according to Eurocode 3 for non-tubular structural elements, Ref./8/.

Structural analyses have been carried out using the structural analysis program SACS, the proprietary software developed by Bentley, Ref. /36/, and the Finite Element Analysis software ANSYS, developed by ANSYS Inc., Ref./37/.

The International System of Units (SI) has been used in all engineering assessments and calculations.

3 REFERENCES

3.1 CODES AND STANDARDS

Doc. No.	Description
/1/ BS EN ISO 19900	General requirements for offshore structures - 2019
/2/ BS EN ISO 19901-1	Specific requirements for offshore structures - Part 1: Metocean design and operating considerations - 2015
/3/ BS EN ISO 19901-3	Specific requirements for offshore structures – Part 3: Topsides structure - 2014
/4/ BS EN ISO 19901-5	Specific requirements for offshore structures – Part 5: Weight Management - 2021
/5/ BS EN ISO 19901-6	Specific requirements for offshore structures – Part 6: Marine Operations – 2009
/6/ BS EN ISO 19902	Petroleum and natural gas industries - Fixed steel offshore structures – 2020
/7/ ISO 19906:2019	Petroleum and natural gas industries – Arctic offshore structures
/8/ EN 1993-1-1	Design of steel structures - Part 1-1: General rules and rules for buildings
/9/ EN 1993-1-5	Design of steel structures - Part 1-5: Plated structural elements
/10/ DNV-ST-N001	DNV GL - Marine operation and marine warranty - 2021
/11/ DNV-ST-0145	DNV, Offshore substations
/12/ DNV-RP-C201	DNV, Buckling Strength of Plated Structures, October 2010.
/13/ DNV-RP-C208	DNV, Determination of structural capacity by non-linear finite element analysis methods, September 2021.
/14/ DNV-ST-0126	Support structures for wind turbine, 2018

3.2 COMPANY SPECIFICATIONS

Doc. No.	Description
/15/ DEP 37.19.00.12	Gen - Offshore Topsides Structures
/16/ DEP 37.19.00.30	Gen - Fixed Steel Offshore Structures (Adoption of IOGP S-631-11, Amendments / Supplements to ISO 19902:2020).
/17/ DEP 37.81.10.19	Gen - Marine Operations Requirements – Loadout, Transportation and Installation of Offshore Structures (Amendments / Supplements to ISO 19901-6)

3.3 PROJECT DOCUMENTS

Doc. No.	Description
/18/ 123A90-DDL-ST-SP-90017	Structural Design Basis
/19/ 123A90-DDL-MD-RE-01006	Monthly Weight Report
/20/ 123A90-DDL-PP-DS-70070	Plot Plans
/21/ 123A90-DDL-ST-SP-90018	Structural Material Specification for Topside and TP
/22/ 123A90-DDL-ST-RE-90000	BLPH Platform – In service Analysis
/23/ 123A90-DDL-ST-RE-90007	BLPH Platform – FE Analysis
/24/ 123A90-DDL-ST-RE-90009	TP – Transportation analysis Report
/25/ 123A90-DDL-ST-RE-90011	TP – Lifting Analysis
/26/ 123A90-DDL-ST-RE-90005	BLPH Platform – Joint check Design Report
/27/ 123A90-DDL-ST-TN-90119	Evaluating ISO and DNV approaches
/28/ 123A90-DDL-ST-DD-90078	TP – Primary Steel – Elevation - Row A
/29/ 123A90-DDL-ST-DD-90081	TP – Primary Steel – Elevation - Row B
/30/ 123A90-DDL-ST-DD-90083	TP – Flange Connection
/31/ 123A90-DDL-ST-DD-90085	TP – Plan El. +21710 TOS
/32/ 123A90-DDL-ST-DD-90086	TP – support points
/33/ 123A90-DDL-ST-DD-90087	TP – Lifting Points General Arrangement
/34/ 123A90-DDL-ST-DD-90088	TP – Lifting Points Details
/35/ 123A90-DDL-ST-DD-99042	TP – Installation AIDS - stabbing guide

3.4 SOFTWARE

Doc. No.	Description
/36/ SACS	Structural Analysis Computer System – BENTLEY Inc- v15
/37/ ANSYS	Ansys Inc.

4 DEFINITIONS AND ABBREVIATIONS

COMPANY or Client (CPY)	Crosswind
CONTRACTOR (CTR)	ROSETTI MARINO S.p.A.
PROJECT	HOLLANDSE KUST NOORD WIND PARK Project
T&I CONTRACTOR	DEME
ALS	Accidental Limit State
BESS	BATTERY ENERGY STORAGE SYSTEM
BG	Box Girder
BLPH	Base Load Power Hub
CL	Concentrated Load
CoG	Centre of Gravity
CW	CrossWind
DAF	Dynamic Amplification Factor
DLB	Ductility Level Blast
DNV	Det Norske Veritas
DP	Dynamic Positioning
El.	Elevation
EPCI	Engineering Procurement Construction and Installation
FEED	Front End Engineering Design
FEA	Finite Element Analysis
FEM	Finite Element Method
HAT	Highest Astronomical Tide
HKN	Hollandse Kust Noord
ICCP	Impressed Current Corrosion Protection
LAT	Lowest Astronomical Tide
LRFD	Load and Resistance Factor Design
MSL	Mean Sea Level
NTE	Not-to-Exceed
OD	Outside Diameter

PAU	Pre-Assembled Unit (H2 Vessels)
PG	Plate Girder
SHL	Static Hook Load
T&I	Transport & Installation
ToS	Top of Steel
UC	Unity Check
UDL	Uniformly Distributed Load
VM	Von Mises
VMS	Von Mises Stress
WTG	Wind Turbine Generator
W2W	Walk to Work

Table 4.1 - Definitions and Abbreviations

5 SUMMARY AND CONCLUSIONS

The analyses performed show that the TP joints, as configured, can withstand the maximum loads that they experience during Lifting and during Topside installation (both impact load and set-down)

Checks are executed according to the criteria imposed by ISO code 19902, Ref. /6/, Eurocode 3, Ref./8/ and DNV-ST-N001, Ref. /10/.

Analysed joints are:

1)	Lifting padeye:	Lifting Conditions, Ref. /25/
2)	TP joint with primary stabbing guide during installation	Installation condition, Ref. /18/, sect.10.6
3)	TP-topside connection during set-down	Installation condition, Ref. /18/, sect 10.6

The results of the Analyses are summarized in the following:

1) The TP Padeye is checked against the loads imposed by the lifting conditions, taking into account the maximum sling force. The FE analysis has shown that globally Von-Mises stresses are largely lower than the allowable stresses, except for some local peaks occurring in the internal edge of the padeye and the cheek plates where load is applied. Elasto-plastic strains have been investigated, and they result to be lower than the admissible value 5%. According to these results, the Padeye is correctly designed to withstand the loads imposed by the lifting condition.
2) The primary stabbing guide is checked against the loads imposed by the installation conditions (impact load). The FE analysis has shown that globally Von-Mises stresses are largely lower than the allowable stresses, except for some local peaks occurring at the load application surface (impact area) and at the base connection to the TP joint. Elasto-plastic strains have been investigated, and they result to be lower than the admissible value 5%. According to these results, the stabbing guide is correctly designed to withstand the impact load of the installation condition. Performed checks cover secondary stabbing guide, since the load it takes is 50% of that of the primary one, and the geometry is the same.
3) The can, prepared with 20mm bevel at 45 deg, can support the vertical and transversal forces acting during the topside set-down. The design load has been calculated by distributing the topside weight proportionally to the support point distance from CoG. Any two diagonally opposite support were designed to withstand 100% of the topside weight. The max UC against bearing is 0.74, against shear is 0.22.

This document undergoes an issuance cycle within EDMS Fusionlive". Its distribution is electronically controlled. Once printed shall be considered as "NOT CONTROLLED" copy

6 TRANSITION PIECE DESCRIPTION

The connection of Topside with Monopile is realized through a **Transition Piece (TP)** structure. TP is composed by a central column with an outer diameter of 7m and Four (4) boxed girders with height ranging from 2.5 m to 1.4 m, oriented along the main diagonals of the TP.

TP main dimensions are 16m x 16m with height 7.9m.

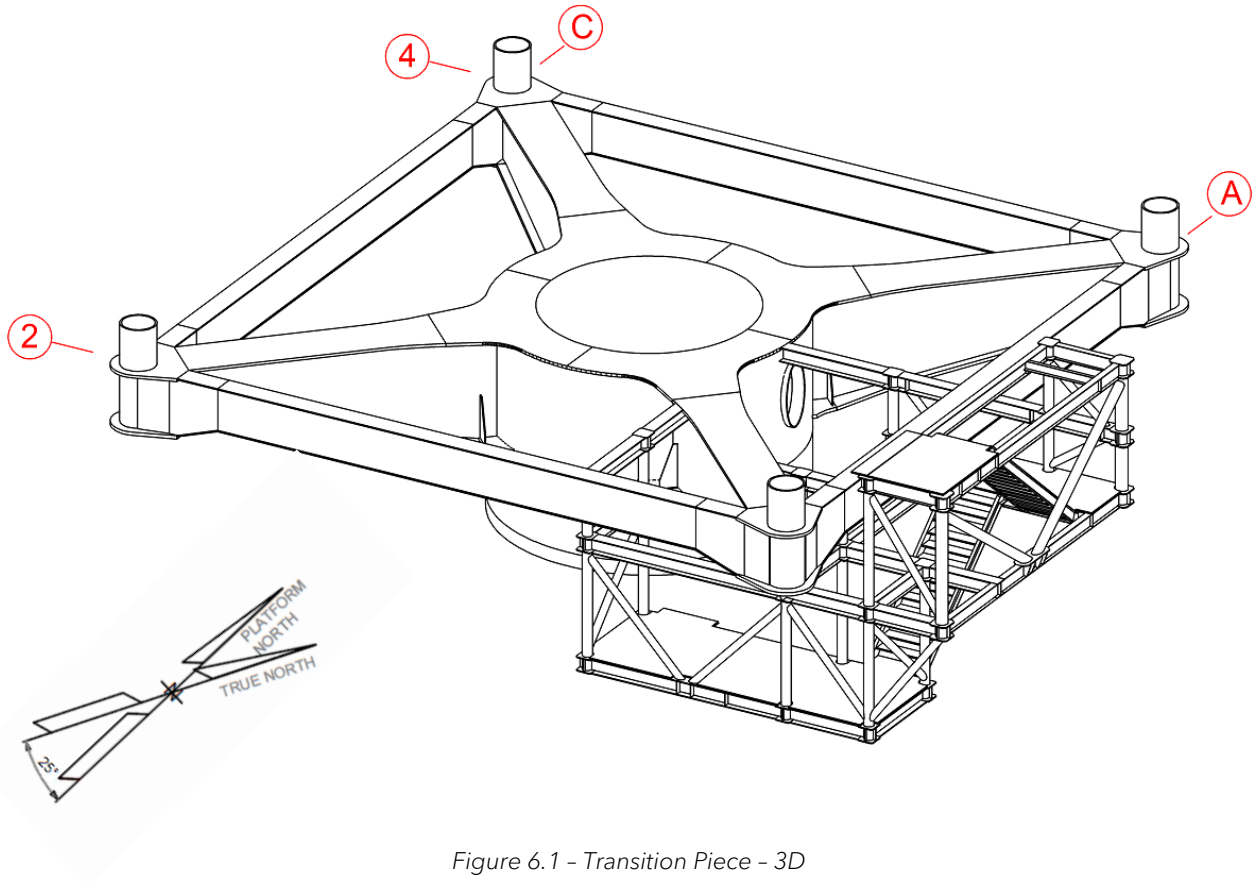


Figure 6.1 – Transition Piece – 3D

7 MATERIALS

7.1 STRUCTURAL STEEL – LINEAR PROPERTIES

For the structural analyses, steel types and grades to be used for the platform structures are summarised in Table 7.1 and specified in Ref. /21/. The material selection criteria are based on the requirements of ISO 19906:2019, ISO 19902:2020 and ISO 19901-3:2014. The materials are in accordance with the EN standard 10225:2019 and 10025:2019 and are suitable for plates, rolled sections and welded / seamless tubulars.

			EN10025-3	ISO EN 19902:2020						
Design Class	**Complexity**	**SQL**	**Hot rolled bar**	**Plates and Welded Tubular**	**Sections**	**SMLS pipes**	**THK**	**CVN inpact test**	**Material Designation**	**List of Options (EN10225-1 to -4) & EN10025-3**
DC1 TP to monopile flange connection	High	I (Z properties)	EN 10025 S420 NL	NA	NA	NA	t > 50	-40	DC1-I-420-40_Z	1, 2, 3, 4, 6, 12, 18, 24, 30
DC1 Complex lift-nodes and element connection to topside, Lift points, main complex joints	High	I (Z properties)	NA	EN 10225 S420 QLO / ML	NA	NA	25 < t <= 50	-40	DC1-I-420-40_Z DC1-I-460-40_Z	2, 7, 8, 11, 12, 17, 18
							t > 50	-40		
DC2 simple joints, primary deck girders and beams of Topside and TP, column and diagonal to lift points, main columns, Davit crane support, Bolts	Low	II	NA	EN 10225 S420 QLO / MLO	NA	NA	t < = 12	0	NA	-
							12 < t <= 25	-20	DC2-II-420-20 / DC2-II-460-20	2, 7, 8, 11 17 for t > 40mm
							25 < t <= 50	-40	DC2-II-420-40 DC2-II-460-40	
							t > 50	-40		
DC3 all other columns and diagonals, stiffeners, Secondary deck beams, Grillage and sea fastening	High	II	NA	EN 10225 S355 NLO / MLO	EN 10025 S355 MLO	EN 10225 S355 NLHHO / QLHHO API 5L X52	t < = 12	0	DC3-II-355-0 / DC3-II-X52-0	2, 7, 8 11 for plate 17 for t > 40mm
							12 < t <= 25	-20	DC3-II-355-20 / DC3-II-X52-20	
							25 < t <= 50	-40	DC3-II-355-40 / DC3-II-X52-40	
							t > 50	-40		
DC4 Equipment and vessel supports, modules supports, equipment fixing bolts	Low	III	NA	EN 10025 S355 J2	EN 10025 S355 J2	API 5L X52 EN 10210 S355 J2H	t < = 12	0	DC4-III-355-0 / DC4-III-X52-0 / DC4-III-X60-0	-
							12 < t <= 25	0		
DC5 floor plating, grating, Walkways, stairs, handrails	Any	IV	NA	EN 10025 S235 JR EN 10025 S275 J0 EN 10025 S355 J0	EN 10025 S235 JR EN 10025 S275 J0 EN 10025 S355 J0	API 5L Gr.B EN 10210 S235 JRH / EN 10210 S275 J0H / EN 10210 S355 J0H	t < = 12	10	DC5-IV-235-10 / DC5-IV-275-10 / DC5-IV-355-10 / DC5-IV-GRB-10	-
							12 < t <= 25	10		

Table 7.1 - Material Selection, Ref /21/

Mechanical properties for structural materials are presented in Table 7.2, Table 7.3 and Table 7.4.

- Young's Modulus = 205000 N/mm^2
- Density = 7850 kg/m^3
- Poisson's Ratio = 0.3

Thickness (mm)	Min. Tensile Strength (N/mm2)	Min. Yield Strength (N/mm2)	Steel Type (1)
t ≤ 16	500	420	I & II
16 < t ≤ 40	500	400	I & II
40 < t ≤ 63	480	390	I & II
63 < t ≤ 100	480	380	I & II
Note: 1) Type I steel has Z (through the thickness) properties.			

Table 7.2 – Mechanical properties of Plates

Thickness (mm)	Min. Tensile Strength (N/mm²)	Min. Yield Strength (N/mm2)	Steel Type
t ≤ 20	460	355	III
20 < t ≤ 40	460	345	III

Table 7.3 – Mechanical properties of Structural Seamless Tubulars

Thickness (mm)	Min. Tensile Strength (N/mm2)	Min. Yield Strength (N/mm2)	Steel Type
t ≤ 16	460/360	355/275	IV
16 < t ≤ 40	460	345	IV

Table 7.4 – Mechanical properties for Structural Sections

7.2 STRUCTURAL STEEL – NON-LINEAR PROPERTIES

For non-linear FEAs, the assumed constitutive law for steel is according to DNV-RP-C208, Ref. /13/.

The assumed constitutive law for both materials is the one proposed by DNV-RP-C208 (Ref./13/) for S420 steel:

$$\sigma = K\left(\varepsilon_p + \left(\frac{\sigma_{yield\ 2}}{K}\right)^{\frac{1}{n}} - \varepsilon_{p_y2}\right)^n \quad \text{for} \quad \varepsilon_p > \varepsilon_{p_y2}$$

Parameters of the previous equations for S420 are reported in Multilinear Isotropic Hardening constitutive law.

S420			
Thickness [mm]	*t≤ 16*	*16< t ≤ 40*	*40< t ≤ 63*
E [MPa]	210000	210000	210000
σ_{prop} **[MPa]**	378.7	360.6	351.6
σ_{yield} **[MPa]**	422.5	402.4	392.3
σ_{yield2} **[MPa]**	426.3	406	395.9
ε_{p_y1}	0.004	0.004	0.004
ε_{p_y2}	0.012	0.012	0.012
K[MPa]	738	703	686
n	0.14	0.14	0.14

Table 7.5 – S420 steel constitutive law parameters

	S355			
Thickness [mm]	*t≤ 16*	*16< t ≤ 40*	*40< t ≤ 63*	*63< t ≤ 100*
E [MPa]	210000	210000	210000	210000
σ_{prop} **[MPa]**	320.0	311.0	301.9	284
σ_{yield} **[MPa]**	357.0	346.9	336.9	316.7
σ_{yield2} **[MPa]**	366.1	355.9	345.7	323.8
ε_{p_y1}	0.004	0.004	0.004	0.004
ε_{p_y2}	0.015	0.015	0.015	0.015
K[MPa]	740	740	725	725
n	0.166	0.166	0.166	0.166

Table 7.6 – S355 steel constitutive law parameters

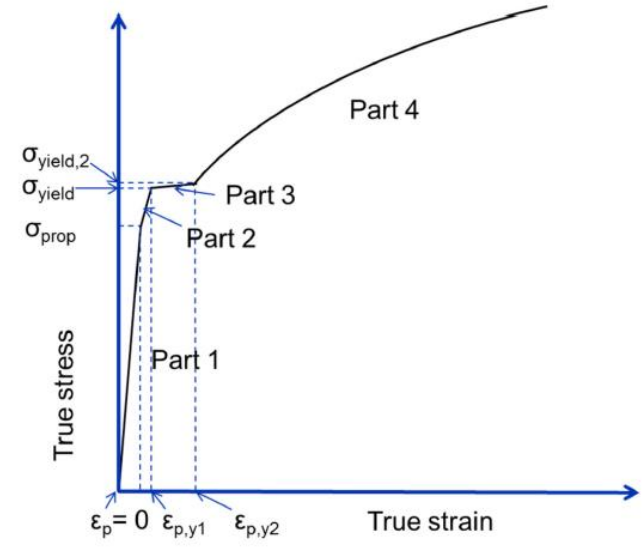


Figure 7.1 – Steel constitutive law according to DNV-RP-C208

8 DESIGN CRITERIA AND REQUIREMENTS

8.1 GENERAL

Structures in SOW is designed in accordance with limit states approach of ISO codes 19901 and 19902, Refs./2/, /3/and /6/. For non-tubular sections, Eurocode, Ref./8/, is adopted.
Member forces and moments derived from the global analyses are used for checking member strength and stability, and joint strength.

The general requirement of ISO code, ref./6/, is that, for the structure as a whole and for each component, the following is satisfied:

$$R_d = \frac{R_k}{\gamma_M} > \gamma_F F_r = F_d$$

Where:

- R_d is the design resistance;
- R_k is the characteristic resistance;
- γ_M is the partial resistance factor reported in section 8.2;
- F_d is the design value of action;
- F_r is the characteristic value of action;
- γ_F is the partial action factor.

Design according to the provisions for load and resistance factor design (LRFD) of EC3, ref. /8/, shall be performed in accordance with following equation:

$$R_u \leq \phi R_n$$

Where:

- R_u = required strength using LRFD load combinations
- R_n = nominal strength
- ϕ = resistance factor reported in Sec.8.2
- ϕ Rn = design strength.

8.2 PARTIAL RESISTANCE FACTORS

8.2.1 Partial Resistance Factors for Tubular Sections:

The partial resistance factors for tubular sections from ISO 19902, Ref./6/, are listed below.

Resistance factor	Value	Definition
$\gamma_{R,t}$	1.05	Partial resistance factor for axial tensile strength
$\gamma_{R,c}$	1.18	Partial resistance factor for axial compressive strength
$\gamma_{R,b}$	1.05	Partial resistance factor for bending strength
$\gamma_{R,v}$	1.05	Partial resistance factor for shear strength
$\gamma_{R,h}$	1.25	Partial resistance factor for hoop buckling strength

Table 8.1- Resistance Factors for Tubular Sections

8.2.2 Partial Resistance Factors for Non-tubular Sections

Referring to ISO 19901-3, Ref./3/, section 8, to be consistent with ISO 19900 inherent the level of safety, a "building code correspondence factor" is to be applied to the AISC partial factors.

The partial resistance factors for non-tubular sections from Eurocode 3, Ref./8/, are listed below.

Resistance factor	Value	Definition
Resistance of members and cross sections		
γ_{M0}	1.00	Cross-section resistance to excessive yield strain including local buckling
γ_{M1}	1.00	Resistance of unstable elements measured during element inspection
γ_{M2}	1.25	Tensile cross-section resistance to mechanical breakdown
Resistance of connections		
γ_{M2}	1.25	Bolts, rivets, pins, welds and slabs on supporting surface
γ_{M3}	1.25	Sliding resistance at Ultimate Limit State Category C
$\gamma_{M3,SER}$	1.10	Sliding resistance at Serviceability Limit State
γ_{M4}	1.10	Injection bolt collapse resistance

γ_{M5}	1.10	Resistance of connections in open lattice trusses
$\gamma_{M6,SER}$	1.00	Resistance of pins at Serviceability Limit State
γ_{M7}	1.10	Preload of high strength bolts

Table 8.2- Resistance for Non Tubular Section

8.3 ACCEPTANCE CRITERIA

FE checks are performed comparing the resulting Von Mises stresses for each analysis with the allowable stresses defined as per Eurocode, Ref./8/, and ISO 19902, Ref./6/. Allowable stresses in terms of ultimate strength (LRFD) are defined using the material strengths listed in Sec.7.1 and the partial resistance factors listed in Sec.8.2:

$\sigma_{VM} \leq \sigma_{adm} = F_Y/\gamma$

Being:

- σ_{VM} = Von Mises Stress
- F_y = Material Yielding Stress
- γ = Partial resisting factor

Stresses greater than the admissible limits are accepted up to the material yield strength F_y, provided that such stresses remain confined to small superficial areas (2tx2t, being t the thickness of the plate considered), thus demonstrating that the corresponding resisting section is not significantly affected.

In the areas where stress checks are not satisfied, plastic checks are performed.

The total strain must fulfill the criterium according to EC3-1-5 (Ref./9/) and must be lower than 5%:

$\varepsilon_t = \leq \varepsilon_{adm} = 0.05$ mm/mm

Being:

- ε_t = Total Strain (elastic + plastic)

The fulfilment of one of the two checks (VM stress, strain) guarantees the structural integrity of the considered joint.

