Base Load Power Hub (BLPH)

Topside Joint Check Design Report

Document Number: 123A90-DDL-ST-RE-90005

Document Class: 1

02	05/11/2024	Accepted for Construction (AFC)	F. Biancheri	A. Balzarotti	D. LoSterzo
01	05/08/2024	Issue for Acceptance (IFA)	F. Biancheri	A. Balzarotti	D. LoSterzo
00	02/05/2024	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 9

1.1 PROJECT OVERVIEW 9

1.1.1 Project Location 9

1.1.2 Project Concept 10

2 DOCUMENT SCOPE 12

3 REFERENCES 13

3.1 CODES AND STANDARDS 13

3.2 COMPANY SPECIFICATIONS 13

3.3 PROJECT DOCUMENTS 14

3.4 SOFTWARE 14

4 DEFINITIONS AND ABBREVIATIONS 15

5 SUMMARY AND CONCLUSIONS 17

6 BLPH TOPSIDE DESCRIPTION 18

7 MATERIALS 19

7.1 STRUCTURAL STEEL – LINEAR PROPERTIES 19

7.2 STRUCTURAL STEEL – NON-LINEAR PROPERTIES 20

8 DESIGN CRITERIA AND REQUIREMENTS 22

8.1 GENERAL 22

8.2 PARTIAL RESISTANCE FACTORS 23

8.2.1 Partial Resistance Factors for Tubular Sections: 23

8.2.2 Partial Resistance Factors for Non-tubular Sections 23

8.3 ACCEPTANCE CRITERIA 24

9 FE JOINT STRUCTURAL ANALYSIS 25

9.1 METHODOLOGY 27

9.1.1 ANSYS FE Model 27

9.1.2 Boundary Conditions 29

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 29
9.2 BASIC LOAD CASES AND LOAD COMBINATIONS 29
10 TRUNNION JOINT 30
10.1 FE MODEL 31
10.1.1 Reference System 32
10.1.2 Material 32
10.1.3 Model Mesh 34
10.1.4 Constraints 36
10.1.5 Loads 36
10.2 FEM ANALYSIS AND RESULTS 40
10.2.1 FEA Results 40
10.2.2 Local checks for out-of-plane force 46
11 TP-TOPSIDE CONNECTION JOINTS 49
11.1 FE MODEL 50
11.1.1 Reference System 51
11.1.2 Material 52
11.1.3 Model Mesh 53
11.1.4 Constraints 56
11.1.5 Loads 56
11.2 FEM ANALYSIS AND RESULTS 58
11.2.1 In-Service Condition 59
11.2.2 Transportation Condition 63
12 TP-TOPSIDE CONNECTION JOINT TU03 65
12.1 FE MODEL 66
12.1.1 Reference System 67
12.1.2 Material 68
12.1.3 Model Mesh 69
12.1.4 Constraints 72
12.1.5 Loads 73
12.2 FEM ANALYSIS AND RESULTS 75
12.2.1 In-Service Condition 75