9 FE STRUCTURAL ANALYSIS

The Ansys model of the Transition Piece is displayed in Figure 9.1. This model is used for the In-Service analysis, as per relative report (Ref./23/).

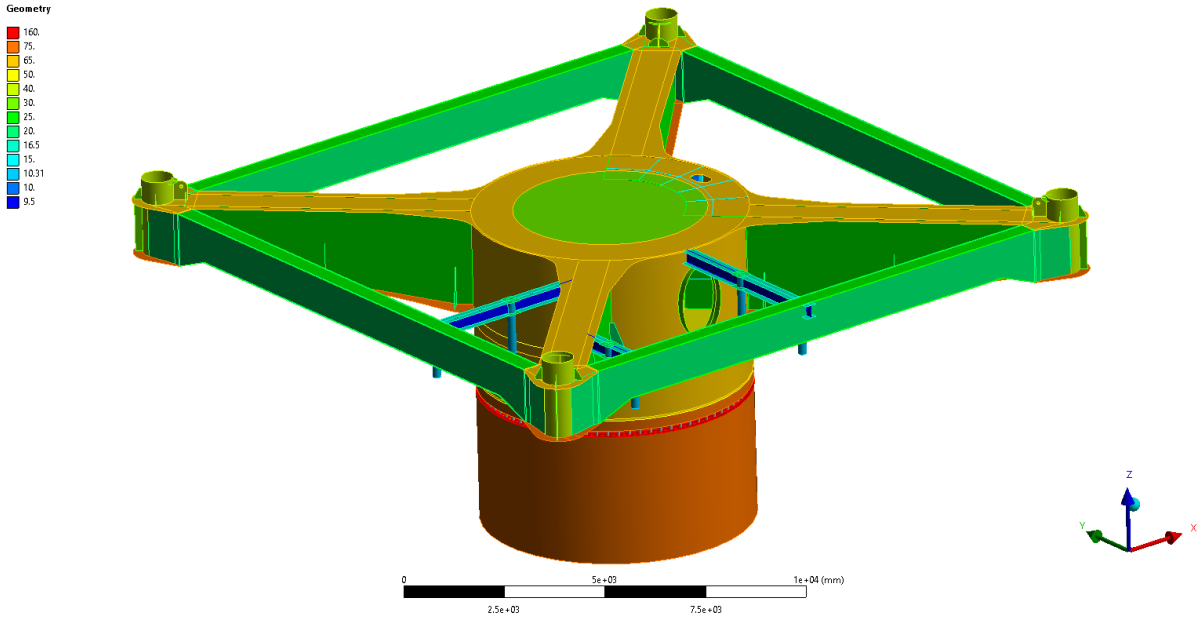


Figure 9.1 - TP FE structural model

The local analysis of the lift point and stabbing guide, using FE approach, is carried out within Ansys Environment Ref./37/, creating a local model of the joint according to structural drawings, from the model displayed in Figure 9.1.

The local models of the selected joints have been meshed using 2D and/or 3D elements, following a Finite Element approach. Every element is analysed separately, and a single analysis is performed for each node.

Every member converging at the node is modelled, and additional top and bottom plates, stiffeners, gusset plates that are present in the node are added to the model.

The analysed joints are the following:

- Lifting padeye: Lifting Conditions, Ref. /25/
- TP joint with primary stabbing guide during installation Installation condition, Ref. /18/, sect.10.6

The TP-Topside connection check is performed in section 12 considering 20mm bevel preparation at 45°, and therefore 20mm of thk only.

- TP-topside connection during set-down Installation condition, Ref. /18/, sect 10.6

These joints are highlighted in the following figures.

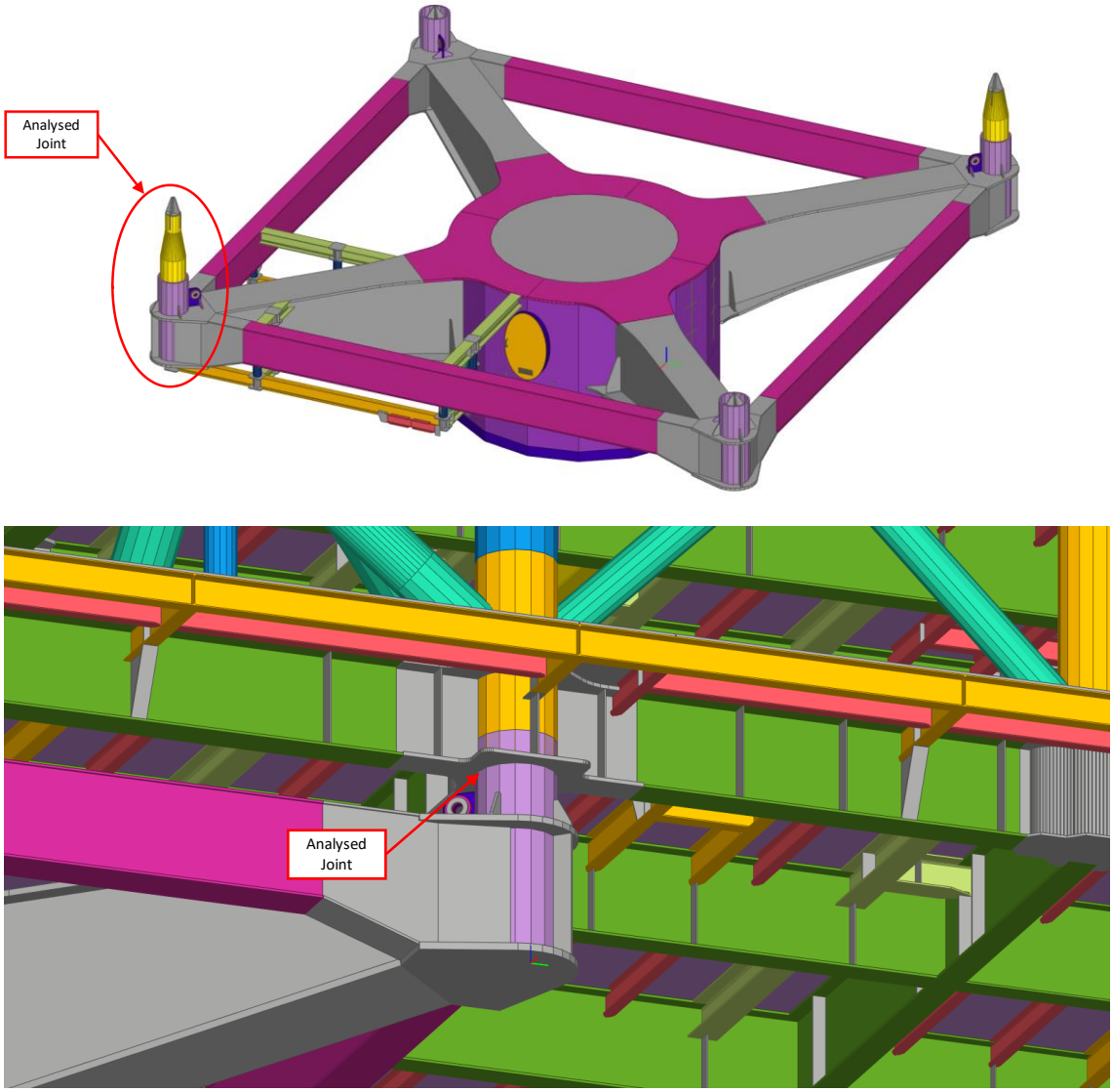


Figure 9.2 – Analyzed critical joints

9.1 METHODOLOGY

The most critical load conditions from Lifting, Installation and Set-Down phases of the BLPH Platform are applied to the generated local FE model of the joints. These models are used to perform local analyses to define their stress state and path.

The analysed joints are fully modelled with all the elements by which they are composed (plates, rings, stiffeners…). The local analyses of the joints are performed for the most critical load combination, and the stress state of all the elements of the node are obtained.

The Von Mises stresses and strains of the node are computed, and the max. values are compared against the admissible values defined by ISO 19902 (Ref./6/) and EC3 (Ref./8/) codes.

9.1.1 ANSYS FE Model

For Ansys FE joint models, all the elements converging to the node are modelled, as well as all the other elements belonging to the node, such as stiffeners, clamps, top and bottom plates, etc. The boundary conditions of the global model are recreated in the separated local FE model.

The stress state of the finite elements forming the joint mesh can be obtained performing a linear or non-linear analysis using Ansys, adding the boundary conditions and the loads.

The FE model is made of 2D or 3D solids. It is meshed with tetrahedrons and hexahedron elements for solid models and with quadrilateral plates for 2D models. All the elements composing the joint FE model are meshed.

The following mesh elements are used for the developed models:

- SOLID186 (Homogeneous Structural Solid Geometry): is a higher order 3-D 20-node solid element that exhibits quadratic displacement behaviour. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element supports plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

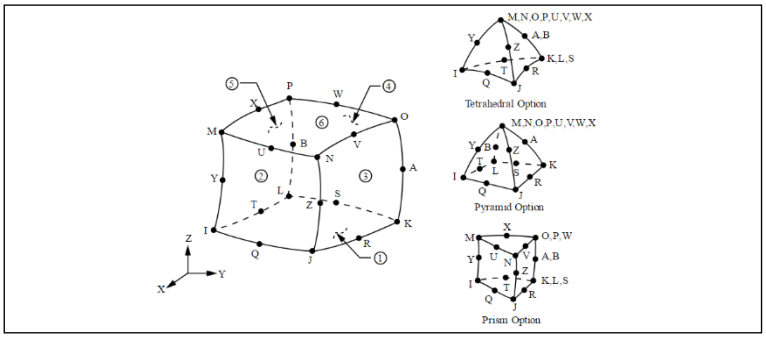


Figure 9.3 – Mesh elements – SOLID186

- SOLID187 (3-D 10-Node Tetrahedral Structural Solid): is a higher order 3-D 10-node element that exhibits quadratic displacement behaviour. It is well suited to modelling irregular meshes (such as those produced from various CAD/CAM systems). The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

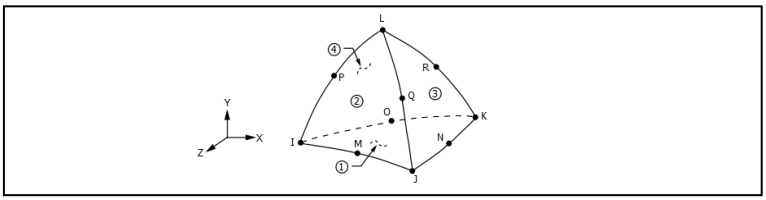


Figure 9.4 – Mesh elements – SOLID187

- SHELL181 (4-node structural Shell): is suitable for analyzing thin to moderately-thick shell structures. It is a four-node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. SHELL181 is well-suited for linear, large rotation, and/or large strain nonlinear applications. The accuracy in modeling composite shells is governed by the first-order shear-deformation theory (usually referred to as Mindlin-Reissner shell theory). The element formulation is based on logarithmic strain and true stress measures.

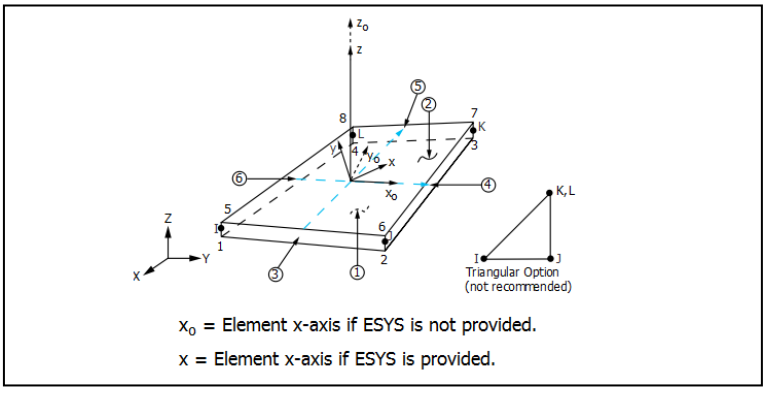


Figure 9.5 – Mesh elements – SHELL181

9.1.2 Boundary Conditions

The same boundary conditions defined in the original SACS model were recreated in the joint local FE model for the members framing into it.

9.1.3 Coordinate System

For joint models developed in Ansys environment, a local coordinate system is assigned. Local axes for the elements converging into the node are also assigned according to the local axes of the original SACS model.

10 LIFTING PADEYE JOINT VERIFICATION

This section presents the local analysis and results for the TP lifting padeye.

According to the structural drawings, lifting padeyes will be welded on the structure after the weighing in order to minimize the out-of-plane actions.

Joint has been modelled and analysed for the Lifting condition with the load taken from Report /25/.

The following sections are reported:

- Description of the FE model of TP Padeye Joint
- Structural analysis results and checks for lifting conditions

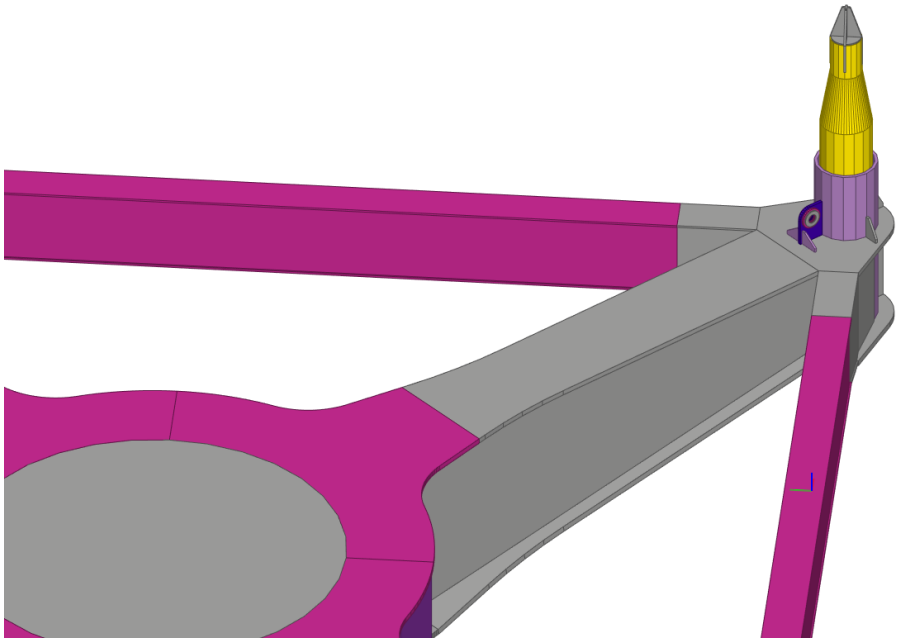


Figure 10.1 – TP Padeye joint – 3D View

10.1 FE MODEL

The TP Padeye Joint is analysed for the Lifting condition. It is a critical joint since the TP structure is lifted at the 4 corner joints, where four padeyes are installed. The padeye geometry is the same for the four lifting points. The model is represented in Figure 10.2.

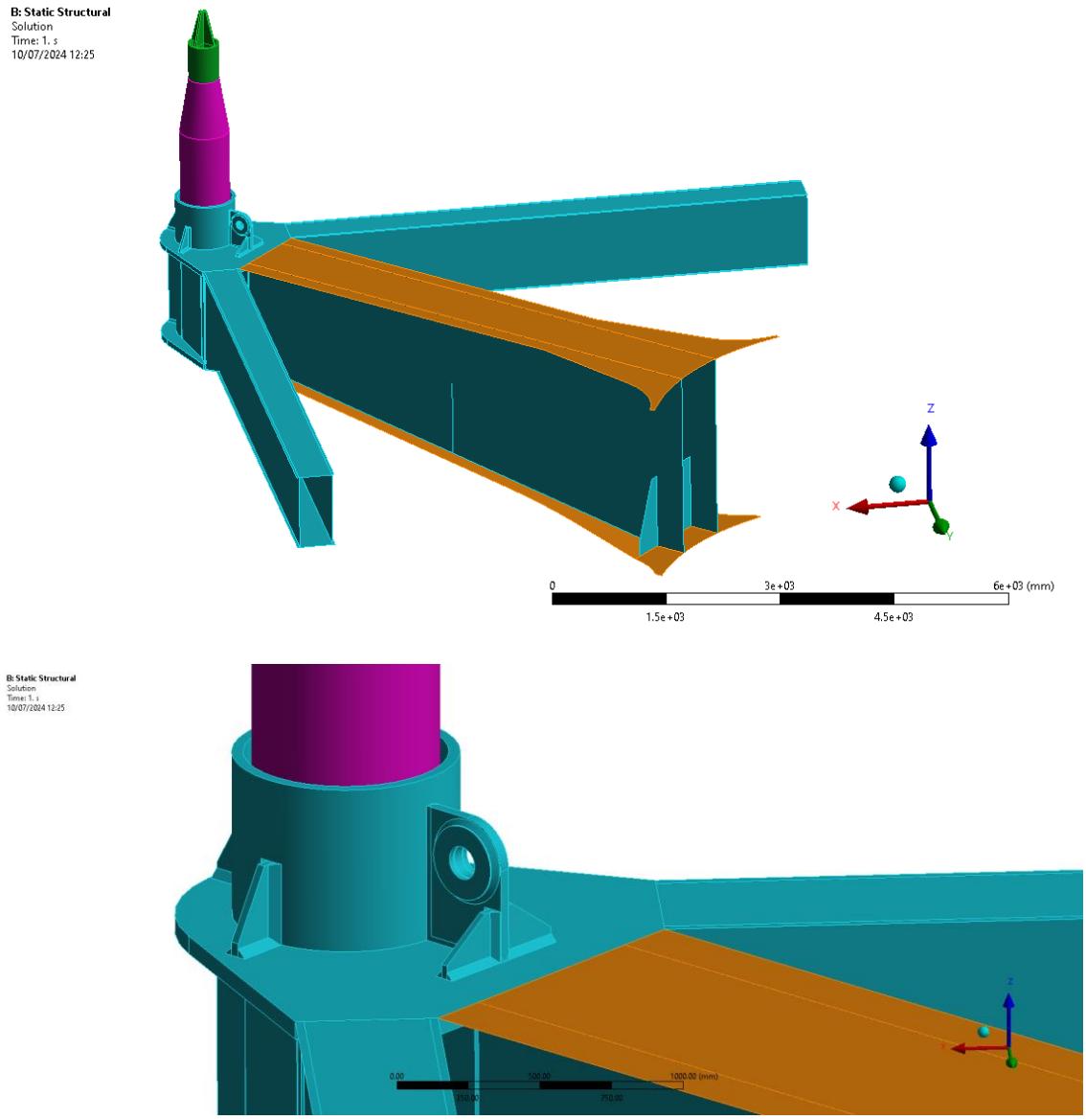


Figure 10.2 – TP Padeye Joint FE Model

The Padeye is a 40mm plate, endowed with two 25mm cheek plates. The padeye is welded to the 65mm thick top flange of the external TP corner joint, and to the 800x40mm vertical stab. Two triangular stiffeners having thickness 25mm are welded to the padeye and the top flange. All the details are in line with the structural drawing 123A90-DDL-ST-DD-90088 (Ref./34/).

The padeye and the parts to which it is welded is made by S420 steel.

The joint is modelled with the correct geometry and thickness of all the elements as per structural drawings Ref./28/ to /35/.

10.1.1 Reference System

The local reference system of the Padeye FE model is represented in Figure 10.3.

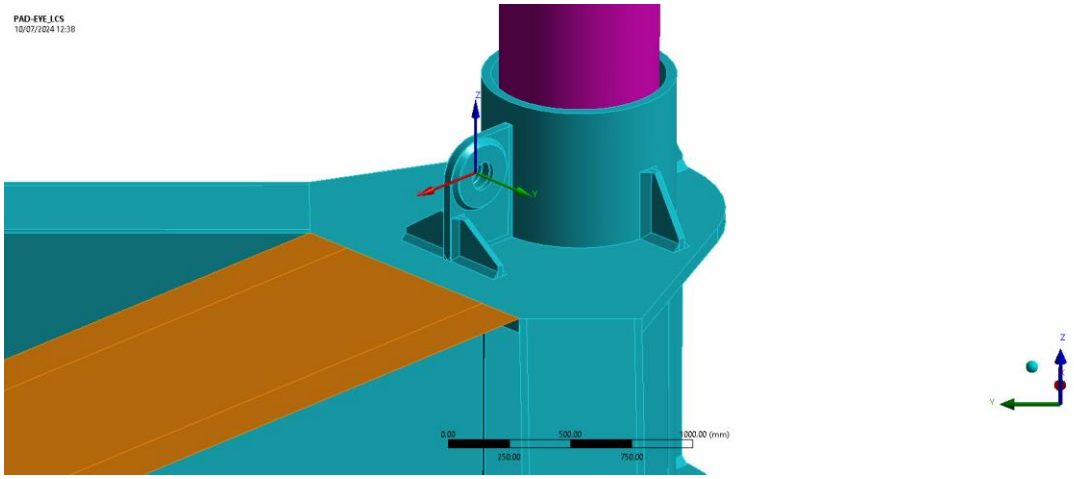


Figure 10.3 – TP Padeye Joint FE Model – Reference system

Other local reference systems defined on the converging beams are presented in the following.

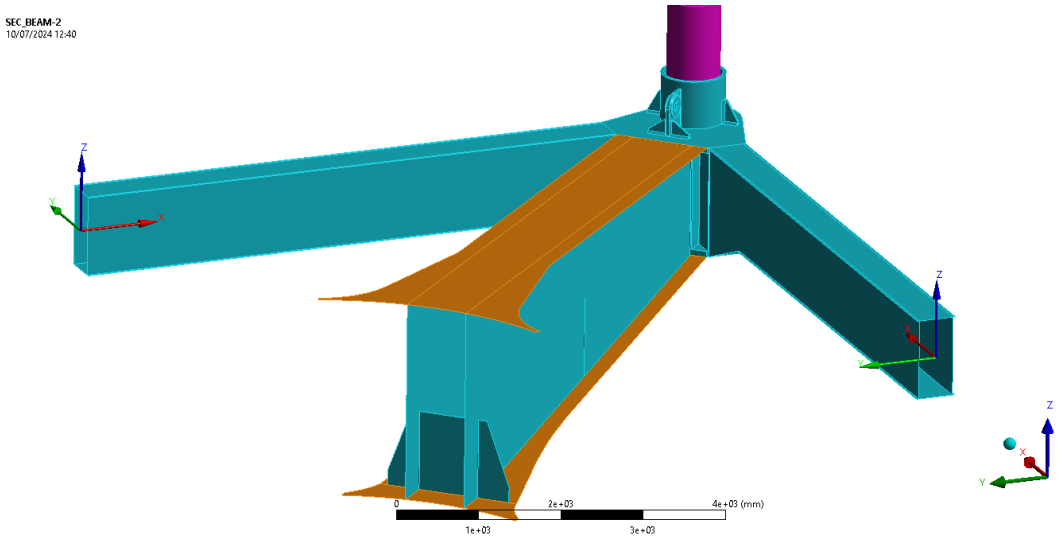


Figure 10.4 – TP Padeye Joint FE Model – converging beams Reference systems

10.1.2 Material

The material used for all the elements of the node is steel S420. The assigned mechanical properties are:

- Young's Modulus 200000 MPa
- Shear Modulus 76923 MPa
- Poisson's Ratio 0.30
- Unit Weight of Steel 7850 kg/m^3
- Coefficient of Thermal Expansion 12·10^{-6} / °C

The assumed constitutive law and non-linear properties are described in sec.7.2 according to DNV-RP-C208 (Ref./13/).

A Multilinear Kinematic Hardening constitutive law having these features is assigned to the FE model in Ansys Environment.

Assigned materials to the model are shown in Figure 10.5.