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 9
Figure 1.2 - Wind Park Details 10
Figure 1.3 – BLPH Concept 10
Figure 6.1 – Base Load Power Hub Plot Plan 18
Figure 7.1 – Steel constitutive law according to DNV-RP-C208 21
Figure 9.1 – Topside structural model 25
Figure 9.2 – Analyzed critical joints 26
Figure 9.3 – Analyzed critical joints – SACS Model 26
Figure 9.4 – Mesh elements – SOLID186 28
Figure 9.5 – Mesh elements – SOLID187 28
Figure 9.6 – Mesh elements – SHELL181 29
Figure 10.1 – Trunnion Joint – 3D View 30
Figure 10.2 – Trunnion Joint FE Model 31
Figure 10.3 – Trunnion Joint FE Model – Reference system 32
Figure 10.4 – Steel constitutive law according to DNV-RP-C208 33
Figure 10.5 – Trunnion Joint FE Model mesh – 3D isometric view 34
Figure 10.6 – Trunnion Joint FE Model mesh – joint detail view 34
Figure 10.7 – Trunnion Joint FE Model mesh – trunnion detail view 35
Figure 10.8 – Trunnion Joint FE Model mesh – trunnion section view 35
Figure 10.9 – Trunnion Joint FE Model – external constraints 36
Figure 10.10 – Grommet distribution forces round trunnion 38
Figure 10.11 – Trunnion Joint FE model – applied loads 39
Figures 10.12 – Trunnion Joint FE model – load application area and point 40
Figure 10.13 – Trunnion Joint FE model – VM Stresses 41
Figure 10.14 – Trunnion Joint FE model – VM Stresses > 340 MPa 41
Figure 10.15 – Trunnion Joint FE model – VM Stresses > 400 Mpa 42
Figure 10.16 – Trunnion Joint FE model – Peak VM Stresses at hot spot areas 43
Figure 10.17 – Trunnion Joint FE model – Plastic strains 44
Figure 10.18 – Trunnion Joint FE model – Total strains 45
Figure 10.19 – Out-of-plane forces – local checks 46

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

Figure 10.20 – Stopper plate resisting cross section 47
Figure 10.21 – Internal Ring resisting cross section 48
Figure 11.1 – Joint TU02 – 3D view 49
Figure 11.2 – Joint TU02 FE Model 50
Figure 11.3 – Joint TU02 FE Model – Reference system 51
Figure 11.4 – Joint TU02 FE Model –Members Local Reference systems 51
Figure 11.5 – Steel constitutive law according to DNV-RP-C208 53
Figure 11.6 – Joint TU02 FE Model mesh – 3D isometric view 54
Figure 11.7 – Joint TU02 FE Model mesh – back detail view 54
Figure 11.8 – Joint TU02 FE Model mesh – front detail view 55
Figure 11.9 – Joint TU02 FE Model mesh – bottom detail view 55
Figure 11.10 – Joint TU02 FE Model – external constraints 56
Figure 11.11 – Joint TU02 FE Model – applied loads – In-Service condition 57
Figure 11.12 – Joint TU02 FE Model – applied loads – Transportation condition 58
Figure 11.13 – Joint TU02 FE model – VM Stresses (front view) 59
Figure 11.14 – Joint TU02 FE model – VM Stresses (back view) 59
Figure 11.15 – Joint TU02 FE model – VM Stresses > 340 MPa 60
Figure 11.16 – Joint TU02 FE model – VM Stresses > 400 MPa 60
Figure 11.17 – Joint TU02 FE model – Peak VM stresses at hot spot area BOTTOM 61
Figure 11.18 – Joint TU02 FE model – Peak VM stresses at hot spot area TOP 61
Figure 11.19 – Joint TU02 FE model – Plastic strains 62
Figure 11.20 – Joint TU02 FE model – Total strains 62
Figure 11.21 – Joint TU02 FE model – VM Stresses (front view) 63
Figure 11.22 – Joint TU02 FE model – VM Stresses (back view) 63
Figure 12.1 – Joint TU02 – 3D view 65
Figure 12.2 – Joint TU03 FE Model 66
Figure 12.3 – Joint TU03 FE Model – Reference system 67
Figure 12.4 – Joint TU03 FE Model –Members Local Reference systems 67
Figure 12.5 – Steel constitutive law according to DNV-RP-C208 69
Figure 12.6 – Joint TU03 FE Model mesh – 3D isometric view 70
Figure 12.7 – Joint TU03 FE Model mesh – back detail view 70
Figure 12.8 – Joint TU03 FE Model mesh – front detail view 71

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

Figure 12.9 – Joint TU03 FE Model mesh – bottom detail view 71
Figure 12.10 – Joint TU02 FE Model – external constraints 72
Figure 12.11 – Joint TU03 FE Model – applied loads – In-Service condition 74
Figure 12.12 – Joint TU03 FE model – VM Stresses (front view) 75
Figure 12.13 – Joint TU03 FE model – VM Stresses (back view) 76
Figure 12.14 – Joint TU03 FE model – VM Stresses > 340 MPa 76
Figure 12.15 – Joint TU03 FE model – VM Stresses > 400 MPa 77
Figure 12.16 – Joint TU03 FE model – Peak VM stresses at hot spot area BOTTOM 78
Figure 12.17 – Joint TU03 FE model – Peak VM stresses at hot spot area TOP 78
Figure 12.18 – Joint TU03 FE model – Peak VM stresses at hot spot area internal 78
Figure 12.19 – Joint TU03 FE model – Peak VM stresses at IPE330 edge 79
Figure 12.20 – Joint TU03 FE model – Plastic strains 80
Figure 12.21 – Joint TU03 FE model – Total strains 81
Figure 12.22 – Joint TU03 FE model – Total strains on upper joint (TOPSIDE part) 81
Figure 12.23 – Joint TU03 FE model – Total strains > 1% 82

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 tables

Table 4.1 – Definitions and Abbreviations 16
Table 7.1 – Material Selection, Ref /21/ 19
Table 7.2 – Mechanical properties of Plates 20
Table 7.3 – Mechanical properties of Structural Seamless Tubulars 20
Table 7.4 – Mechanical properties for Structural Sections 20
Table 7.5 – S420 steel constitutive law parameters 21
Table 8.1- Resistance Factors for Tubular Sections 23
Table 8.2- Resistance for Non Tubular Section 24
Table 10.1 – S420 steel constitutive law parameters 33
Table 10.2 – Trunnion Joint FE model – mesh statistics 34
Table 10.3 – BLPH Platform – Lifting reactions at lifting points 37
Table 10.4 – Trunnion Joint FE model – applied loads 38
Table 11.1 – S420 steel constitutive law parameters 52
Table 11.2 – Joint TU02-TU04 FE model – mesh statistics 53
Table 11.3 – Member loads – In-Service condition 56
Table 11.4 – Member loads – Transportation condition 57
Table 11.5 – Joint TU02 FE Model – applied loads 57
Table 12.1 – S420 steel constitutive law parameters 68
Table 12.2 – Joint TU03 FE model – mesh statistics 69
Table 12.3 – Member TA44-TU03 loads – In-Service condition 73
Table 12.4 – Joint TU03 FE Model – applied loads 73

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.