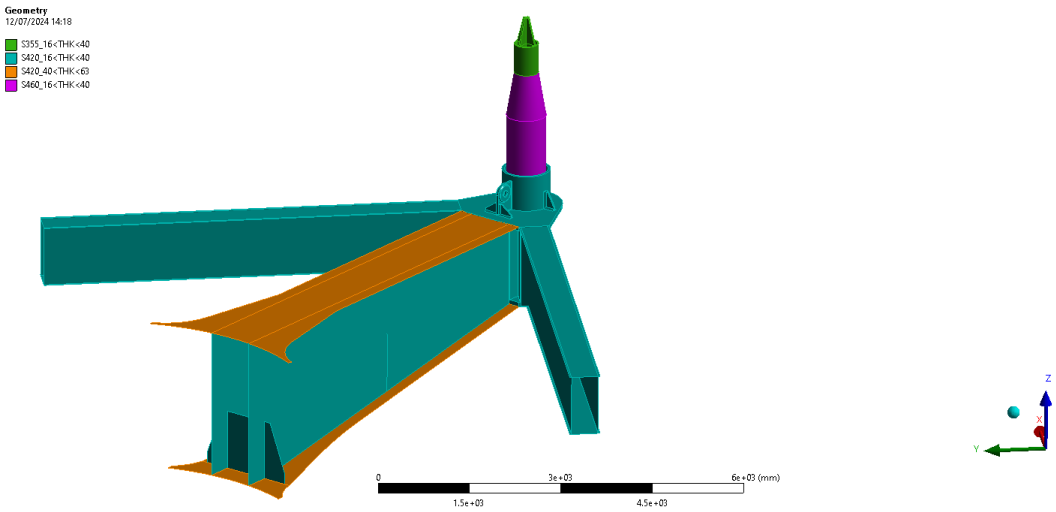


Figure 10.5 –FE Model – assigned materials

10.1.3 Model Mesh

The FE model is meshed using 2D quadrilateral and 3D tetrahedra elements, as reported in 9.1.1.

The part of the joint and the padeye is modelled with 3D brick elements, whereas the converging beams are modelled with 2D elements. The padeye part is the one of interest for lifting checks.

Mesh is refined in the joint, expecially on the padeye and the weldings. Mesh statistics are reported in Table 10.1.

Mesh Part	Number
Nodes	1743364
Elements	1113246

Table 10.1 – TP Padeye Joint FE model – mesh statistics

In the following figures, FE model mesh is presented.

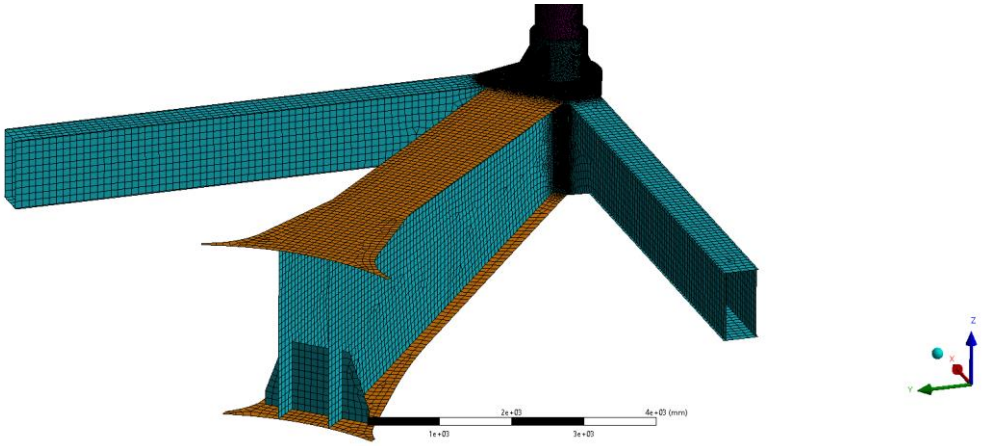


Figure 10.6 – TP Padeye Joint FE Model mesh – 3D isometric view

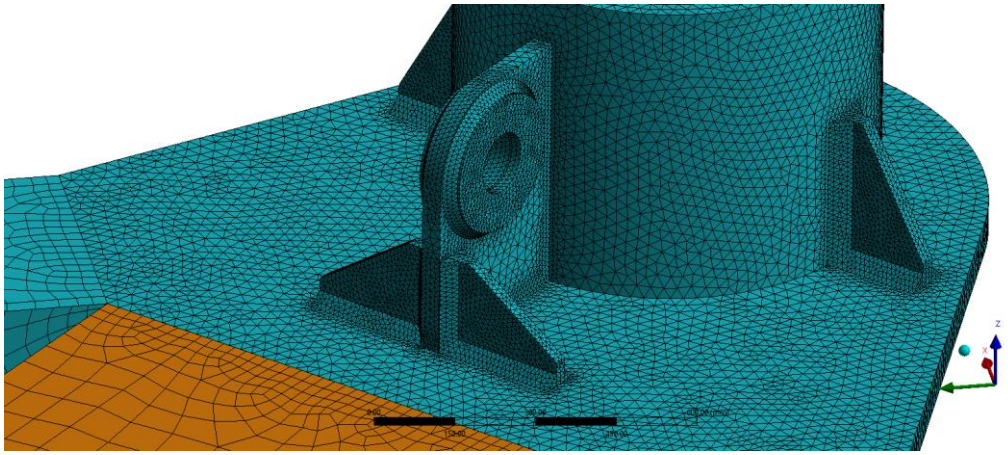


Figure 10.7 – TP Padeye Joint FE Model mesh – joint detail view

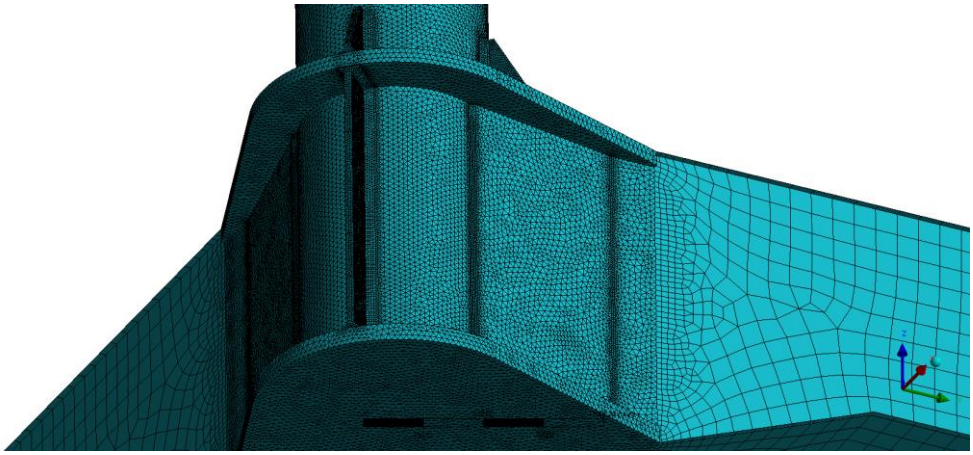


Figure 10.8 – TP Padeye Joint FE Model mesh – trunnion detail view

10.1.4 Constraints

External constraints are applied on the edges of the modelled beams that converge into the node in order to restore the actual boundary conditions of the node. Applied constraints are displayed in Figure 10.9, according to the local reference system as shown.

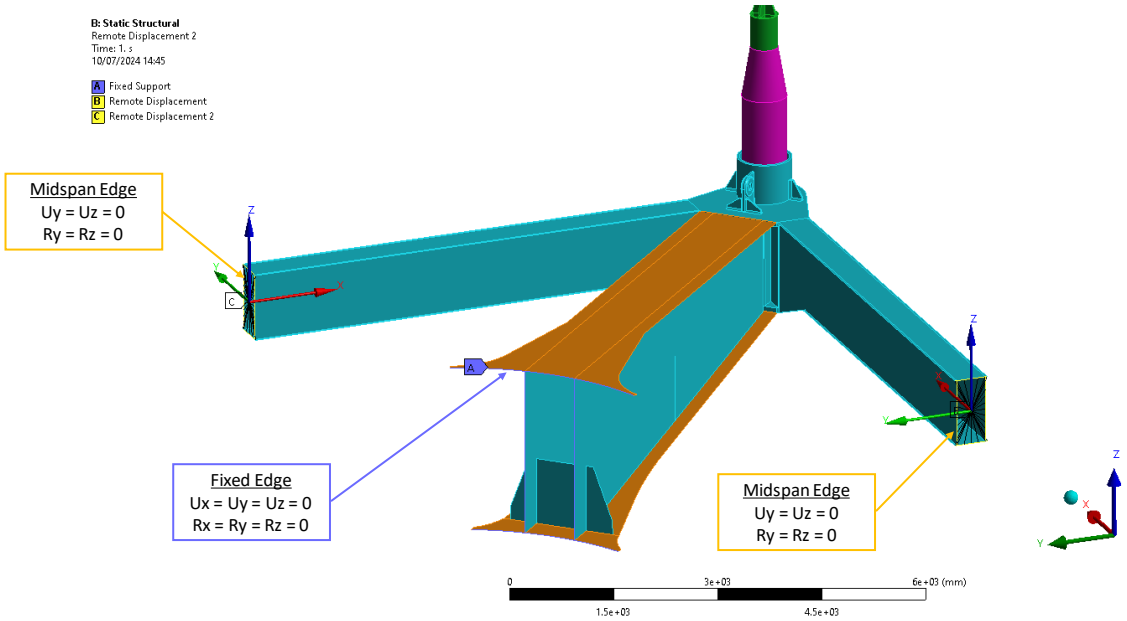


Figure 10.9 – TP Padeye Joint FE Model – external constraints

10.1.5 Contacts

The following contacts have been introduced into the model:

- **Bonded:** bonded contact is used to connect all the welded surfaces together. It is used for the welds' sides to be bonded to the corresponding surface, when bodies are not connected through the mesh joints.
- **Frictional:** frictional contacts are used to connect the surfaces which are in contact to each other but not connected, assuming friction coefficient 0.20. An example is the cheek plates internal surface which is in contact with the padeye main lug but not directly bonded to it (connection is modelled through bonded contacts at the welding parts).
- **Joints:** joint contact is used to model the connection between the 2D surfaces of the edges of the 3D joint and the 2D end edges of the beams, in order to have the perfect continuity between the two bodies and to transfer all the internal actions, assuming the connection section as a rigid section (De Saint Venant).

Examples of the aforementioned contacts are shown in the following figures.

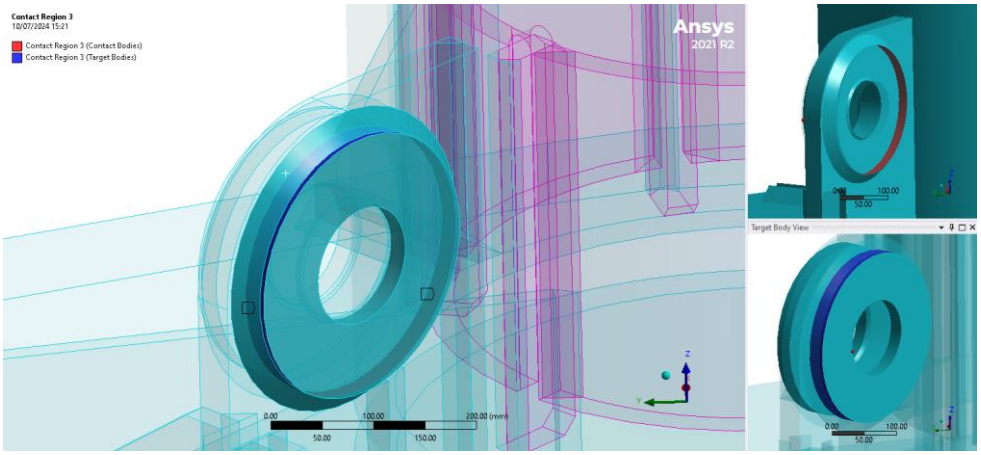


Figure 10.10 – TP Padeye Joint FE Model – bonded contact between cheek plate and weld

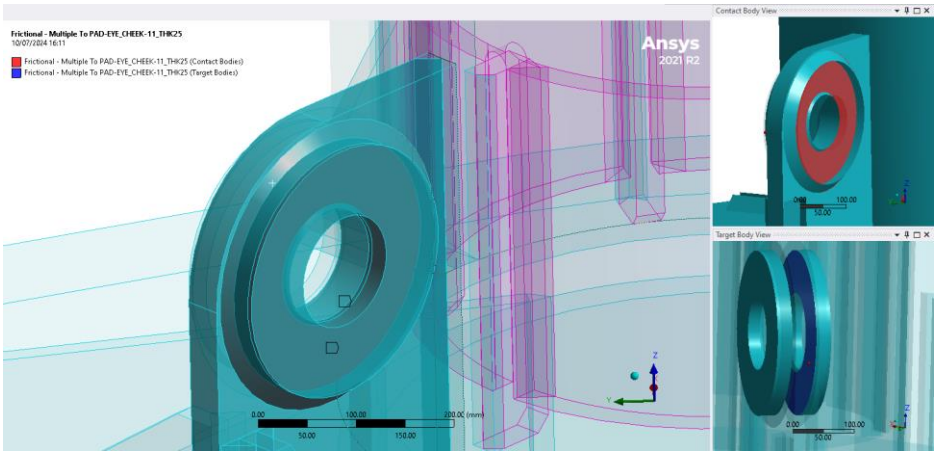


Figure 10.11 – TP Padeye Joint FE Model – friction contact between cheek plate and padeye main lug

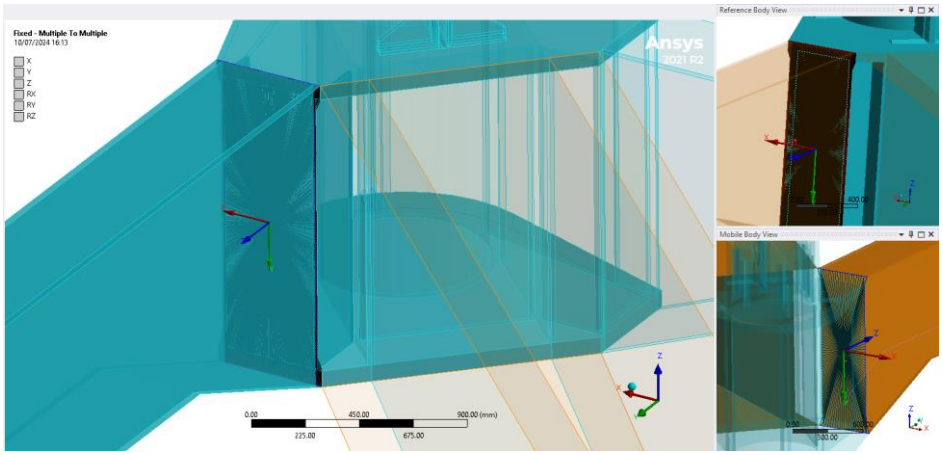


Figure 10.12 – TP Padeye Joint FE Model – joint contact between 3D joint and 2D beam

10.1.6 Loads

The critical load condition for this joint is the lifting phase, hence the joint is analysed for this condition only.

In the following table, the reaction loads at the lifting points from the lifting analysis are reported (Ref./25/),

Lift point	critical load comb.	Max force (kN) (CF and ULS included)
LIF1	LC21 (cog shift towards C2)	2045.52
LIF2	LA21 (cog shift towards A2)	2433.15
LIF3	LA41 (cog shift towards A4)	2237.43
LIF4	LC41 (cog shift towards C4)	2059.66

Table 10.2 – Summary Table - Slings internal actions: CF = 1.30 and ULS factor = 1.30 included. To be used for padeye design

From the lifting analysis report (Ref./25/), the maximum ULS force acting on SACS joint LA21: F = 2433.15 kN. This load takes into account all combination coefficients for lifting condition, including the consequence factor CF = 1.30.

The total force assumed for the padeye check is then F_{Ed} = 2500 kN inclined at 60deg. As per DNV-ST-N001, an out-of-plane force is added to the model, which is 3% of the total lifing force, therefore F_{op} = 75 kN.

A Resume of the applied forces is reported in Table 10.3, and they are assigned according to the padeye local reference system as specified in sec.10.1.1.

Element	Fx [kN]	Fy [kN]	Fz [kN]
Padeye	1250	-75	2165

Table 10.3 – TP Padeye Joint FE Model – applied loads

Applied loads to the FE model are shown in Figure 10.13.

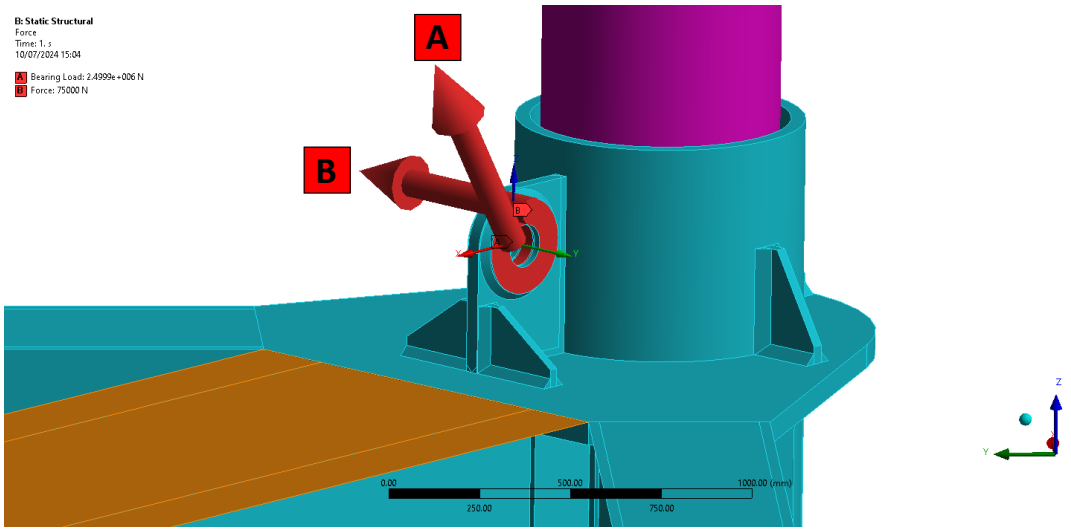


Figure 10.13 – TP Padeye Joint FE Model – applied loads

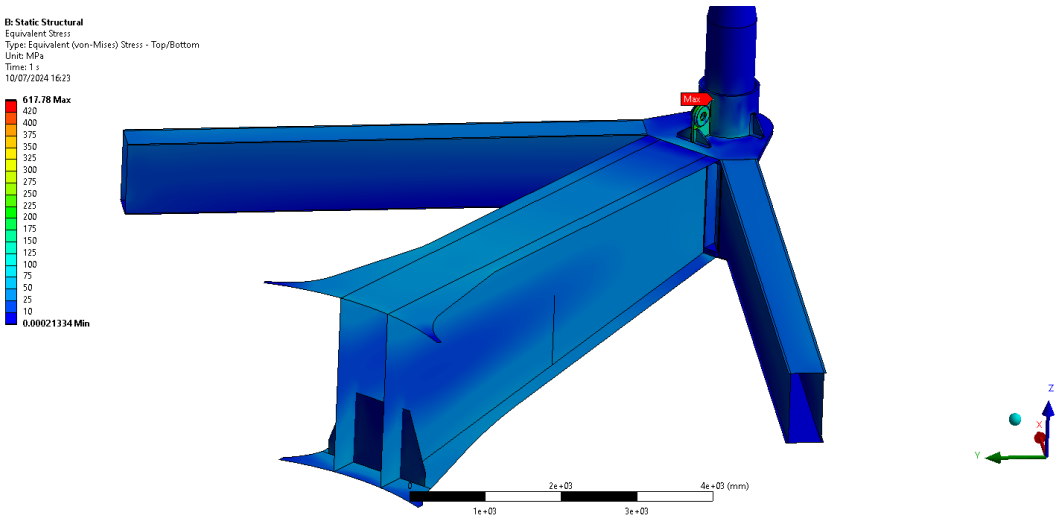
10.2 FEM ANALYSIS AND RESULTS

TP Padeye joint is checked for Lifting conditions, the most critical ones for this joint. Loads are taken from the Lifting analysis report Ref./25/, considering all coefficients, including consequence factor.

The joint is checked against ISO 19902 Ref./6/ and UNI EN 1993, Refs./8/, /9/. Checks are performed according to what stated in sec.8.3, at ULS.

A non-linear analysis is carried out, and the results are plotted in the following, in terms of Von Mises stresses and strains. Checks are performed on the obtained results as follows.

In the following figures, the Von Mises stresses for the padeye are plotted.