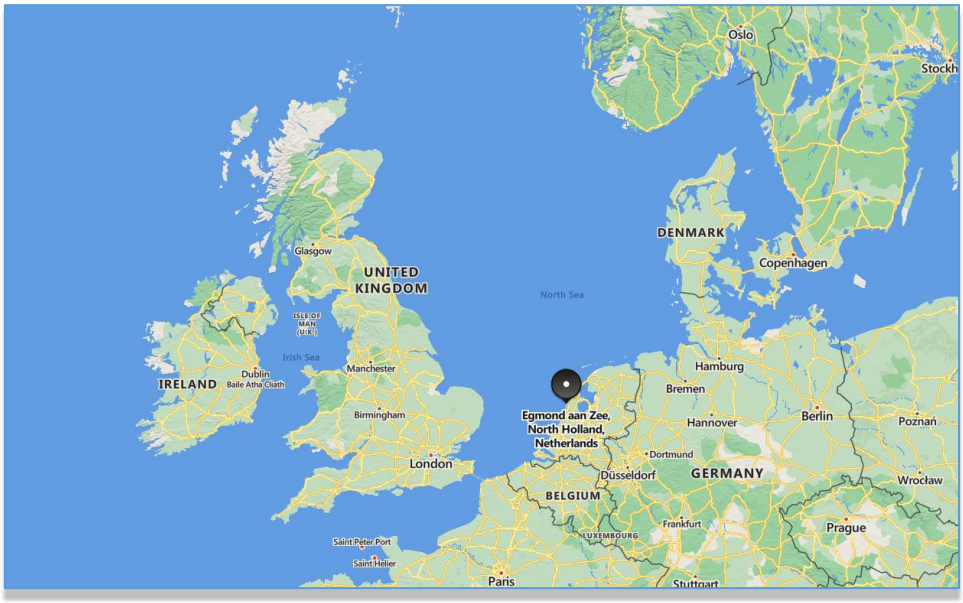


Figure 1.1 - HOLLANDSE KUST NOORD WIND PARK Location Map

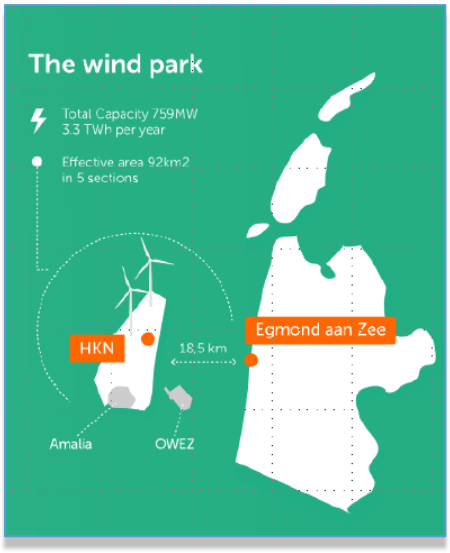


Figure 1.2 - Wind Park Details

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.

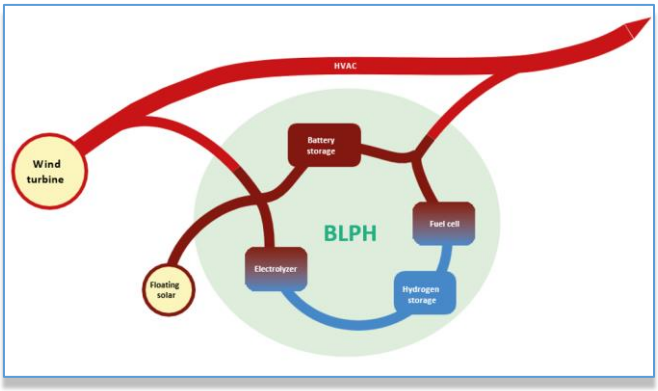


Figure 1.3 – BLPH Concept

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 Topside structure for in-service, lifting and transportation conditions.

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. /35/, and the Finite Element Analysis software ANSYS, developed by ANSYS Inc., Ref./36/.

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/ UNI EN 1993-1-1	Design of steel structures - Part 1-1: General rules and rules for buildings
/9/ UNI 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
/22/ 123A90-DDL-ST-RE-90000	BLPH Platform – In service Analysis
/23/ 123A90-DDL-ST-RE-90002	BLPH Platform – Transportation analysis Report
/24/ 123A90-DDL-ST-RE-90004	BLPH Platform – Lifting Analysis
/25/ 123A90-DDL-ST-RE-90007	BLPH Platform – FE Analysis
/26/ 123A90-DDL-ST-TN-90119	Evaluating ISO and DNV approaches
/27/ 123A90-DDL-ST-DD-90042	BLPH Platform – Primary Steel Elevation at Row -A-
/28/ 123A90-DDL-ST-DD-90048	BLPH Platform – Primary Steel Elevation at Row -2&4-
/29/ 123A90-DDL-ST-DD-90049	BLPH Platform – Primary Steel Plan at El. +23310 T.O.S.
/30/ 123A90-DDL-ST-DD-90050	BLPH Platform – Primary Steel Plan at El. +30310 T.O.S.
/31/ 123A90-DDL-ST-DD-90051	BLPH Platform – Primary Joints Lower Deck – Joint 1 Detail
/32/ 123A90-DDL-ST-DD-90052	BLPH Platform – Primary Joints Upper Deck – Joint 10 Detail
/33/ 123A90-DDL-ST-DD-90055	BLPH Platform – Lifting Points General Arrangement
/34/ 123A90-DDL-ST-DD-90056	BLPH Platform – Lifting Points Details

3.4 SOFTWARE

Doc. No.	Description
/35/ SACS	Structural Analysis Computer System – BENTLEY Inc- v15
/36/ ANSYS	Ansys Inc.