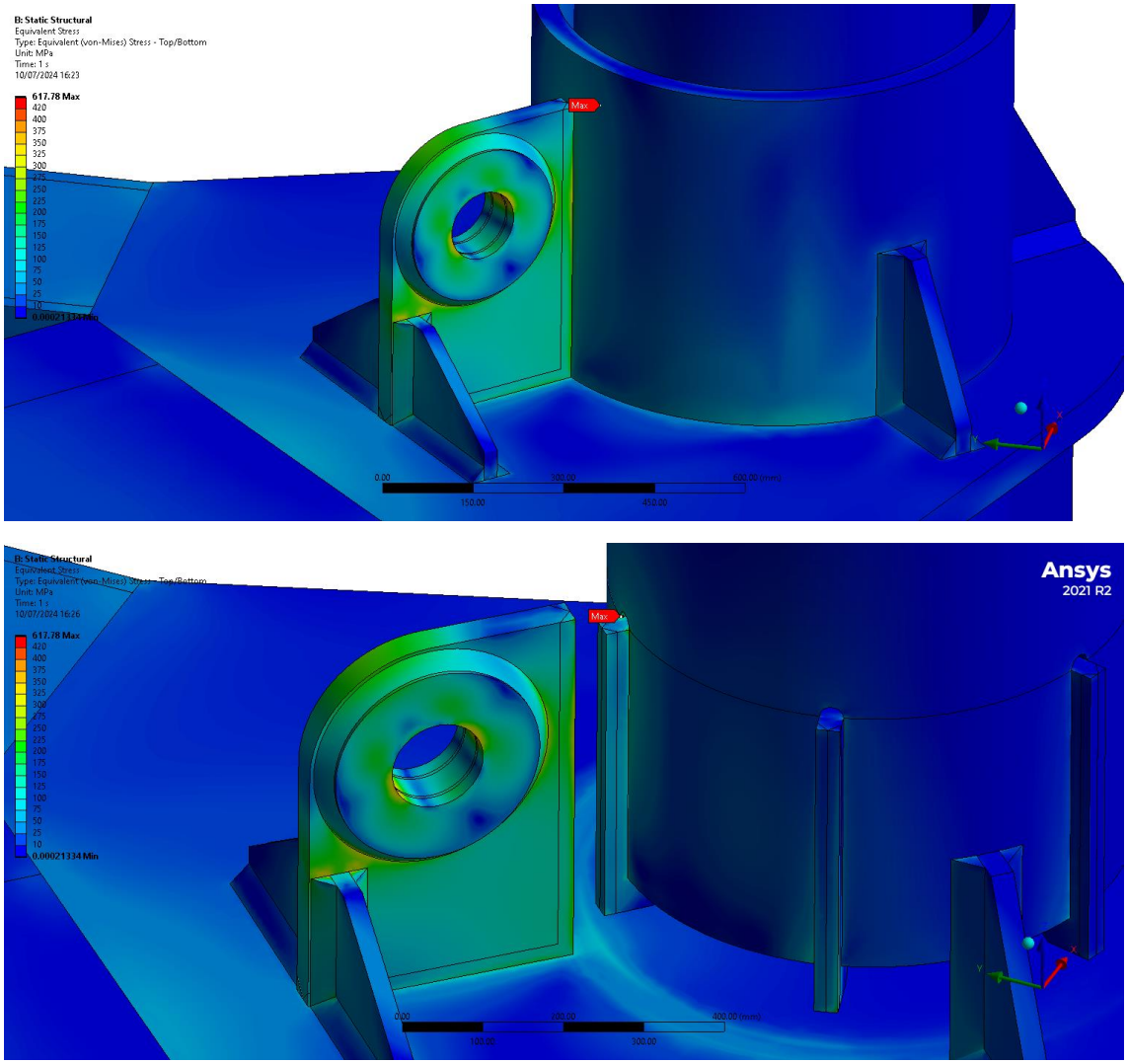


Figure 10.14 – TP Padeye Joint FE model – VM Stresses

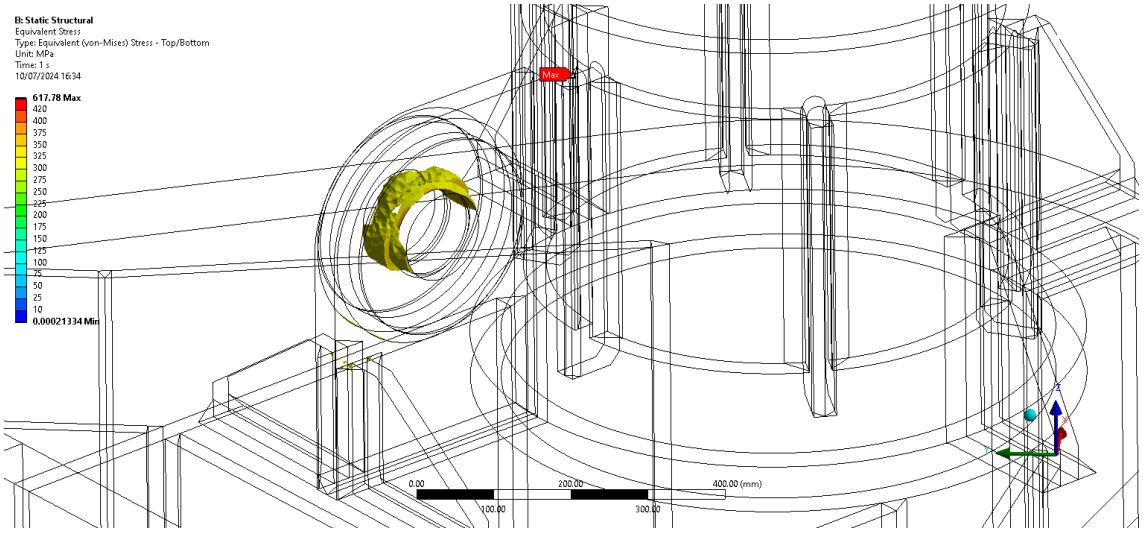


Figure 10.15 – TP Padeye Joint FE model – VM Stresses > 320 MPa

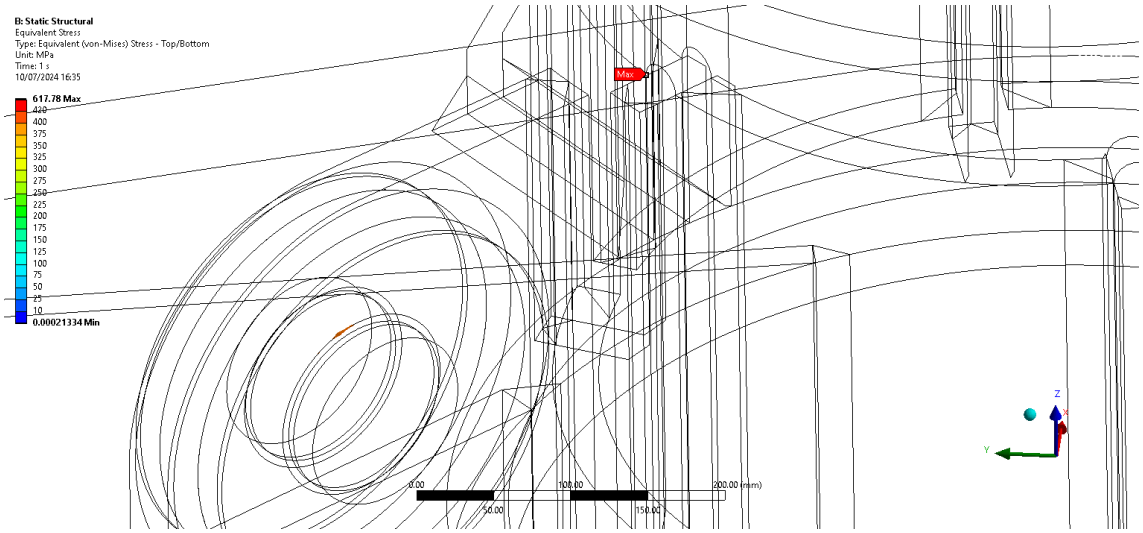


Figure 10.16 – TP Padeye Joint FE model – VM Stresses > 400 MPa

For structural checks, it is taken the most restrictive value of admissible stress, both for tubular and non-tubular elements, on the safe side: for t>16 mm, F_y = 400 MPa, therefore:

- **Members:** $\sigma_{adm} = \Phi * F_y = 0.85*400 = 340$ MPa
- Welds: $\sigma_{adm} = F_y/\gamma_{m2} = 400/1.25 = 320$ MPa

It can be observed that the VM stresses are largely lower than the admissible stress σ_{adm} = 320 MPa for global checks, and they are exceeded in local portions that are the load application surface at the main lug and the cheek plates, and the intersection corner between the 30mm plate coplanar with the main lug and the

stabbing guide. This portion is confined in an area smaller than 2tx2t, not interesting the whole thickness of the plate.

The F_y value is exceeded in a very limited area with a peak max. value of σ_{VM} = 618 MPa, and the σ_{adm} is exceeded in a significant area in the padeye, so plastic strains in these portions are investigated as well.

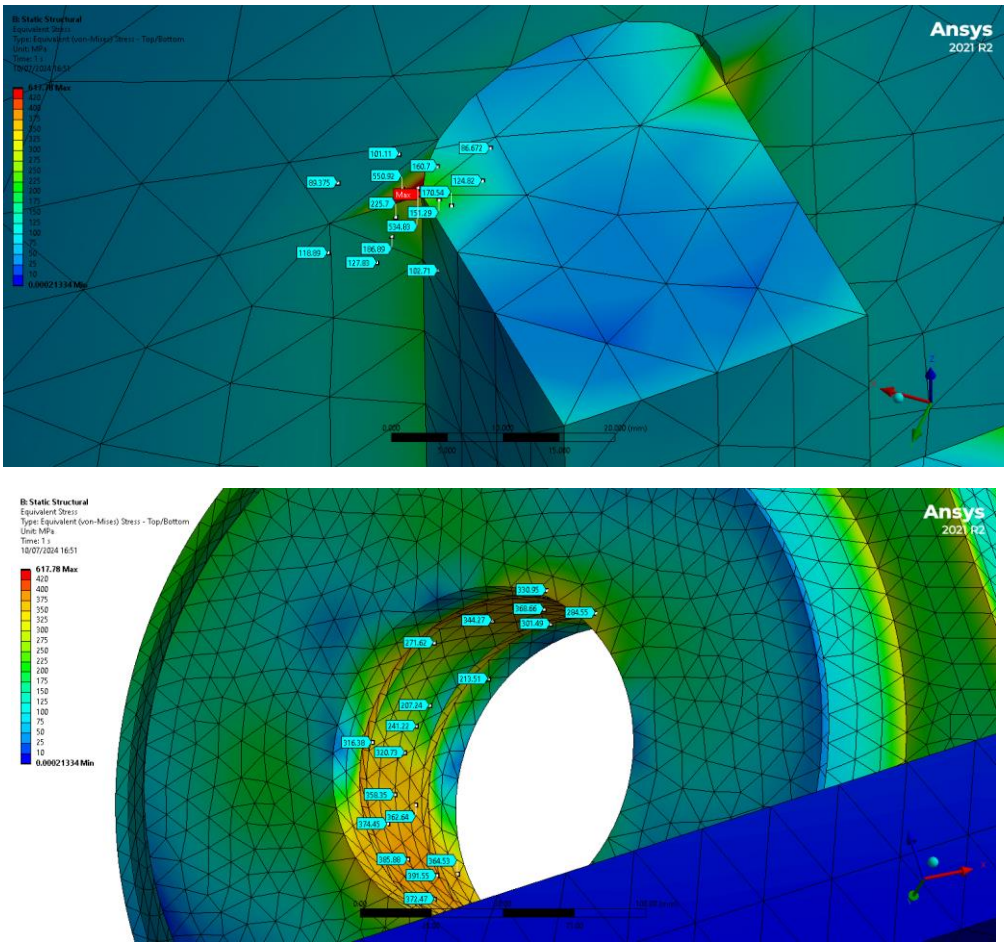


Figure 10.17 – TP Padeye Joint FE model – Peak VM Stresses at hot spot areas

For the hot spot areas, elasto-plastic strains are measured according to the performed non-linear analysis. They shall be less than 5% mm/mm, as per design criteria defined in 8.3.

In the following figures, plastic strains (Figure 10.18) and total strains (Figure 10.19) are plotted.

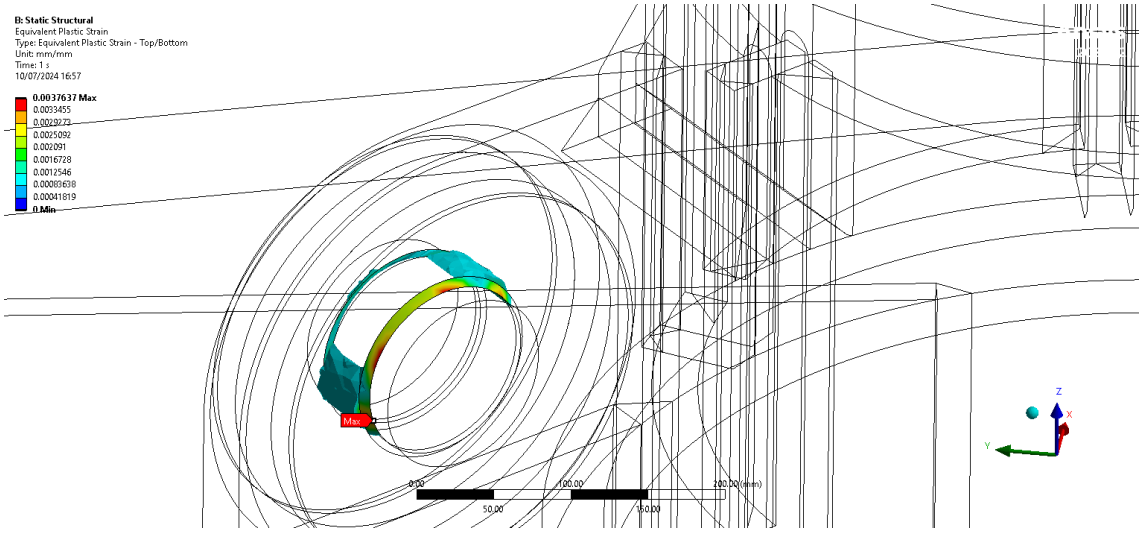
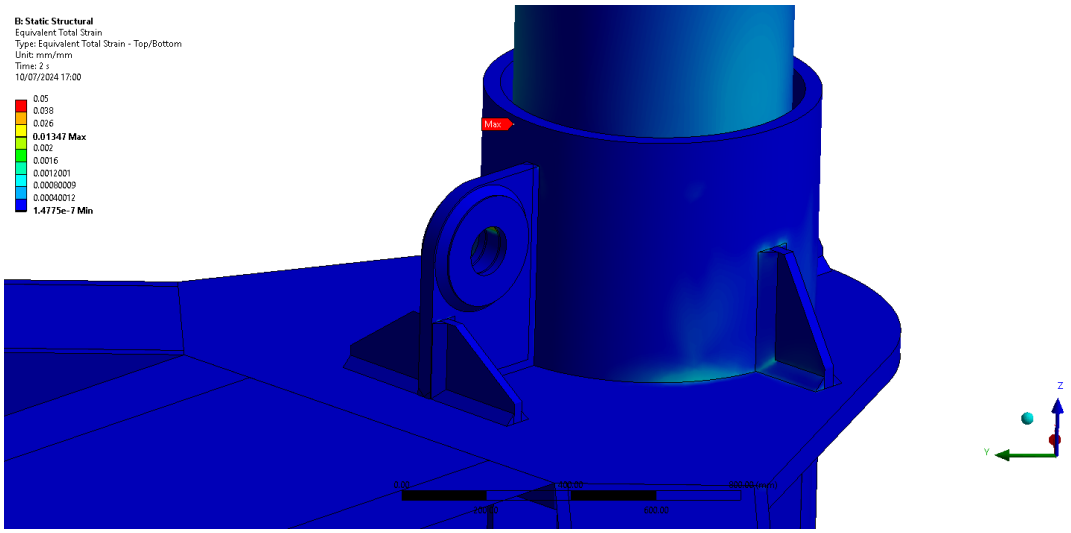


Figure 10.18 – TP Padeye Joint FE model – Plastic strains > 0.1%



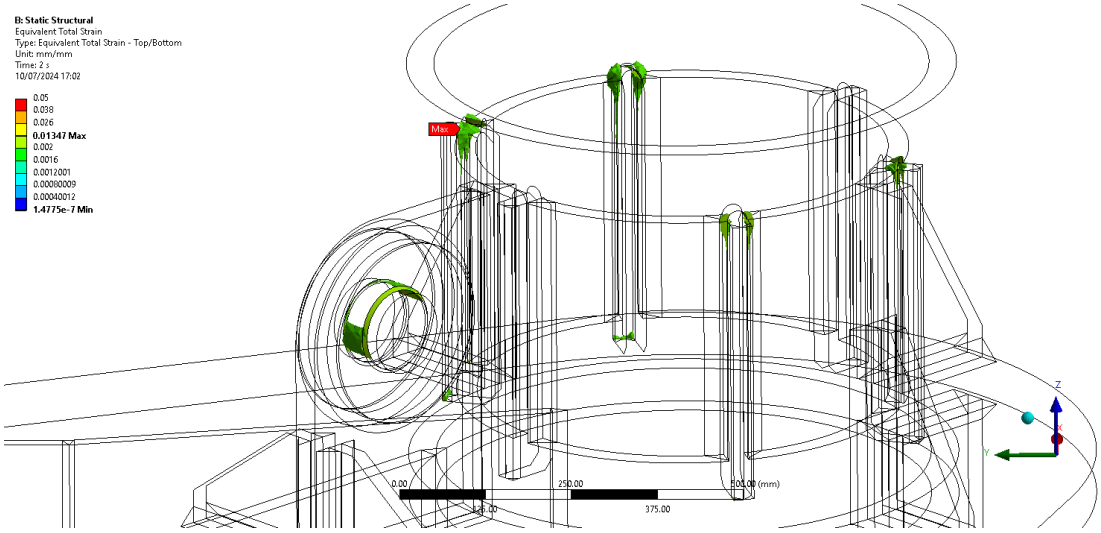


Figure 10.19 – TP Padeye Joint FE model – Total strains > 0.2%

It is observed that plastic strains occur in a limited portion at the padeye internal surfaces where the load is applied, and in some upper corner of the connecting stiffeners of the stabbing guide to the Φ800x40 stab to which padeye is welded.

The max. value of total strains is ε_t = 0.013 mm/mm < ε_{adm} = 0.05 mm/mm. Strains are within the admissible value.

Checks are fulfilled.

The TP Padeye has enough resistance capacity to withstand the loads for lifting conditions.

Being the four padeyes geometrically identical and having verified the one subjected to the max. sling load, all padeyes pass the structural checks.

11 TP-STABBING GUIDE JOINT VERIFICATION

This section presents the local analysis and results for the primary stabbing guide of the Transition Piece (TP).

The stabbing guide is used as installation aid during the installation phase, when the topside is laid down on the TP.

The primary stabbing guide is the first to be hit during installation, and it is the tallest one, that is why it is chosen for checks. The secondary stabbing guide takes half of the impact load and it is shorter, having the same geometry in the lower part, so its checks are covered from the primary guide checks.

This joint has been analysed for installation conditions. In the following sections are reported:

- Description of the FE model
- Structural analysis results and checks for the two conditions

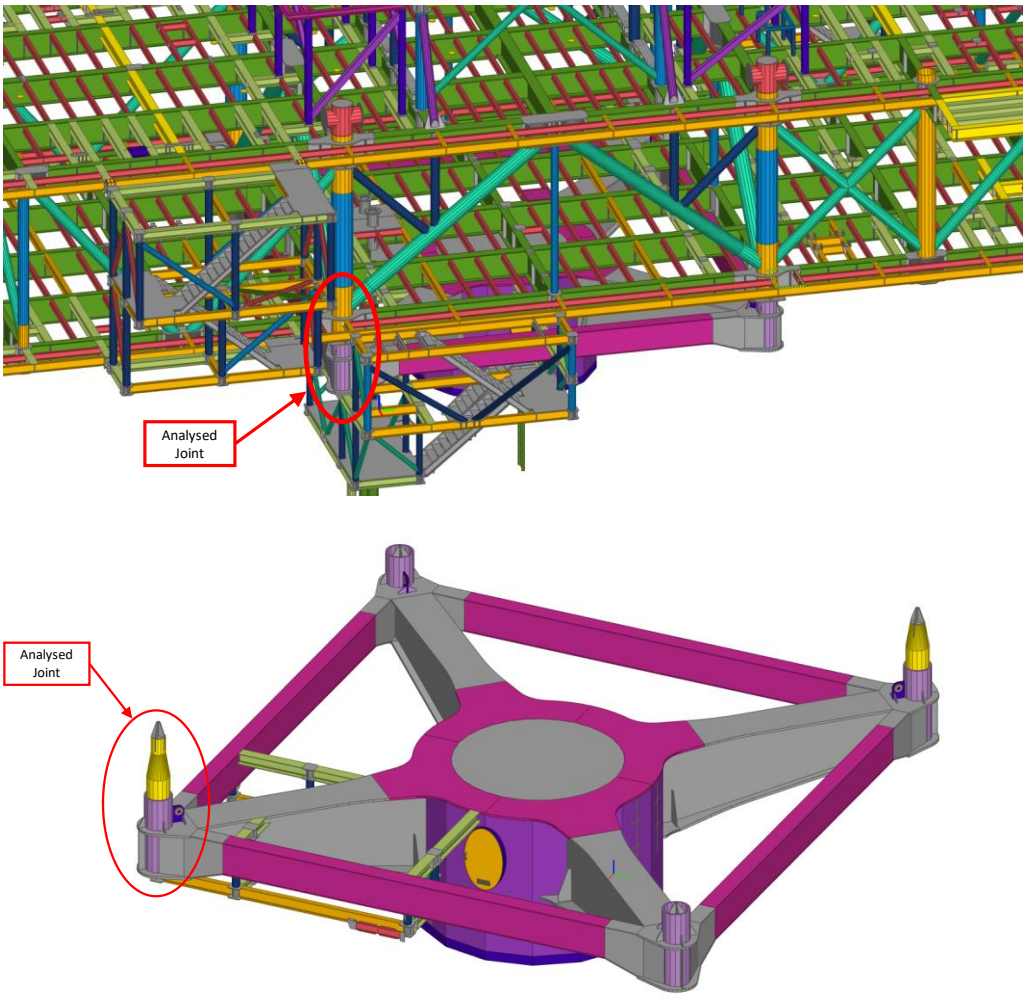


Figure 11.1 – Stabbing-guide Joint – 3D view

11.1 FE MODEL

It is used the same model described in Sec.10.1. Specific details regarding the stabbing guide are highlighted in the following.

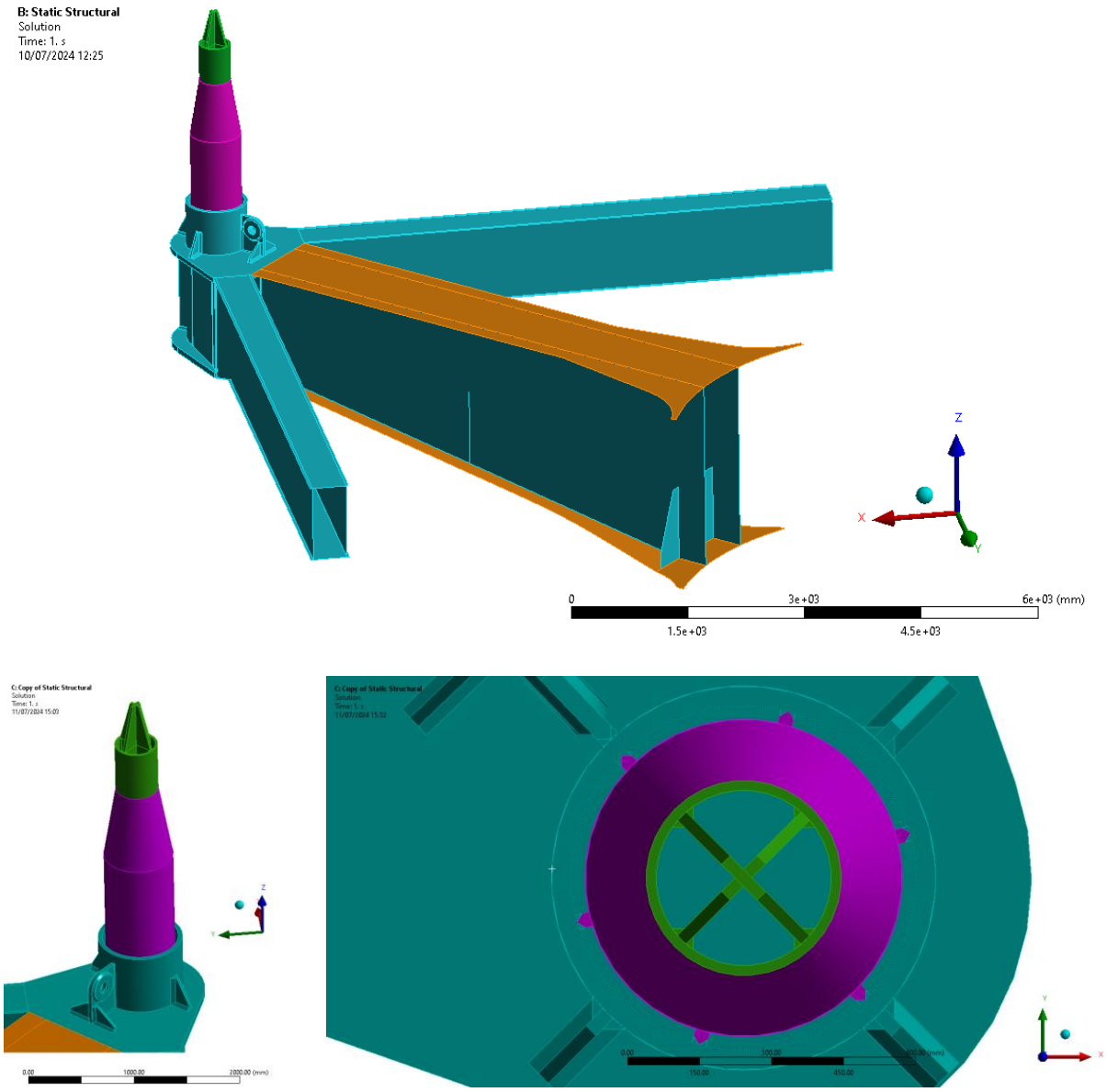


Figure 11.2 – Stabbing-guide FE Model

The stabbing guide is an element made of a Φ660x30 mm base tube, connected to the 600mm long Φ800x40 stab of the top corner flange (65mm) by 6 plates 30mm thick with full pen weld. The base tube reduces through a 20mm transition cone to a tube Φ406.4x20 mm. At the top of this tube, a spike is made

by four crossing plates having thickness 25mm. these plates are welded to the Φ406.4x20 tube with double fillet weld. The joint is made by S420 steel for the base tube and the cone, while the upper Φ406.4x20 mm tube and the plates are in S355 steel. The joint is modelled with the correct geometry and thickness of all the elements as per structural drawings Ref./35/.