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

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

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

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 Topside joints, as configured, can withstand the maximum loads that they experience for In-Service, Transportation and Lifting conditions. Checks are executed according to the criteria imposed by ISO code 19902, Ref. /6/ and Eurocode 3, Ref./8/.

The most critical joints of the Topside were selected and analysed for the worst conditions they are subjected: the trunnion joint used for lifting and the joints connecting the topside to the TP for the in-service and transportation case. Analysed joints are:

- Trunnion Joint (Sacs Joint PEA2): Lifting Conditions
- TP-Topside connection joint (Sacs jointsTU01, TU02, TU04): In-Service and Transportation Conditions
- Crane Joint (Sacs jointsTU03): In-Service Conditions (Crane loads)

The results of the Finite Element Analyses are summarized in the following:

1) The Trunnion Joint is checked against the loads imposed by the lifting condition, which is the most critical one. 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 trunnion connection with the vertical column. Elasto-plastic strains have been investigated, and they result to be lower than the admissible value 5% (fatigue non-sensitive joint). According to these results, the Trunnion joint is correctly designed to withstand the loads imposed by the lifting condition.
2) Joint TU02 (TP-Topside connection joint) is checked against the loads imposed by the In-Service and Transportation conditions, which are the most critical ones. The obtained results are valid for the joints TU01, TU02 and TU04, which share the same geometry, since the model was checked for the max. envelope of the loads acting on the three joints. The FE analysis has shown that globally Von-Mises stresses are largely lower than the allowable stresses, except for local peaks occurring in the vertical column connecting the joint to the TP, particularly at the intersection with the TP upper 65mm plate and joint bottom 40mm flange. Elasto-plastic strains have been investigated, and they result to be lower than the admissible value 1% (fatigue sensitive joint). According to these results, the TU02 joint (as well as TU01 and TU04) is correctly designed to withstand the loads imposed by the In-Service and Transportation conditions.
3) Joint TU03 (TP-Topside connection joint with crane connection) is checked against the loads imposed by the In-Service conditions, which are the most critical ones when the crane is operating. The FE analysis has shown that globally Von-Mises stresses are largely lower than the allowable stresses, except for local peaks occurring in the vertical column connecting the joint to the TP, particularly at the intersection with the TP upper 65mm plate and joint bottom 40mm flange. Elasto-plastic strains have been investigated, and they result to be lower than the admissible values 1% (fatigue sensitive joint). According to these results, the TU03 joint is correctly designed to withstand the loads imposed by the In-Service conditions.

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

6 BLPH TOPSIDE DESCRIPTION

The BLPH Topside comprises of two main deck levels with overall deck plan dimensions of 40 m x 16 m, excluding external walkways, laydown areas and stairs, with an overall deck height of 7 m between decks. There are currently 6 hydrogen storage vessels which lie horizontally, aligned north – south, within the PAU structure located above the upper deck. The Topsides layout is shown in Figure 6.1.

Main Deck elevations are:

- Lower Deck, El. + 23.310 m ToS;
- Upper Deck, El. + 30.310 m ToS.

The PAU steel module 12m x 9m for the storage of N.6 H2 Vessel 60t each is located on top of Topside's Upper Deck. PAU height is 9m and on top roof it acts as rigging platform with lifting slings.

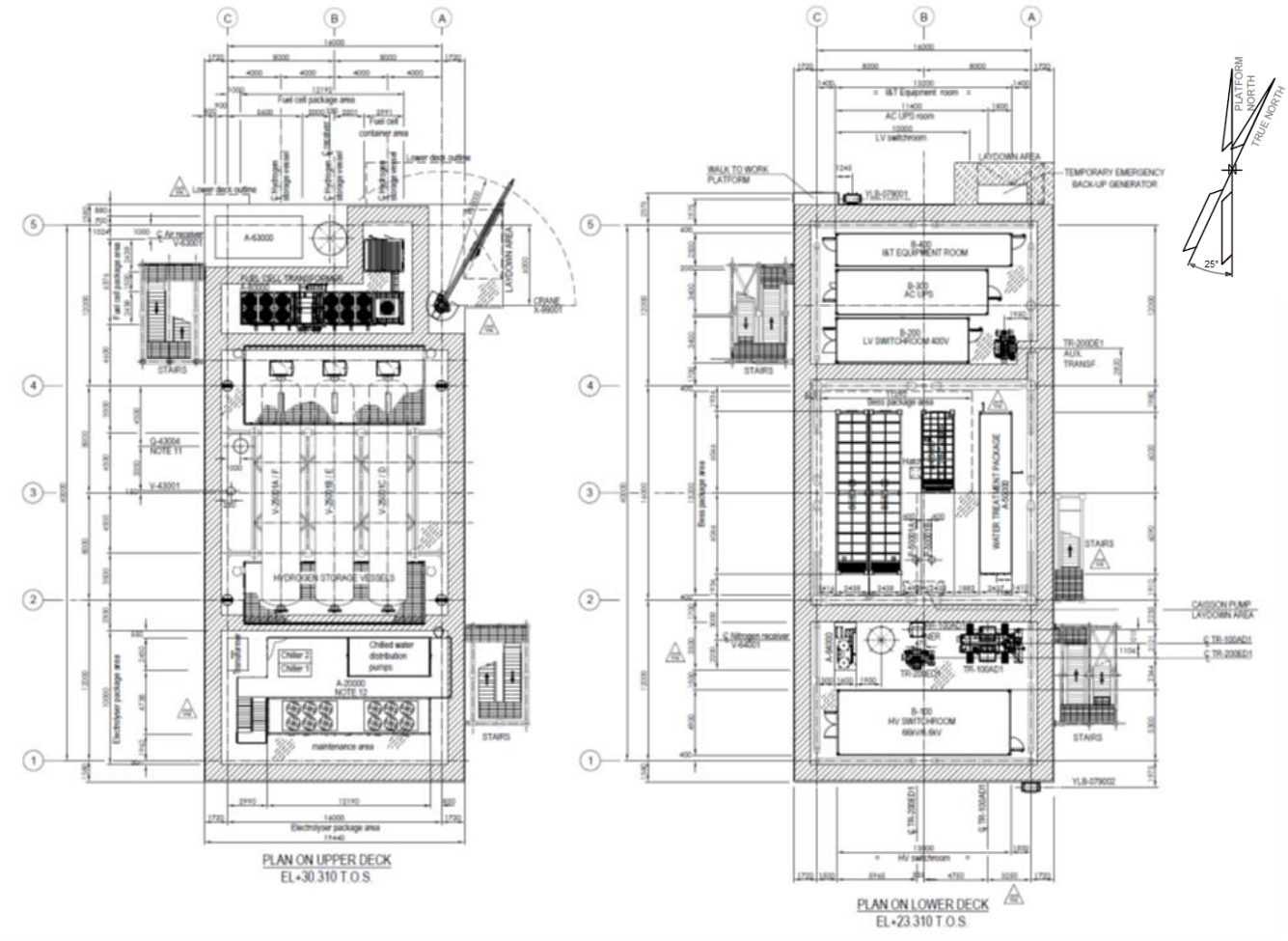


Figure 6.1 – Base Load Power Hub Plot Plan

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

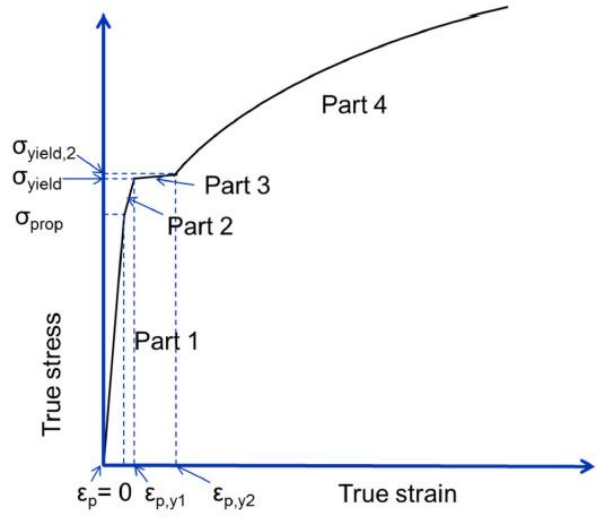


Figure 7.1 – Steel constitutive law according to DNV-RP-C208

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:

- $\varepsilon_t = \leq \varepsilon_{adm} = 5\%$ for fatigue non-sensitive joints
- $\varepsilon_t = \leq \varepsilon_{adm} = 1\%$ for fatigue sensitive joints

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 JOINT STRUCTURAL ANALYSIS

The software SACS™, ref./35/, is used for all global analyses of the structures in the SoW. The model of the platform is displayed in Figure 9.1. This model is used for the In-Service, Transportation and Lifting analyses, as per relative reports (Ref./22/, /23/, /24/).