11.1.1 Reference System

The local reference system on the spike of the stabbing guide is presented in Figure 11.3.

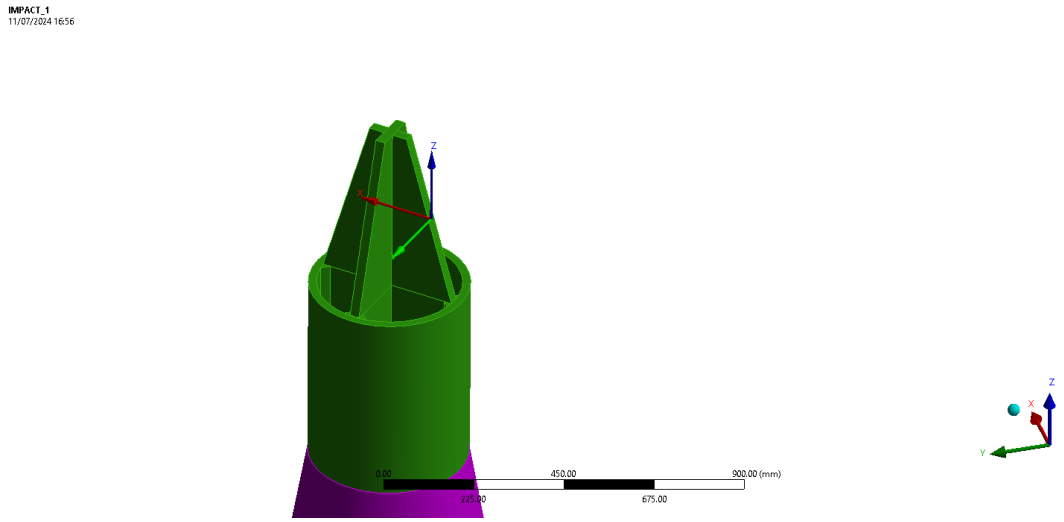


Figure 11.3 – Stabbing guide FE model – local impact Reference system

Other reference systems used in the model have been introduced in sec.10.1.1.

11.1.2 Material

The material used for all the elements of the node is S420, except for the spike of the stabbing guide, where S355 is used for the Φ406.4x20 mm tube and the 25mm plates. The assigned mechanical properties are:

- Young's Modulus 200000 MPa
- Shear Modulus 76923 MPa
- Poisson's Ratio 0.30
- Unit Weight of Steel 7850 kg/m^3
- Coefficient of Thermal Expansion $12 \cdot 10^{-6}$ / °C

The assumed constitutive law and non-linear properties are described in sec.7.2 according to DNV-RP-C208 (Ref./13/).

A Multilinear Kinematic Hardening constitutive law having these features is assigned to the FE model in Ansys Environment. Assigned materials to the model are shown in Figure 10.5.

11.1.3 Model Mesh

The mesh of the model is described in sec. 10.1.3.

In the following figures, details of the FE model mesh for the stubbing guide are shown.

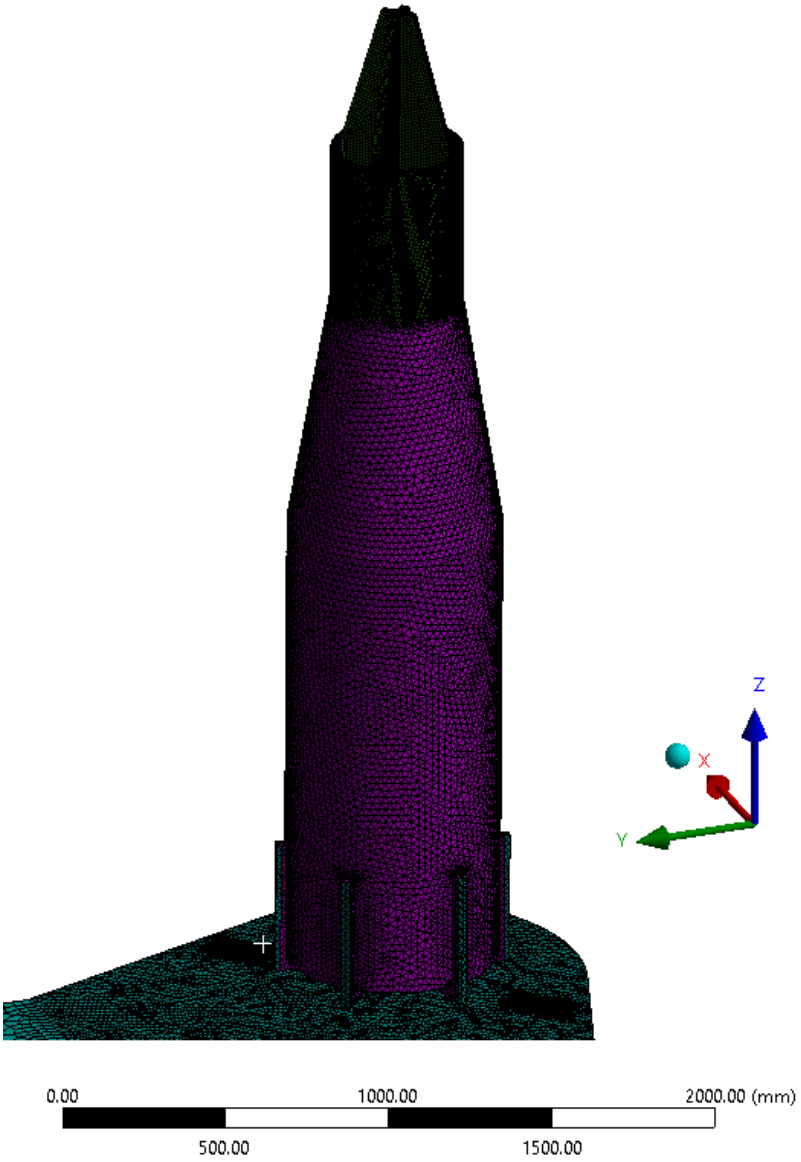


Figure 11.4 – Stabbing guide FE Model mesh – 3D isometric view

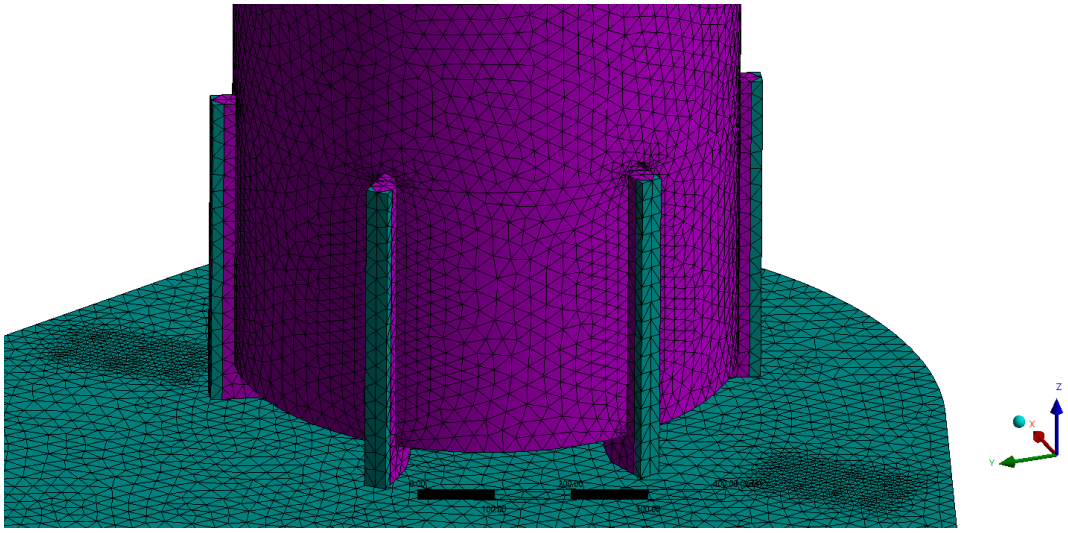


Figure 11.5 – Stabbing guide FE Model mesh – lower connection mesh

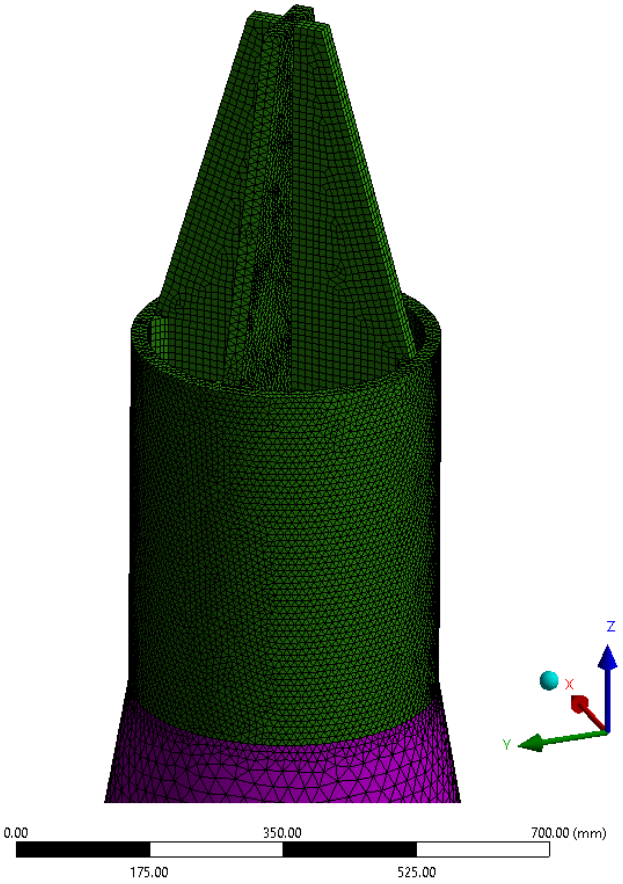


Figure 11.6 – Stabbing guide FE Model mesh – top detail view

11.1.4 Constraints

Constraints are described in Sec.10.1.4.

11.1.5 Contacts

Introduced contacts are described in Sec.10.1.5.

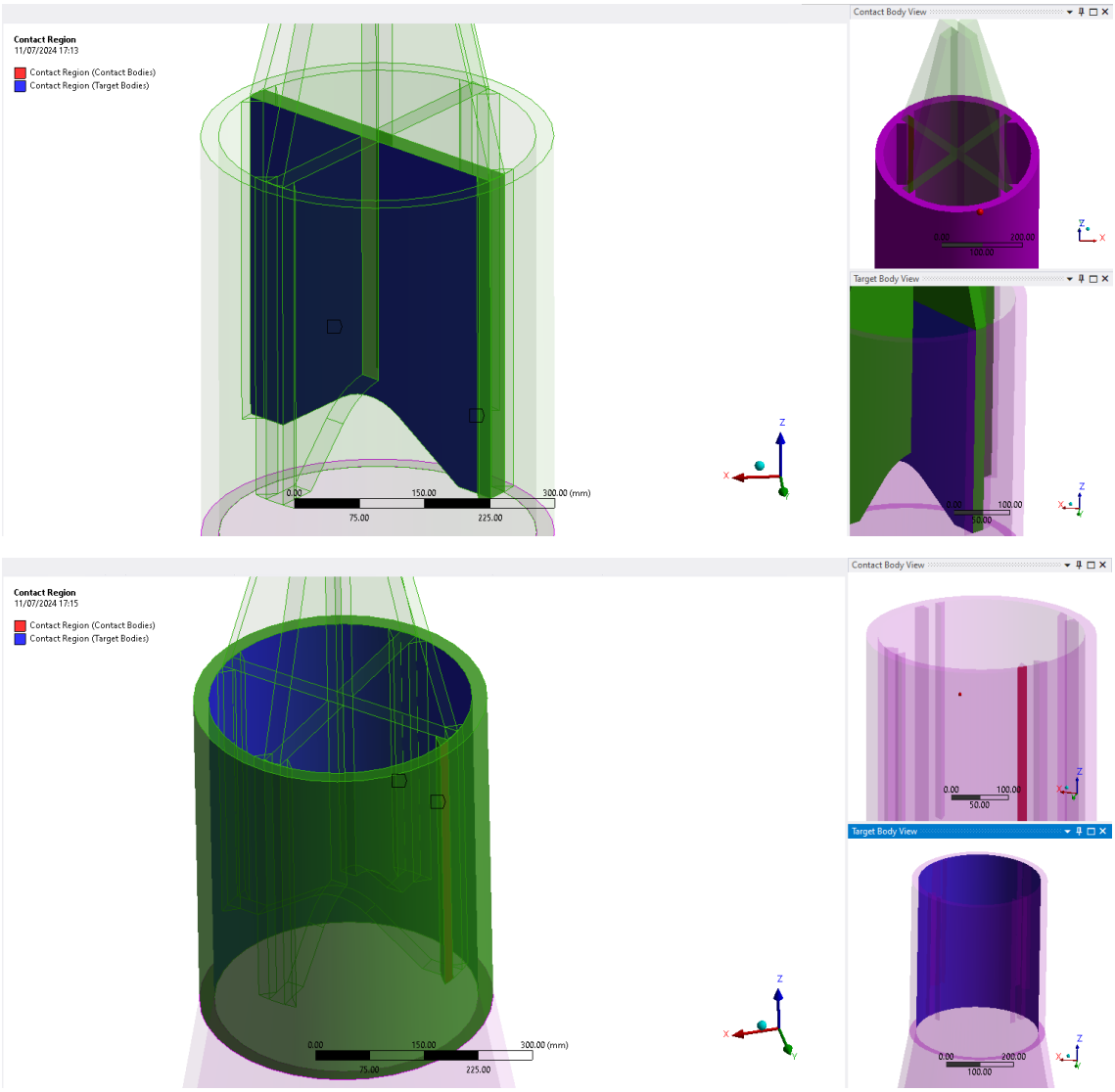


Figure 11.7 – TP Stabbing Guide FE Model – bonded contact between cross plate and tube weld

11.1.6 Loads

According to section 10.6 of Design Basis, Ref. /18/, the impact load is equivalent to

Design loads are (primary guide):

- Fh = 5% of the static hook load (horizontal):
- Fv = 10% of the static hook load (vertical force to be combined with Fh to exacerbate applied loads).

Acting at the top of primary stabbing guide. The hook load is reported in TP lifting report Ref. /25/.

Design load for secondary guide is 50% of the design load for primary one; considering that the structural configuration of primary and secondary guide is the same and the reduced performance required, the verification here performed is considered valid for both the stabbing guides.

Load condition	Vertical	Horizontal
Impact load	0.10*2000 =-200 t	0.05*2000 =-100 ton

Table 11.1 – Stubbing guide FE Model – applied loads

The forces listed in Table 11.1 are applied on the top stiffener of the stabbing guide, which is the first point of impact, distributed on the lateral surface as shown in Figure 11.8. Vertical and horizontal components are assigned according to the local reference system introduced in sec.11.1.1.

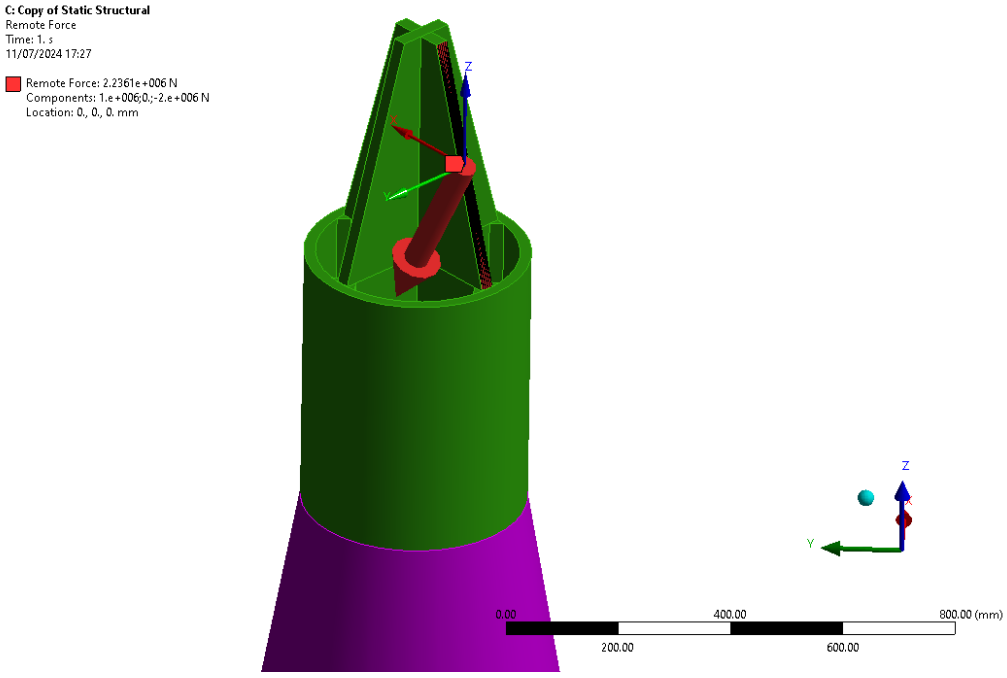


Figure 11.8 – Stubbing guide FE Model – applied loads – impact condition

11.2 FEM ANALYSIS AND RESULTS

TP stabbing guide is checked for impact conditions, as per loads defined in sec. 11.1.6.

The joint is checked against ISO 19902 Ref./6/ and UNI EN 1993, Refs./8/, /9/. Checks are performed according to what stated in sec.8.3, at ULS.

A non-linear analysis is carried out, and the results are plotted in the following, in terms of Von Mises stresses and strains. Checks are performed on the obtained results as follows.

In the following figures, the Von Mises stresses for the stubbing guide are plotted.

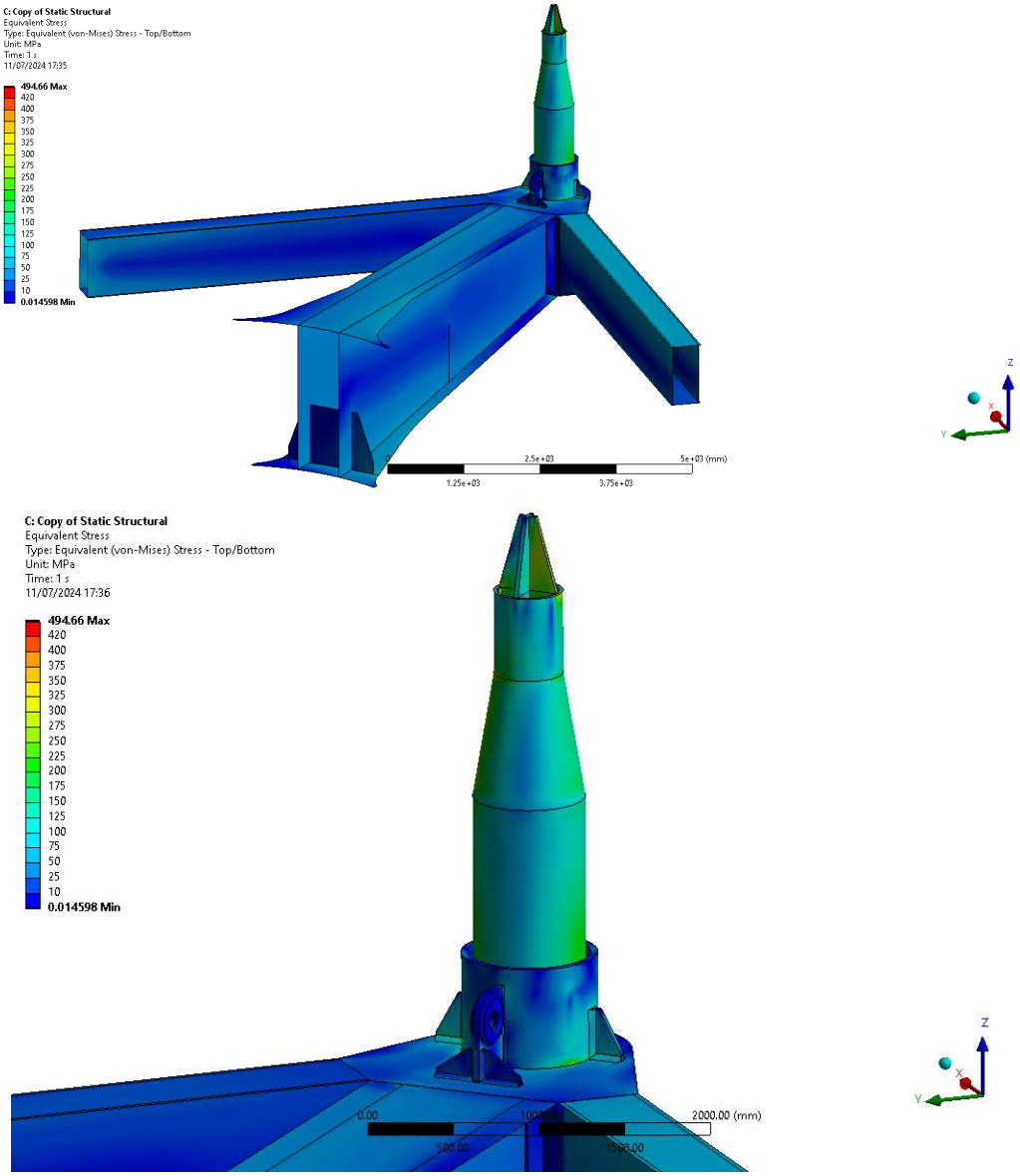


Figure 11.9 – Stubbing guide FE model – VM Stresses (front view)

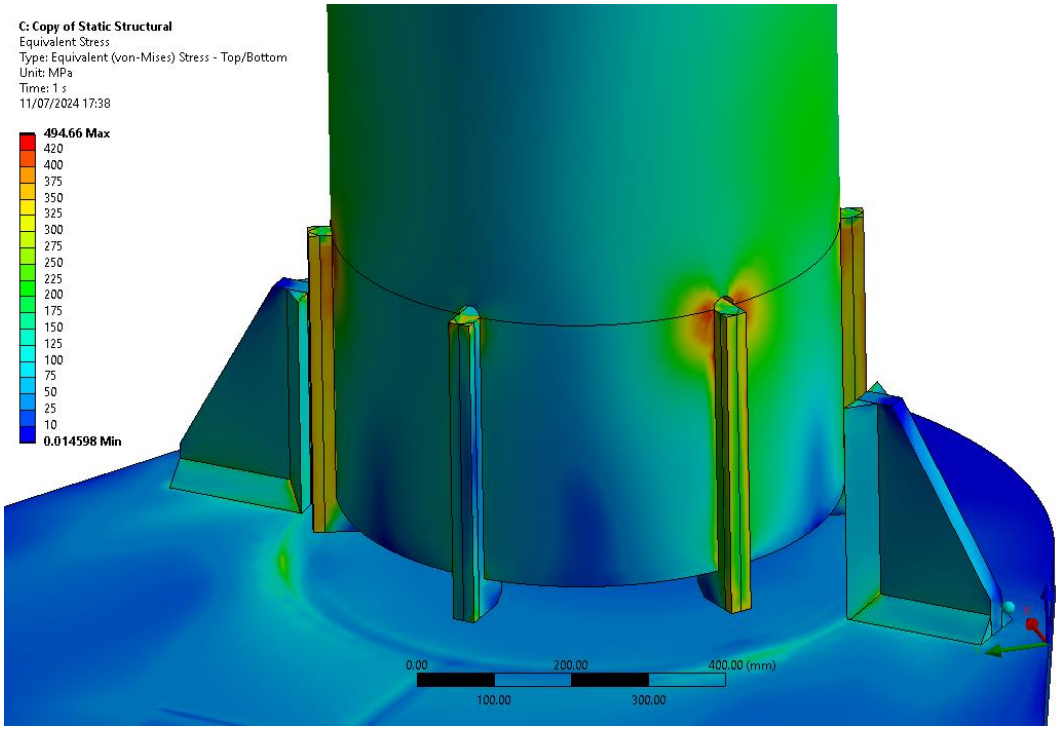


Figure 11.10 – Stubbing guide FE model – VM Stresses (lower connection detail view)

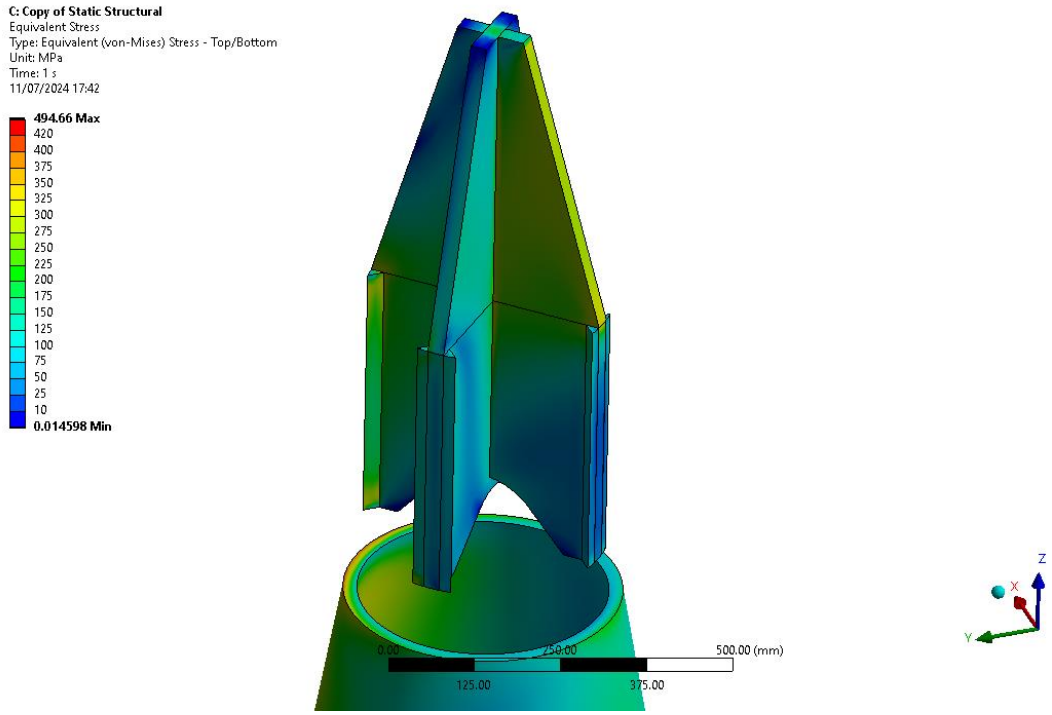


Figure 11.11 – Stubbing guide FE model – VM Stresses (upper detail view)

11.2.1 S460 Steel structural checks

Here are reported FEA results and checks for the S460 steel part (lower part of stabbing guide).

In the following figures, the Von Mises stresses are plotted.

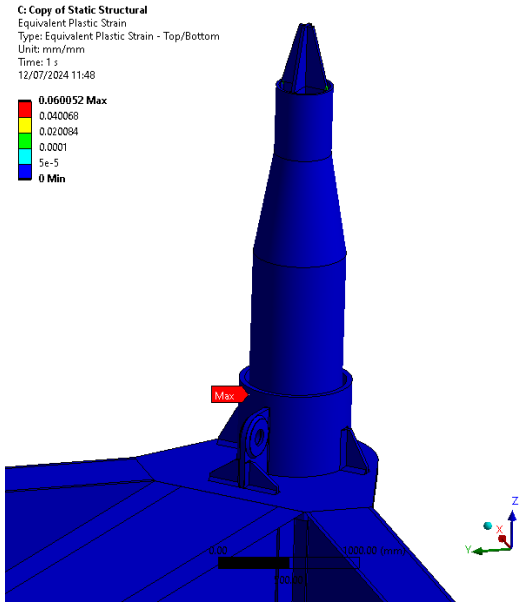


Figure 11.12 – Stabbing guide FE model – S460 VM Stresses (front view)

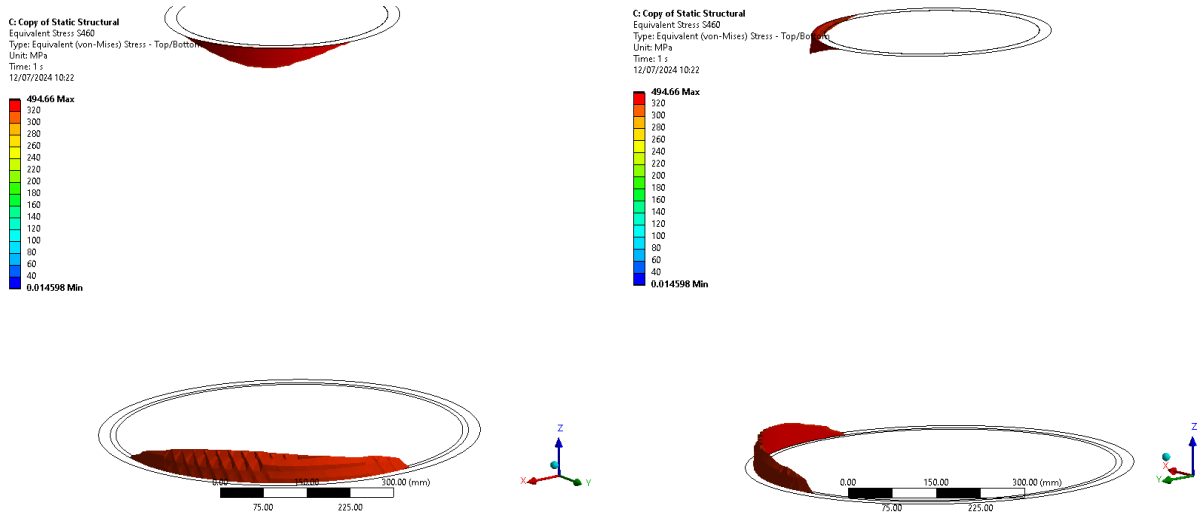


Figure 11.13 – Stabbing guide FE model – S460 VM Stresses > 320 MPa – upper cone

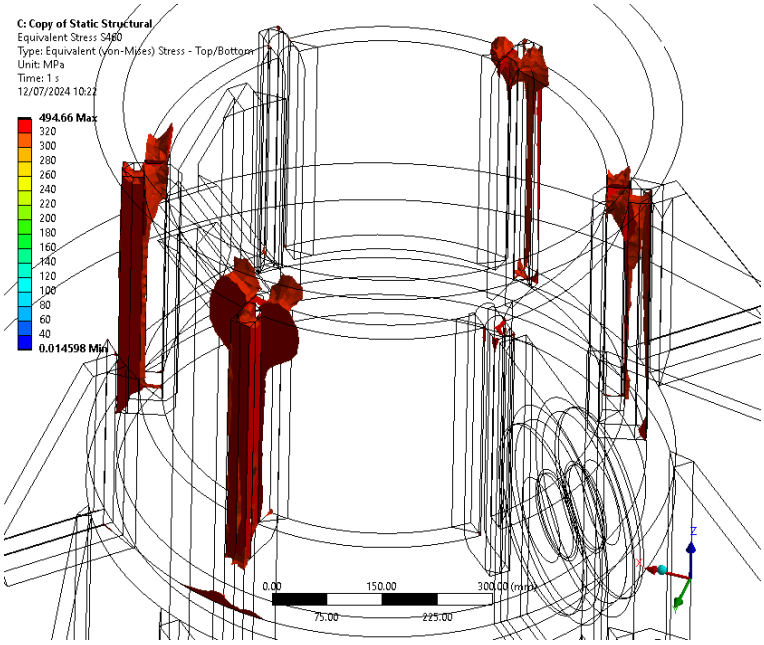


Figure 11.14 – Stabbing guide FE model – S460 VM Stresses > 320 MPa – lower connection

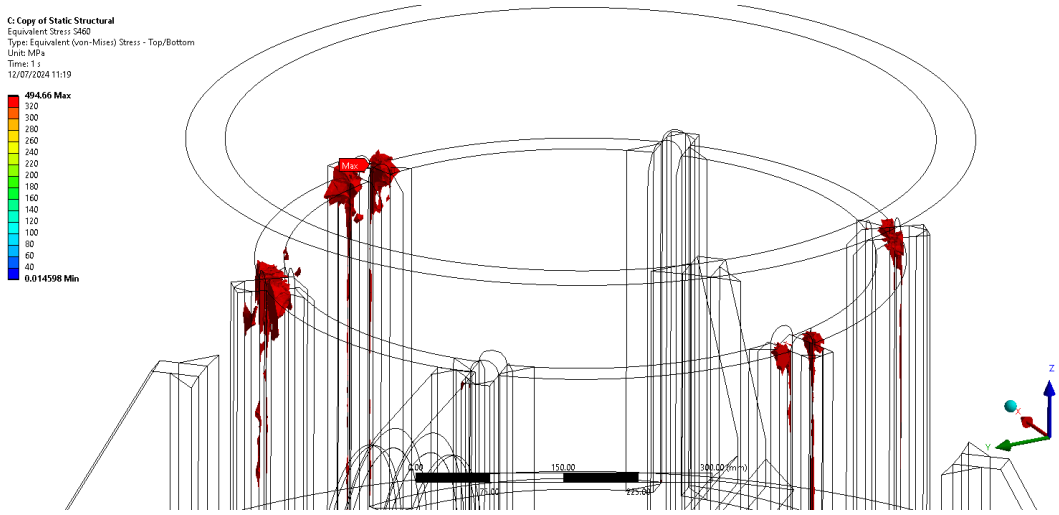


Figure 11.15 – Stabbing guide FE model – S460 VM Stresses > 400 MPa

For structural checks, it is taken the most restrictive value of admissible stress, both for tubular and non-tubular elements, on the safe side: for t>16 mm, F_y = 400 MPa, therefore:

- Members: $\sigma_{adm} = \Phi * F_y = 0.85*400 = 340$ MPa
- Welds: $\sigma_{adm} = F_y/\gamma_{m2} = 400/1.25 = 320$ MPa

It is observed that VM stresses are largely lower than the admissible stress σ_{adm} = 320 MPa for global checks. For these parts structural checks are satisfied.

The admissible value is exceeded at the edges of the 20mm cone, with an area greater than 2tx2t, but not affecting the whole thickness of the plate (superficial layers).

The admissible value is also exceeded at the lower connection of the stabbing guide with the Φ800x40 stab, with peak values that exceed the F_y = 400 MPa value.

Since structural checks are not fulfilled in these areas, deformations are investigated, with focus on these portions.

For the hot spot areas, elasto-plastic strains are measured according to the performed non-linear analysis. They shall be less than 5%, as per design criteria defined in Sec. 8.3.

Deformations are investigated in sec.11.2.3

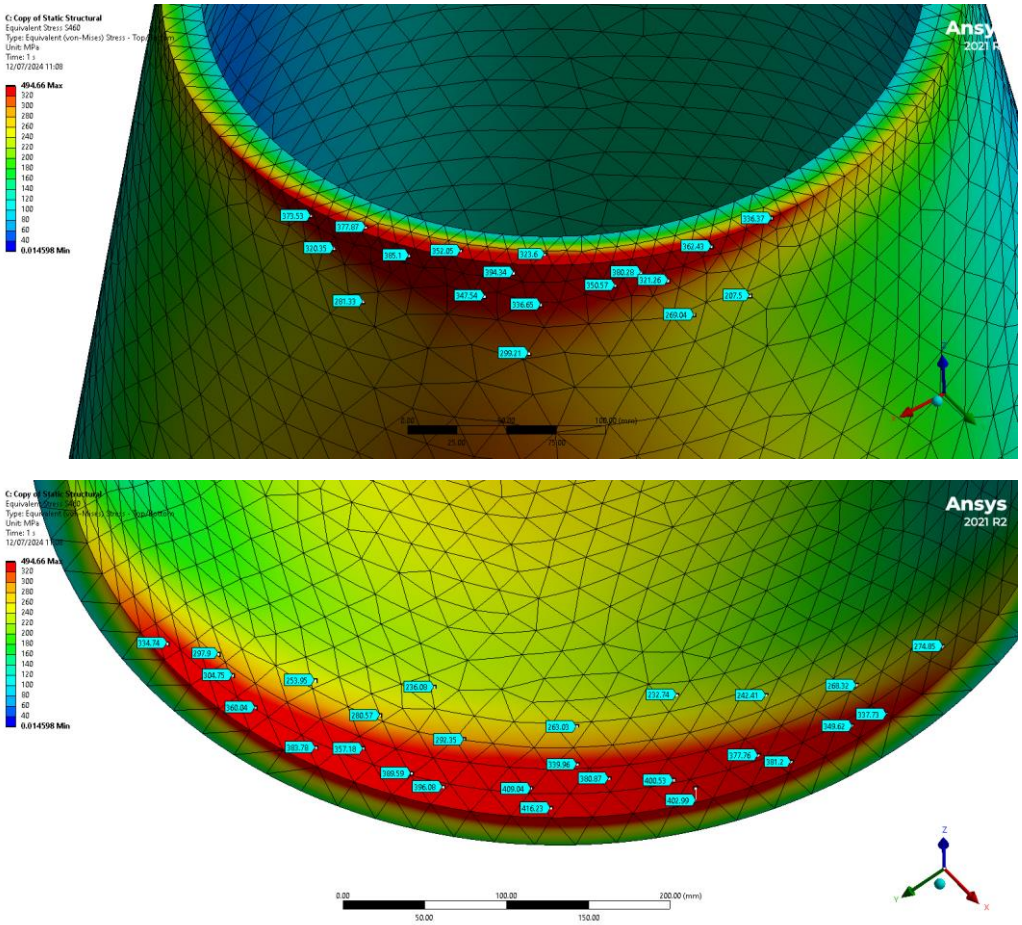


Figure 11.16 – Stabbing guide FE model – S460 VM Stresses > 320 MPa – upper cone peak stress areas

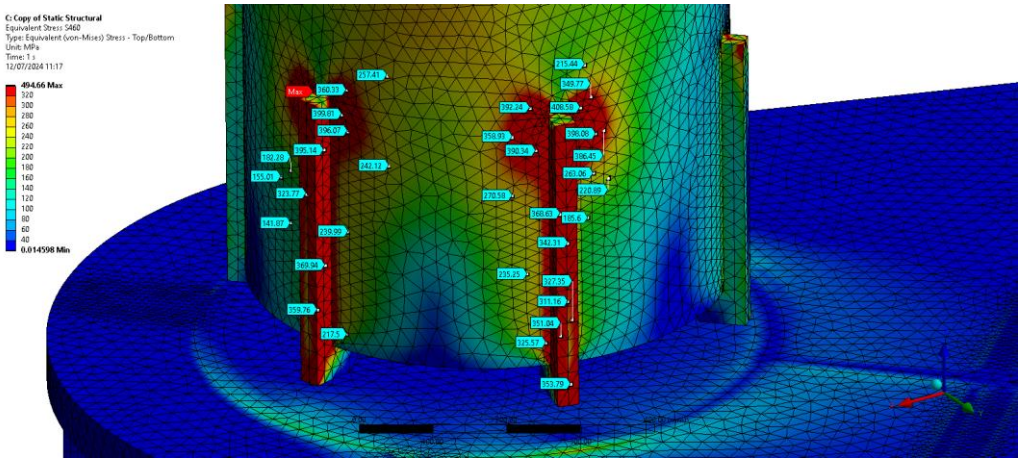


Figure 11.17 – Stabbing guide FE model – S460 VM Stresses > 320 MPa – lower connection peak stress areas

11.2.2 S355 Steel structural checks

Here are reported FEA results and checks for the S355 steel part (upper part of stabbing guide).

In the following figures, the Von Mises stresses are plotted.

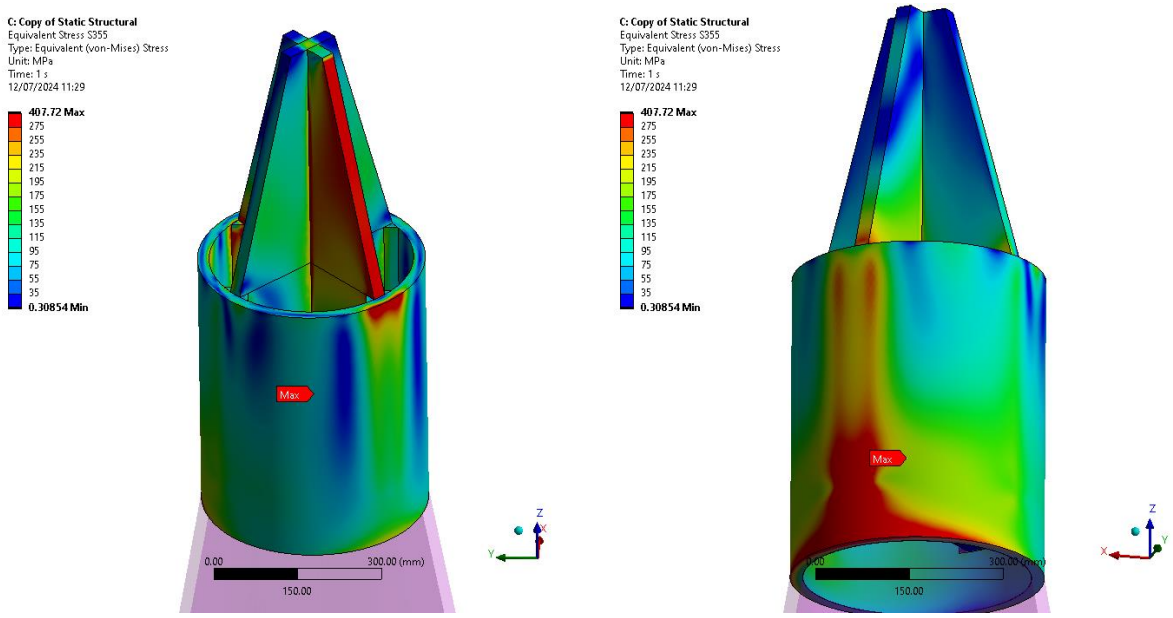


Figure 11.18 – Stabbing guide FE model – S355 VM Stresses

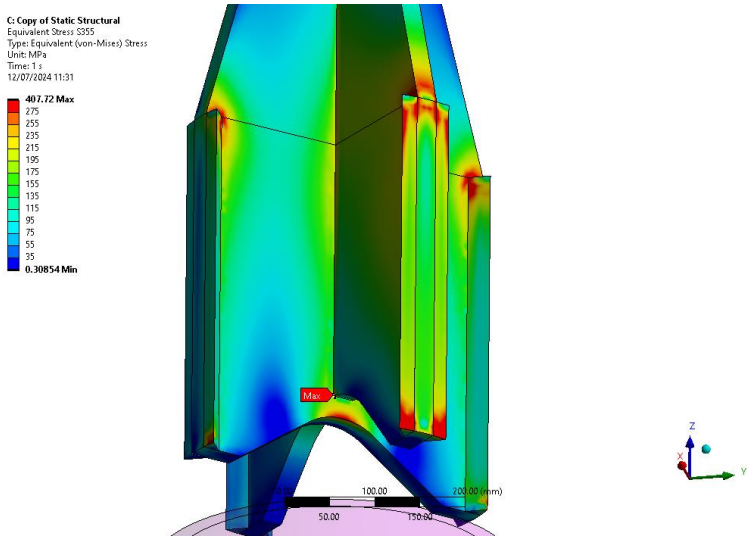


Figure 11.19 – Stabbing guide FE model – S355 VM Stresses – internal plates/welds

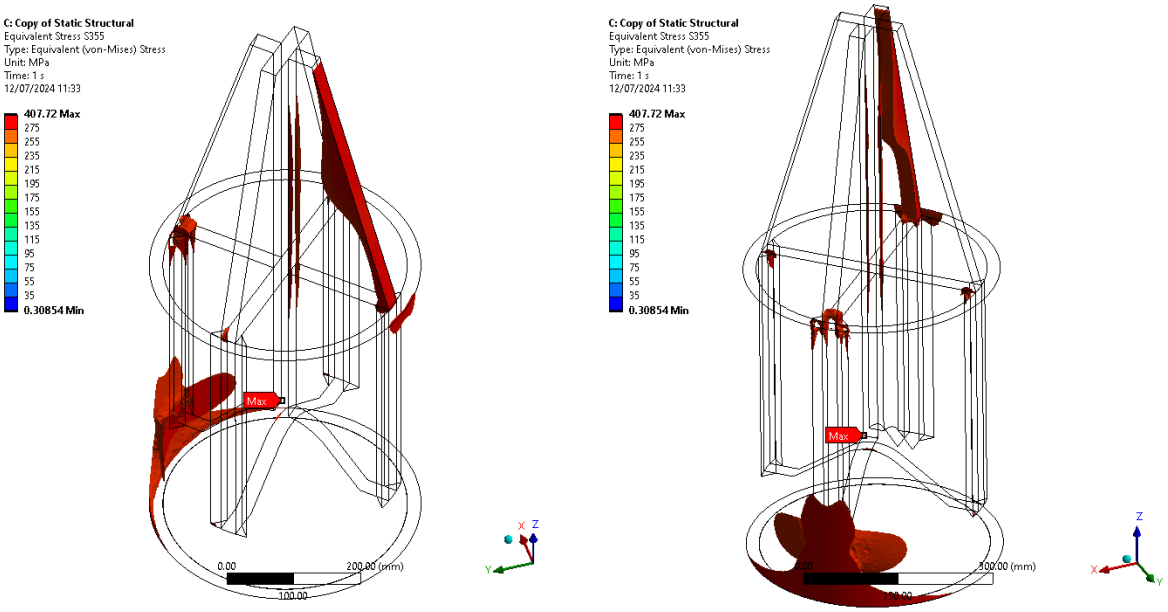


Figure 11.20 – Stabbing guide FE model – S355 VM Stresses > 276 MPa

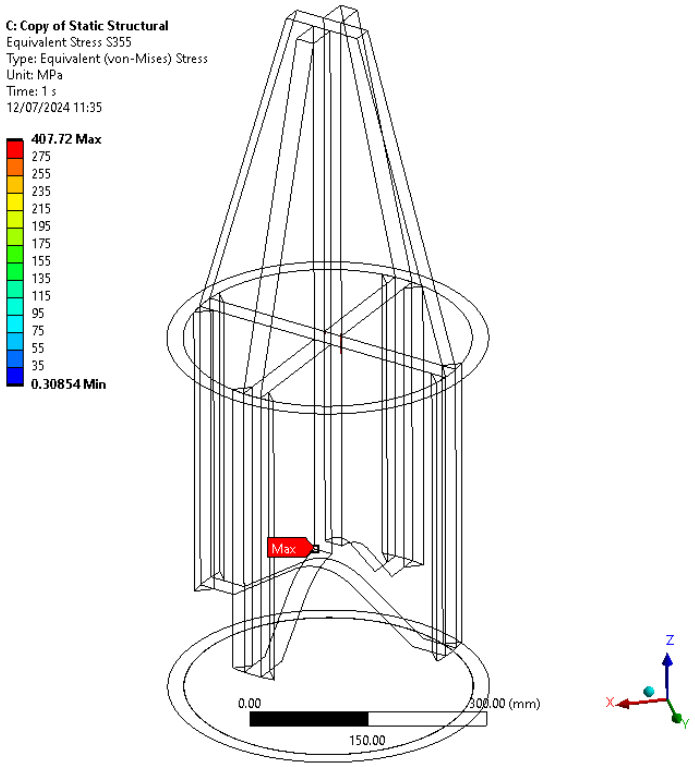


Figure 11.21 – Stabbing guide FE model – S355 VM Stresses > 345 MPa

For structural checks, it is taken the most restrictive value of admissible stress, both for tubular and non-tubular elements, on the safe side: for t>16 mm, F_y = 345 MPa, therefore:

- Members: $\sigma_{adm} = \Phi * F_y = 0.85*345 = 293$ MPa
- Welds: $\sigma_{adm} = F_y/\gamma_{m2} = 345/1.25 = 276$ MPa

It is observed that VM stresses are lower than the admissible stress σ_{adm} = 276 MPa for global checks, for the largest part of the S355 material. For these parts structural checks are satisfied. No part exceeds the yielding stress.

The admissible value is exceeded in a large part of the tubular element, with an area greater than 2tx2t, but not affecting the whole thickness of the plate (superficial layer).

The admissible value is also exceeded at the vertical stiffener where the impact load is applied, with an area greater than 2tx2t and affecting part of the thickness.

The admissible value is also exceeded at the upper corners of the 20mm plates connected to the Φ406 segment with part of the fillet weld involved. In this case, the area is confined in the dimensions 2tx2t, so checks are considered satisfied for these portions.

Since structural checks are not fulfilled in some areas, deformations are investigated, with focus on these portions. For the hot spot areas, elasto-plastic strains are measured according to the performed non-linear analysis. They shall be less than 5%, as per design criteria defined in Sec. 8.3.

Deformations are investigated in sec.11.2.3.

11.2.3 Deformations check

For the areas of the stabbing guide not fulfilling the structural checks, as detailed in sec.11.2.1 and 11.2.2, deformations are investigated.

Elasto-plastic strains are measured according to the performed non-linear analysis. They shall be less than 5%, as per design criteria defined in Sec. 8.3.

In the following figures, plastic strains and total elasto-plastic strains are plotted.

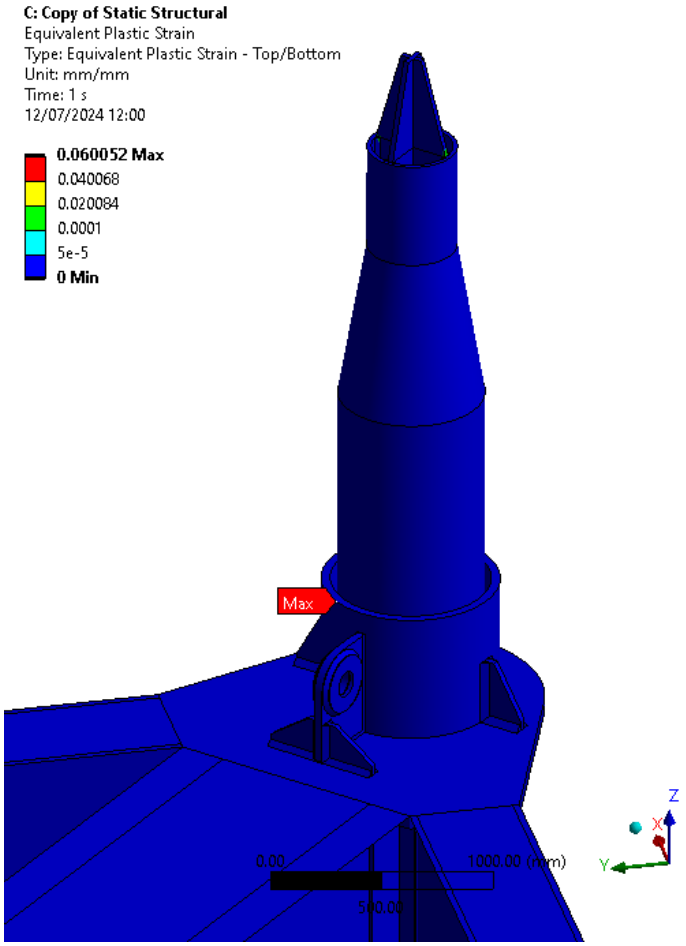
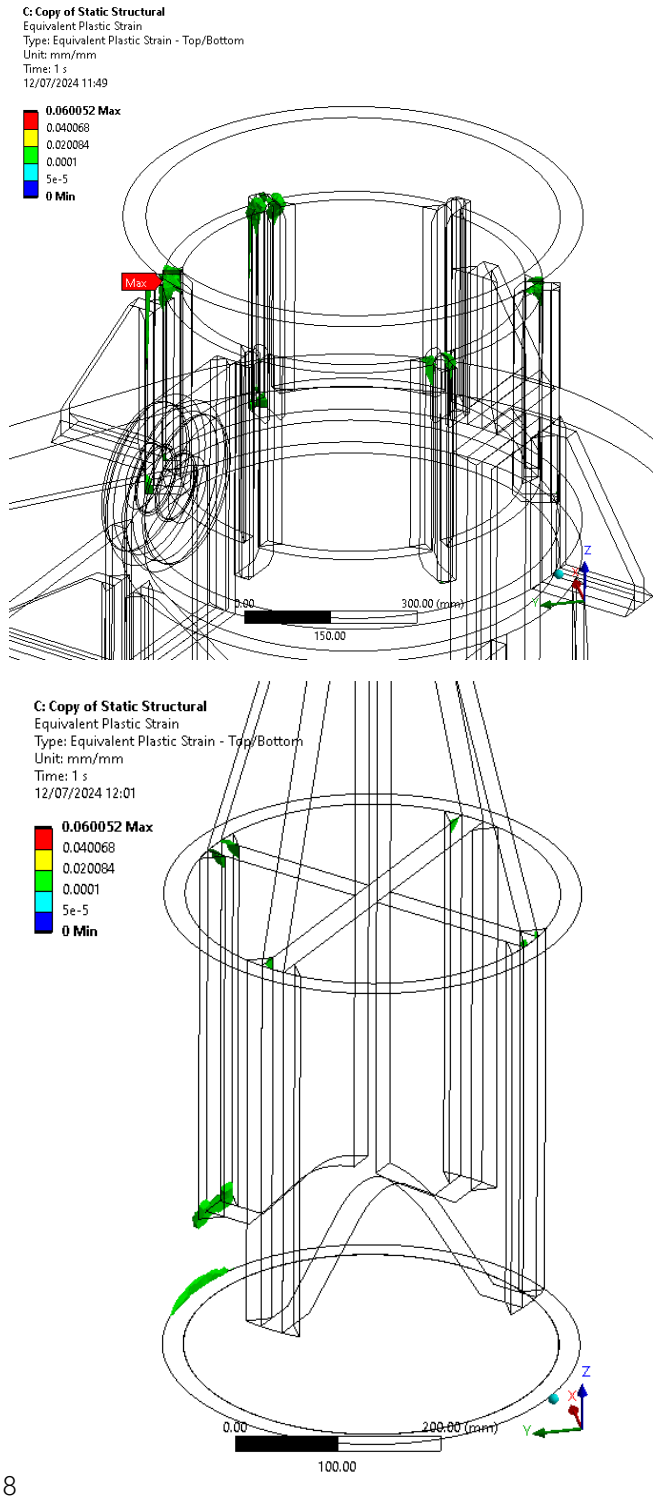


Figure 11.22 – Stabbing guide FE model – Plastic strains



8

Figure 11.23 – – Stuabbing guide FE model – Plastic strains > 0

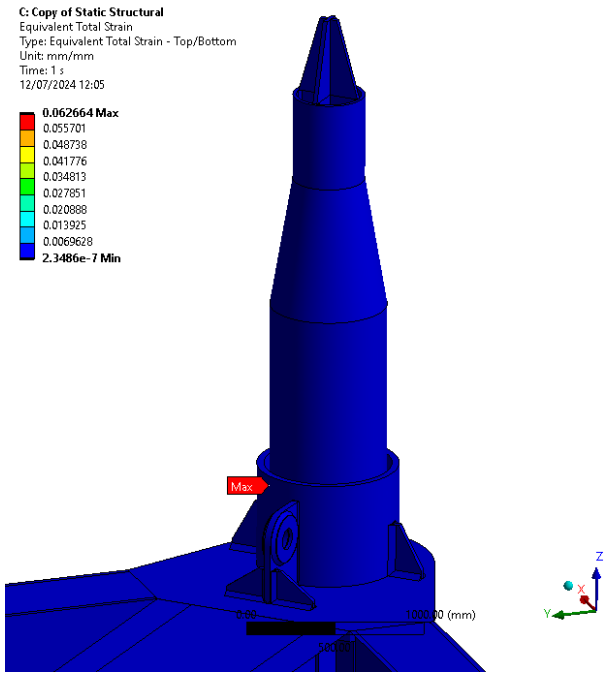


Figure 11.24 – Stabbing guide FE model – total strains

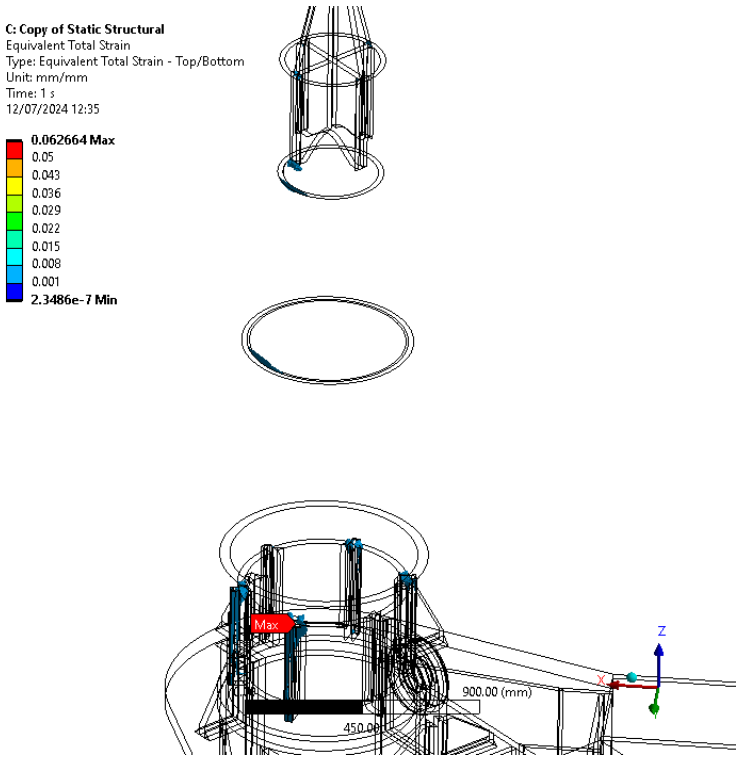


Figure 11.25 – Stabbing guide FE model – total strains > 0.2%

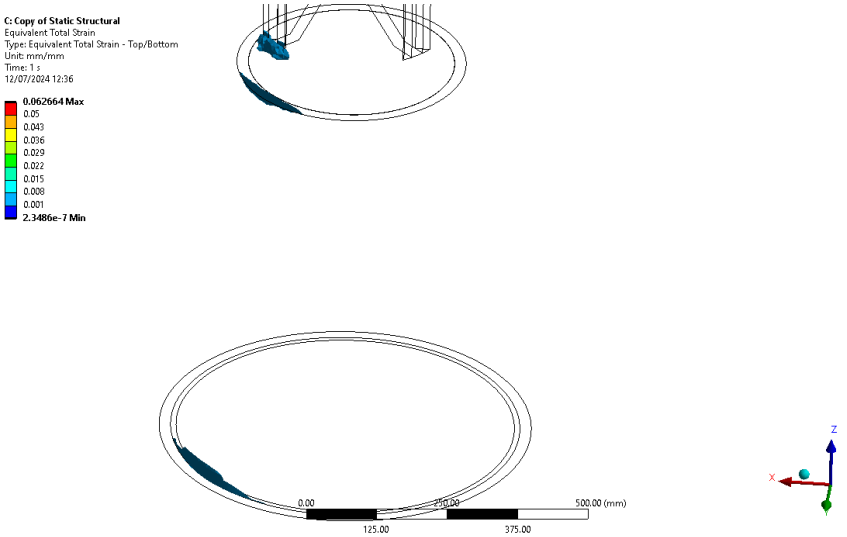


Figure 11.26 – Stabbing guide FE model – total strains > 0.2% upper cone

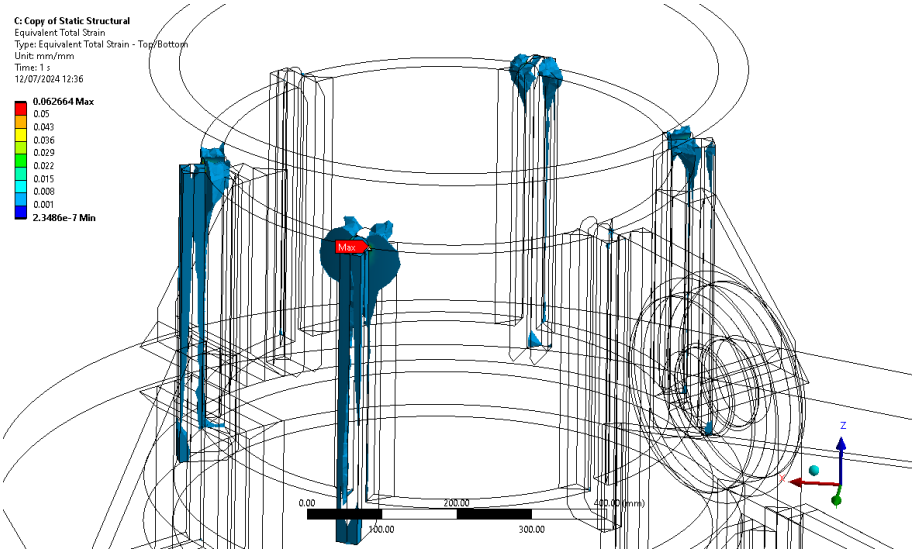


Figure 11.27 – Stabbing guide FE model – total strains > 0.2% lower connection

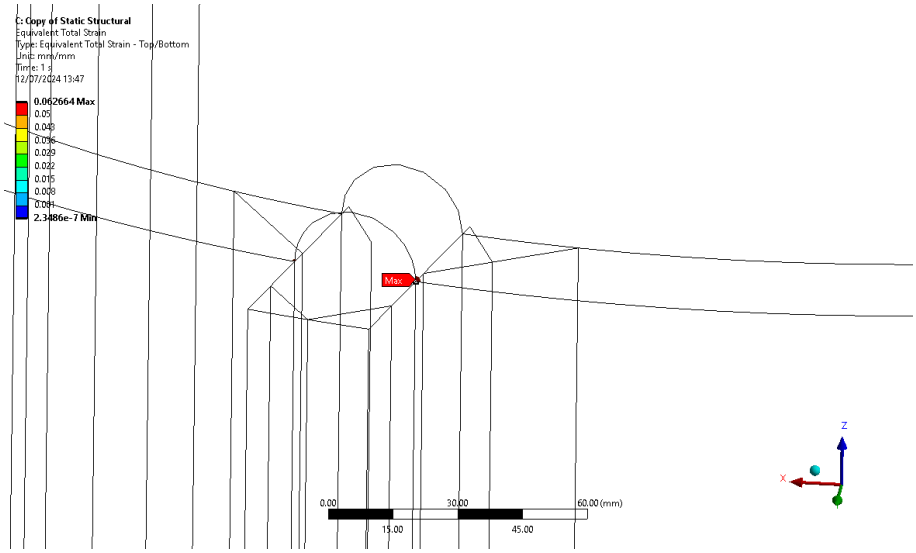


Figure 11.28 – Stabbing guide FE model – total strains > 5%

Plastic strains occur at the connection stiffeners between the stabbing guide and the Φ800x40 stab, and in the upper spike in some limited portions of the Φ406x20 tube and in the 20mm crossing plates at the intersection with the tube.

Total strains are within the admissible value. ε_{adm} =0.05 mm/mm. There is only a peak in the corner of one lower connecting plate which has value 0.06 mm/mm but it is negligible for the extension of the peak (punctual, as per Figure 11.28).

Checks are fulfilled, the stabbing guide has enough resistance capacity to withstand impact loads during installation phase.

12 TP-TOPSIDE CONNECTION CHECK DURING SET DOWN

According to section 10.6 of Design Basis, Ref. /18/, the set-down of topside over TP is considered to happens over a diagonal supporting 100% of full topside load.

The WCR, Ref /19/, states topside weight in lifting condition = 1850 ton.

BLPH - TOPSIDE (NO LIVE LOAD AND SNOW) - Summary by Phase

Phase Code	Phase Description	Base Wt. (t)	Center of Gravity			Factored Wt. (t)	Center of Gravity			Design Net Wt. (t)	Center of Gravity			Design Gross (Expected) Wt. (t)	Center of Gravity		
			Northing (m)	Easting (m)	Elevation (m)		Northing (m)	Easting (m)	Elevation (m)		Northing (m)	Easting (m)	Elevation (m)		Northing (m)	Easting (m)	Elevation (m)
LO	Loadout	1654	20,1	8,0	29,0	1850	20,1	8,0	29,0	1850	20,1	8,0	29,0	1850	20,1	8,0	29,0
ST	Sea Transport	1654	20,1	8,0	29,0	1850	20,1	8,0	29,0	1850	20,1	8,0	29,0	1850	20,1	8,0	29,0
OP	Operating weight (Dry + Contents)	1655	20,1	8,0	29,1	1851	20,1	8,0	29,0	1851	20,1	8,0	29,0	1851	20,1	8,0	29,0
L	Lifting	1654	20,1	8,0	29,0	1850	20,1	8,0	29,0	1850	20,1	8,0	29,0	1850	20,1	8,0	29,0

Figure 12.1 – WCR - topside

Considering the CoG position the max load insisting on one support is $=1850*(8^2+8.1^2)^{0.5}/22.6=932$ ton, and lateral load 5% of vertical one.

Assumed values: Fv = 932 t = 9150 kN

Fh = 0.05*1850 = 92.5 t = 910 kN

12.1 LOCAL CHECK

The proposed bevel preparation is shown in following picture:

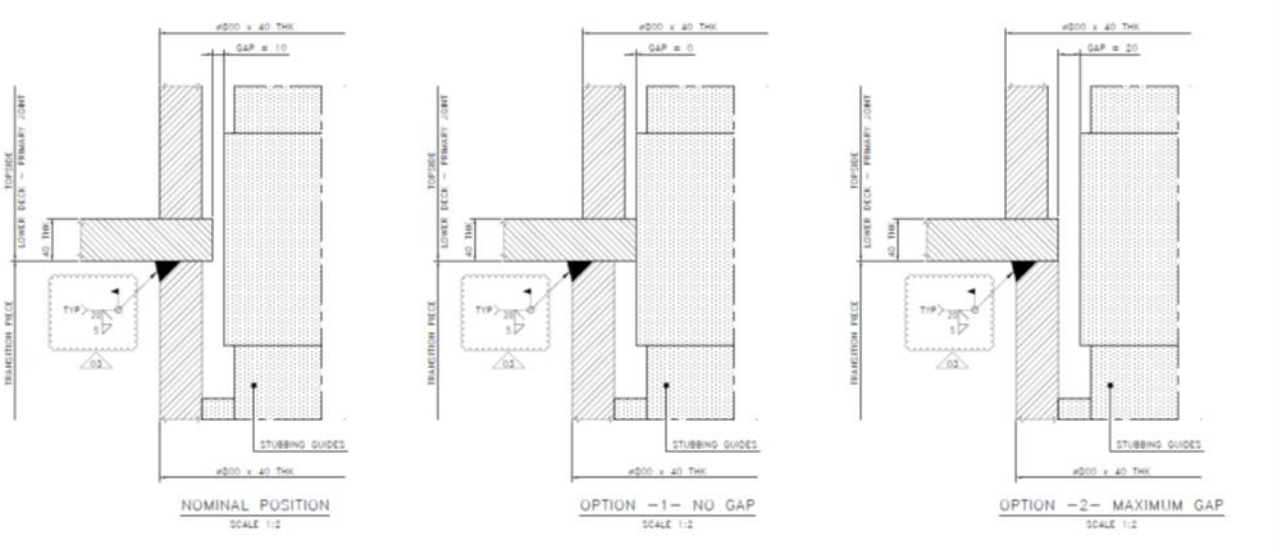


Figure 12.2 – TP-topside connection, ref. 123A90-DDL-ST-DD-90054

CYLINDRICAL MEMBER CHECK FOR STRENGTH AND STABILITY								
Element:			Material:	API5LX60		Fy =	420	N/mm^2
D =	800.00	mm	Area =	49,009	mm2			
t =	20.00	mm	Inertia =	3.7296E+09	mm4			
Length =	0.60	m	Ky =	2.00		Cmy =	1.00	
			Kz =	2.00		Cmz =	1.00	
D/t =	40		(Kl/r)y =	4		F'$_e$y =	54,411	N/mm^2
Cc (Fy) =	98		(Kl/r)z =	4		F'$_e$z =	54,411	N/mm^2
Loading:						A.s.i.f. =	1.00	
Nx =	9,150.0	kN	fa =	186.7	N/mm^2	Ft/Fy =	0.60	eq.3.2.1-1
My =	0.0	kNm	fby =	0.0	N/mm^2			
Mz =		kNm	fbz =	0.0	N/mm^2	Fb/Fy =	0.70	eq.3.2.3-1b
Ty =		kN						
Tz =	910.0	kN	fv =	37.1	N/mm^2	Fv/Fy =	0.40	eq.3.2.4-2
Mt =		kNm	fvt =	0.0	N/mm^2	Fvt/Fy =	0.40	eq.3.2.4-4
Maximum interaction ratios:			**I.R. $_v$ =**	**0.22**	Shear	**I.R. $_b$ =**	**0.74**	Stab.
								eq.3.3.1-2
					Check Ok.			

Figure 12.3 – Beveled section check

12.2 TP GLOBAL CHECK

The TP global check has been performed starting from the TP SACS model used in Loadout and Lifting analyses, and accounting for the same basic load conditions, ref. /25/ with different restrains.

The set down force is applied with load factor 1.30, over the two diagonals alternately.

The horizontal force is applied 100% in primary pin and 50% in secondary touching point. Local effects on TP have been verified in this report, see the above section 11, with FEM analysis, showing negligible effect on TP main girders and connection joint.

12.2.1 Load combinations

Two load combinations were generated, summing up the TP weight with the impact loads:

a) STD1 = TPLF*1.30 + DIA1*1.30

b) STD2 = TPLF*1.30 + DIA2*1.30

With TPLF as described in report ref. /25/

Load combination TPLF includes maximum permanent loads summed up in G001 with installation aids.

TP in Lifting conditions – Load combination TPLF		
Basic Load Cases	Description	Load Factors
G001	TP PERMANENT LOAD BASIC COMBINATION	1.02
AIDS	INSTALLATION AIDS AND RIGGING LOADS	1.00

Table 12.1 – Lifting Basic Combination TPLF

************* SEASTATE LOAD CASE CENTER REPORT *************
RELATIVE TO STRUCTURAL ORIGIN

LOAD CASE	LOAD LABEL	********* X - DIRECTION ******** FORCE (KN)	X (M)	Y (M)	Z (M)	********* Y - DIRECTION ********* FORCE (KN)	X (M)	Y (M)	Z (M)	********* Z - DIRECTION ******** FORCE (KN)	X (M)	Y (M)	Z (M)
G001		0.00				0.00				-2440.14	-0.39	-0.77	5.84
AIDS		0.00				0.00				-52.00	0.00	0.00	6.57
TPLF		0.00				0.00				-2540.94	-0.39	-0.75	5.85

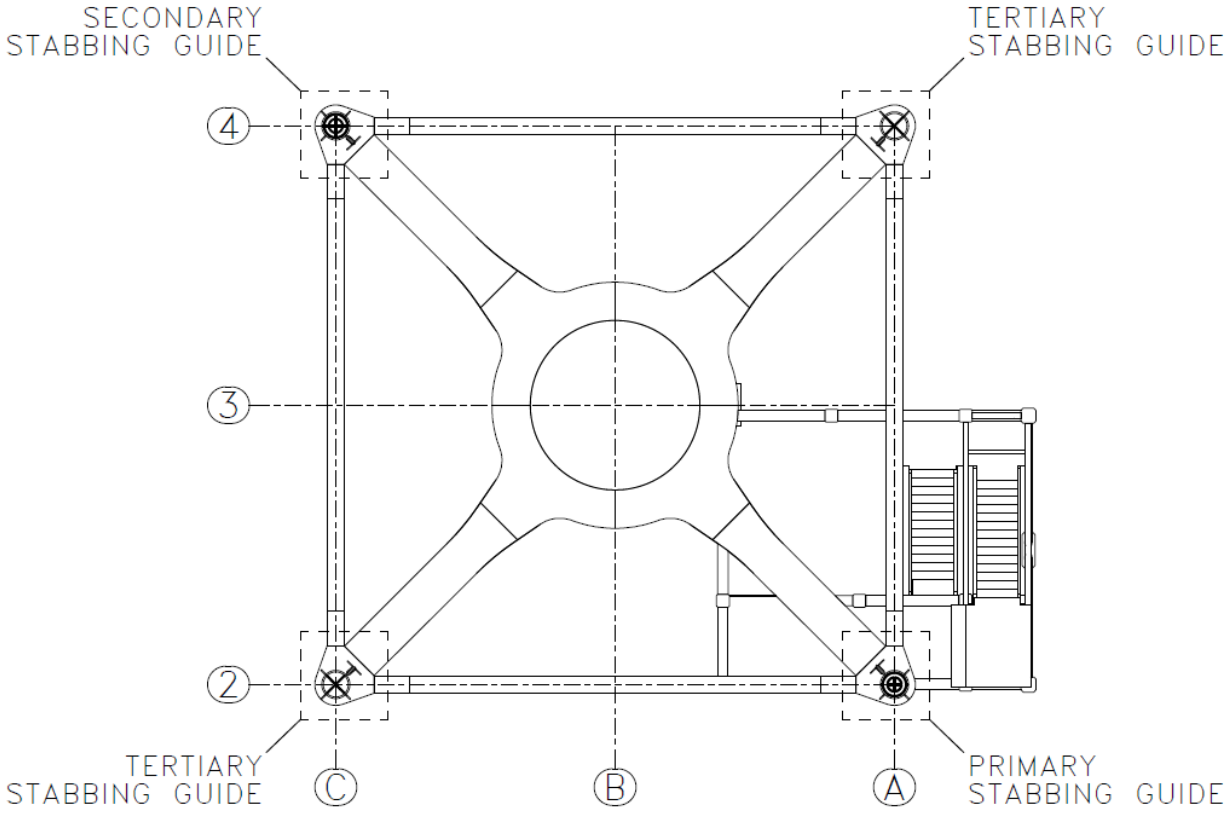


Figure 12.4 – TP plan view – Stabbing guides position

Load condition DIA1:

Vertical TOPSIDE load applied on diagonal C2 – A4: Fz = -9150 kN each

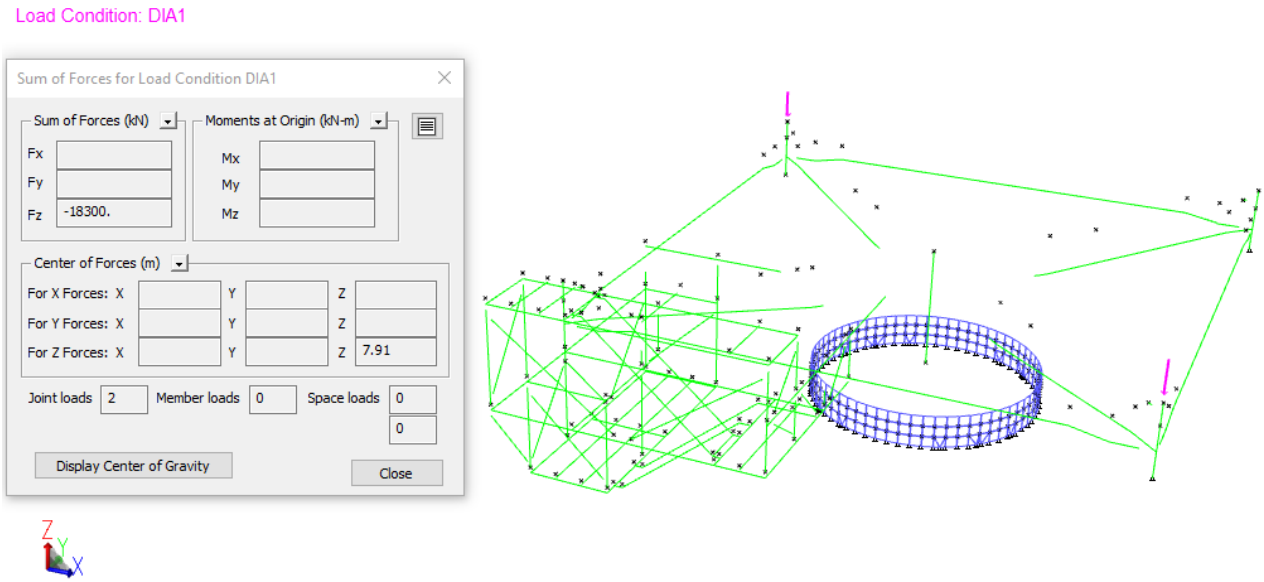


Figure 12.5 – load condition DIA1

Load condition DIA2:

Vertical and horizontal TOPSIDE load applied on diagonal A2 – C4.

Point A2: Fz = -9150 kN
Fy = 910 kN
Mx = 910*3.25m = 2957.5 kNm

Point C4: Fz = 9150 kN
Fy = 910/2 = 455 kN
My = 910*2.80*0.5 = 1274 kNm

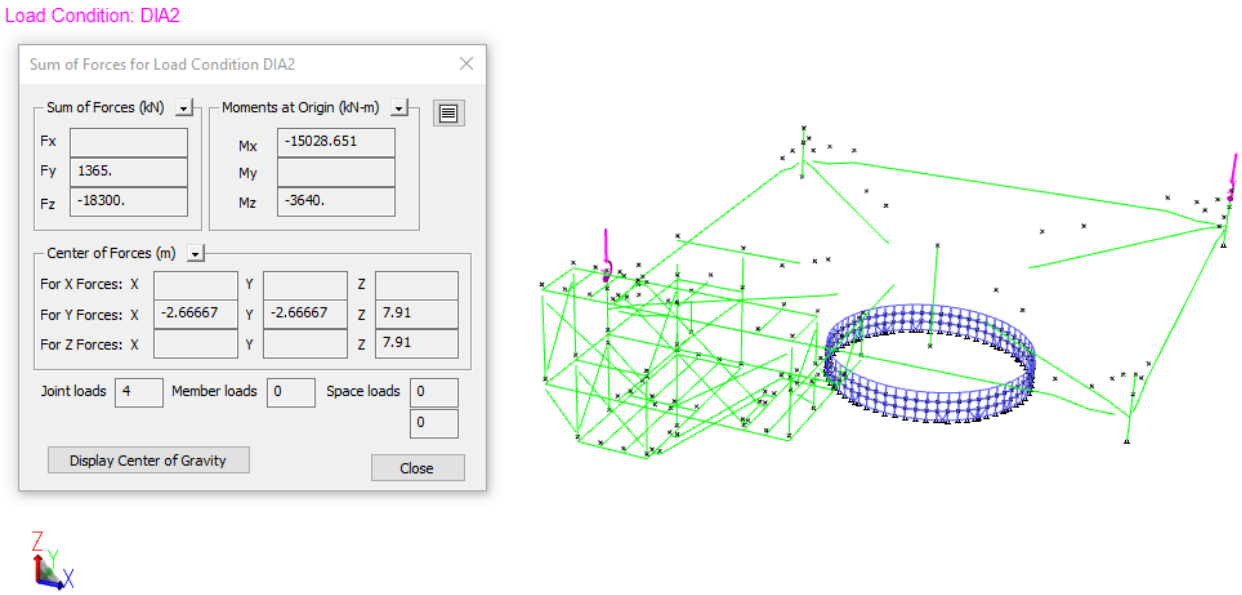


Figure 12.6 – load condition DIA2

12.2.2 Boundary Conditions

TP is fully simply supported at flange level (111000).

12.3 STRUCTURAL CHECKS

The results of the performed analysis in terms of members' Unity Check (UC) and Joints' Punching Shear Unity Checks are detailed in the following.

The base reactions are reported below:

SACS-IV SYSTEM FIXED JOINTS REACTION FORCES AND MOMENTS SUMMARY
*** MOMENTS SUMMED ABOUT ORIGIN ***

LOAD CASE	FORCE(X)	FORCE(Y)	FORCE(Z)	MOMENT(X)	MOMENT(Y)	MOMENT(Z)
	********** KN **********			********** KN-M **********		
STD1	-0.000	-0.000	27093.225	-2489.128	1277.526	-0.000
STD2	0.000	-1774.500	27093.215	17048.143	1277.530	4731.991

12.3.1 Member checks

All members were checked according to ISO 19902 and Eurocode 3, Ref./6/and /8/, respectively for tubular and non-tubular elements.

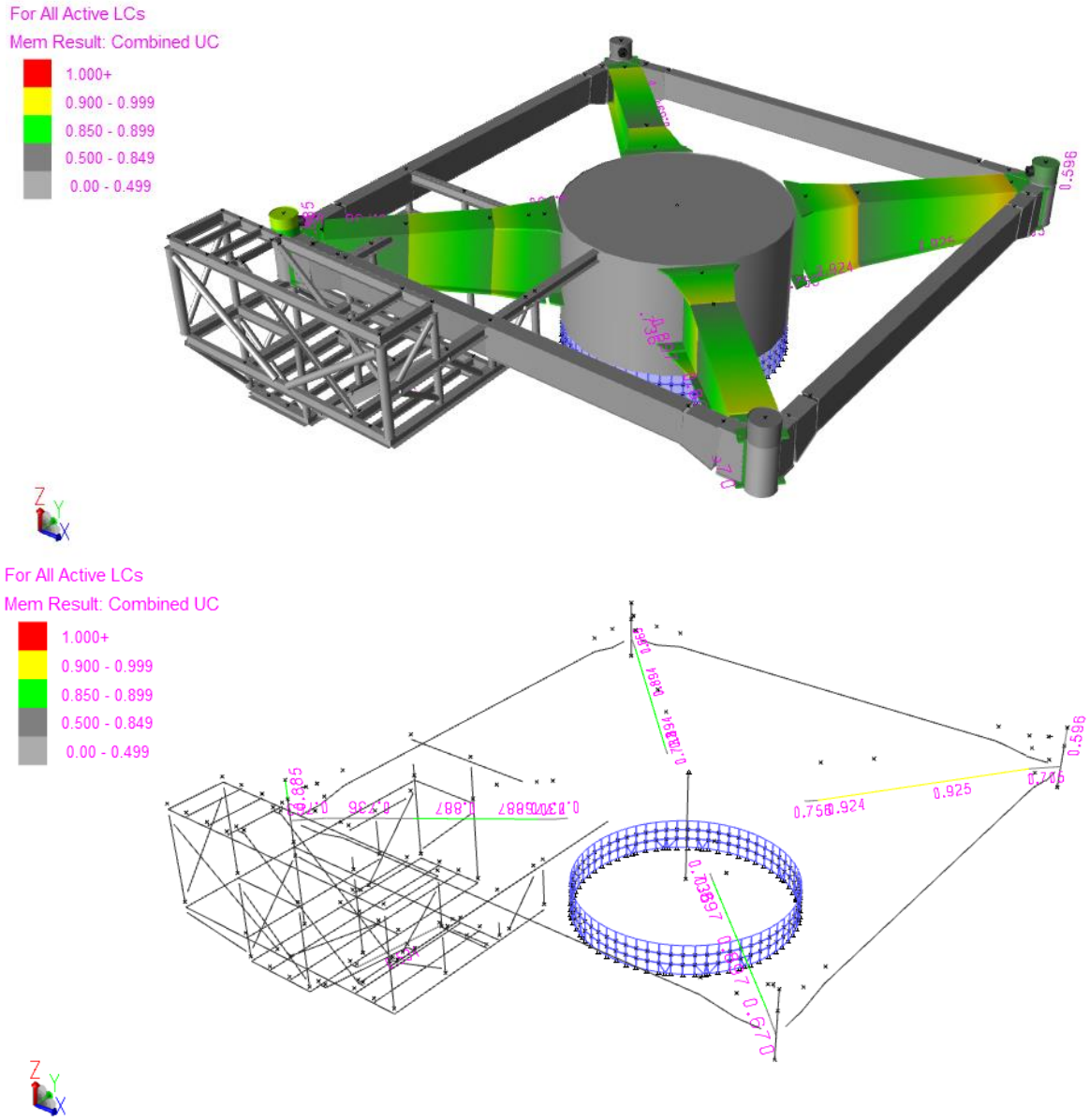


Figure 12.7 – Member UC

SACS-IV MEMBER UNITY CHECK RANGE SUMMARY

GROUP III - UNITY CHECKS GREATER THAN 0.90 AND LESS THAN 999.00

HIGHEST MEMBER	GROUP ID	MAXIMUM COMBINED UNITY CK	LOAD COND NO.	DIST FROM END	AXIAL STRESS N/MM2	BENDING STRESS Y N/MM2	BENDING STRESS Z N/MM2	SHEAR FORCE FY KN	SHEAR FORCE FZ KN	KLY/RY	KLZ/RZ	SECOND-HIGHEST UNITY CHECK	SECOND-HIGHEST LOAD COND	THIRD-HIGHEST UNITY CHECK	THIRD-HIGHEST LOAD COND
TC41-TC42	T02	0.924	STD2	1.6	-13.16	-332.62	-5.96	88.78	12024.46	15.1	18.0	0.067	STD1	-0.000	
TC42-TC43	T03	0.925	STD2	0.0	-13.21	-332.64	-5.95	88.78	12020.97	22.8	23.0	0.077	STD1	-0.000	

* * * M E M B E R G R O U P S U M M A R Y * * *
ISO 19902:2007 / EC3:2005
(EN 1993-1-1:2005/NA NONE)

GRUP ID	CRITICAL MEMBER	LOAD COND	MAX. UNITY CHECK	DIST FROM END M	* APPLIED STRESSES * AXIAL N/MM2	BEND-Y N/MM2	BEND-Z N/MM2	*** ALLOWABLE STRESSES *** AXIAL N/MM2	EULER N/MM2	BEND-Y N/MM2	BEND-Z N/MM2	CRIT COND	EFFECTIVE LENGTHS KLY M	KLZ M	CM * VALUES * Y	Z
HA3	SC08-T000	STD1	0.23	2.8	5.1	-54.9	-25.4	322.7	532.6	338.6	487.7	BN+AX	2.80	4.60	0.00	0.00
IP2	SM17-SM23	STD2	0.10	0.7	-12.6	11.6	13.6	322.7	3220.0	344.3	496.2	BN+AX	1.35	0.67	0.60	0.60
IP3	SM07-SM16	STD2	0.39	0.0	-10.8	-67.3	11.3	68.9	90.7	303.4	1018.1	BN+COM	5.30	5.30	0.42	0.40
L15	SM15-SM14	STD2	0.02	0.0	-0.1	0.3	-10.4	190.5	673.5	430.6	512.9	BN+COM	1.20	1.20	0.00	0.00
M01	TMJ1-T000	STD2	0.07	0.0	-18.9	-0.5	-1.7	301.4	4865.8	366.6	366.6	ACBI	50.00	50.00	0.85	0.85
ST3	SL06-SM04	STD1	0.08	2.8	-6.6	-0.4	28.1	300.8	836.1	448.9	448.9	ACBI	2.81	2.81	0.85	0.85
ST4	SM32-SU12	STD1	0.03	1.3	-0.0	-11.4	0.3	.3E+03	.1E+05	.5E+03	.5E+03	ACBI	1.27	1.27	0.85	0.85
ST5	SC09-SM27	STD2	0.33	0.4	5.4	-131.9	-50.1	.3E+03	.2E+05	.5E+03	.5E+03	TN+BN	0.80	0.40	0.85	0.85
T00	T000-TC41	STD2	0.76	0.4	-7.1	-264.7	-6.2	345.5	9524.8	365.1	589.6	BN+AX	16.00	8.00	0.98	0.98
T01	T029-SC05	STD2	0.36	0.0	44.5	-90.5	13.0	363.6	296.2	438.2	442.2	BN+AX	16.00	16.00	0.00	0.00
T02	TC41-TC42	STD2	0.92	1.6	-13.2	-332.6	-6.0	345.5	6244.3	380.1	524.8	BN+AX	16.00	8.00	0.91	0.89
T03	TC42-TC43	STD2	0.92	0.0	-13.2	-332.6	-6.0	345.5	3809.8	380.2	524.8	BN+AX	16.00	8.00	0.67	0.55
T04	TA23-TA24	STD2	0.76	0.0	-17.2	-55.7	-5.9	314.2	2716.3	368.6	467.1	SHEAR	16.00	8.00	0.40	0.92
T05	T022-T019	STD1	0.24	0.0	34.2	-50.8	14.0	363.6	337.9	438.2	442.2	BN+AX	16.00	16.00	0.00	0.00
T06	TA24-T028	STD2	0.23	0.0	24.2	-52.0	11.3	345.5	260.3	401.9	437.0	BN+AX	16.00	16.00	0.00	0.00
T07	T028-T029	STD2	0.35	1.0	42.8	-90.5	13.4	363.6	337.9	438.2	442.2	BN+AX	16.00	16.00	0.00	0.00
T2A	TA21-TA2A	STD2	0.78	0.5	-14.2	-265.3	-9.0	345.5	6811.6	368.6	580.8	BN+AX	16.00	8.00	0.97	0.96
T2B	TA2A-TA22	STD2	0.89	1.1	-17.9	-309.7	-10.7	345.5	5604.3	380.2	524.8	BN+AX	16.00	8.00	0.93	0.97
T3A	TA22-TA2B	STD2	0.89	0.0	-17.9	-309.7	-10.7	345.5	4031.1	380.2	524.8	BN+AX	16.00	8.00	0.82	0.93
T3B	TA2B-TA23	STD2	0.74	2.4	-21.8	-55.6	-8.4	320.2	3682.4	372.5	478.0	SHEAR	16.00	8.00	0.67	0.90
TPS	TA24-TU02	STD2	0.88	0.0	-124.6	0.0	-263.5	.3E+03	.1E+06	.5E+03	.5E+03	ACBI	1.20	1.20	0.85	0.85
UN2	SS03-SM12	STD2	0.53	0.0	-14.9	37.7	0.0	132.7	266.4	89.4	490.2	LATBUC	4.95	2.00	0.51	1.00

12.3.2 Plate checks

No criticalities were found on plates.

SACS-IV SYSTEM PLATE STRESS UNITY CHECK RANGE SUMMARY

GROUP III - UNITY CHECKS GREATER THAN 0.90 AND LESS THAN 999.00

** NO UNITY CHECKS IN THIS GROUP **

***** PLATE STRESS GROUP SUMMARY *****
N/MM2

GRUP	CRIT. PLATE	TYPE	MAX UNITY CHECK	LOAD COND NO.	**** MEMBRANE **** SX	SY	TXY	BENDING-UPPER SURF. ** SX	SY	TXY	MAX ** VM	X-STIFFENER S-TOP	S-BOT	Y-STIFFENER S-TOP	S-BOT
M65	B196	ISO	0.080	STD2	-3.2	-23.7	-1.3	1.8	6.0	-0.0	27.6				

12.3.3 ULS Joint checks

All tubular joints punching shear checks and strength checks are detailed in following SACS excerpt and provide utilization factors lower than 1.0, according to ISO19902, ref./6/. All the verifications were performed against combination with CF = 1.3.

* * J O I N T C A N S U M M A R Y * *

(UNITY CHECK ORDER)

	ORIGINAL					LOAD DESIGN					STRENGTH ANALYSIS		
JOINT	DIAMETER (CM)	THICKNESS (CM)	YLD STRS (N/MM2)	LOAD UC	STRN UC	DIAMETER (CM)	THICKNESS (CM)	YLD STRS (N/MM2)	LOAD UC	STRN UC	BRACE JOINT	LOAD CASE
SM30	21.910	1.031	355.000	0.096	0.113	21.910	1.031	355.000	0.096	0.113	SM33	STD1
SL18	21.910	1.031	355.000	0.095	0.111	21.910	1.031	355.000	0.095	0.111	SM03	STD2
SM01	21.910	1.031	355.000	0.064	0.075	21.910	1.031	355.000	0.064	0.075	SL02	STD2
SL06	21.910	1.031	355.000	0.062	0.073	21.910	1.031	355.000	0.062	0.073	SM02	STD2
SM02	21.910	1.031	355.000	0.059	0.069	21.910	1.031	355.000	0.059	0.069	SL03	STD2
SL01	21.910	1.031	355.000	0.051	0.059	21.910	1.031	355.000	0.051	0.059	SM05	STD2
SM03	21.910	1.031	355.000	0.050	0.059	21.910	1.031	355.000	0.050	0.059	SL18	STD2
SM05	21.910	1.031	355.000	0.043	0.050	21.910	1.031	355.000	0.043	0.050	SL01	STD2
SU01	21.910	1.031	355.000	0.028	0.033	21.910	1.031	355.000	0.028	0.033	SM04	STD2
SU11	21.910	1.031	355.000	0.026	0.030	21.910	1.031	355.000	0.026	0.030	SM33	STD2
SU12	27.310	1.270	355.000	0.023	0.027	27.310	1.270	355.000	0.023	0.027	SM11	STD2
SU02	27.310	1.270	355.000	0.016	0.018	27.310	1.270	355.000	0.016	0.018	SM10	STD1

This document undergoes an issuance cycle within EDMS Fusionlive". Its distribution is electronically controlled. Once printed shall be considered as "NOT CONTROLLED" copy