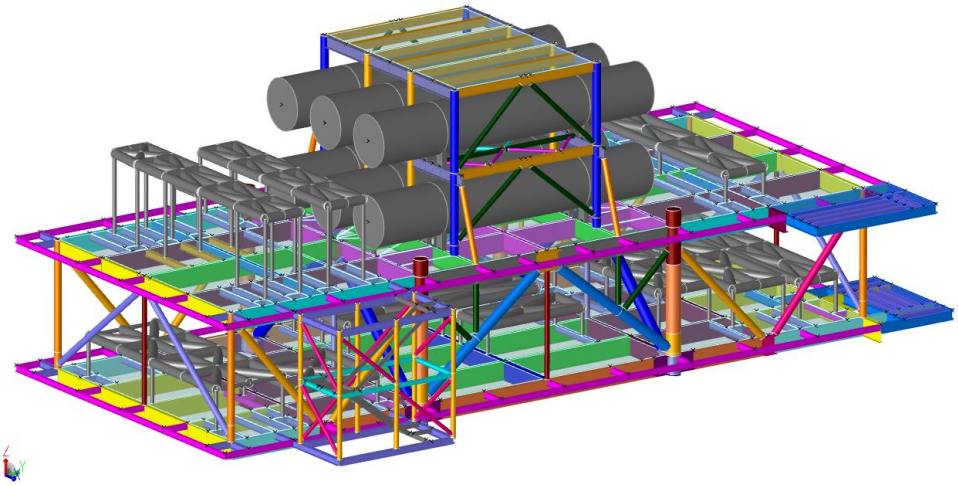


Figure 9.1 – Topside structural model

The local analysis of the most critical joints, using FE approach, is carried out using Ansys Environment Ref./36/, creating a local model of the joint according to structural drawings (Ref./27/,/28/,/29/,/30/,/31/,/32/,/33/,/34/).

The local models of the selected joints have been meshed using 2D or 3D elements, following a Finite Element approach. Every node 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.

Von Mises stresses in the Joint are extracted for the most critical load combinations for In-Service, Transport and Lifting phases.

The joints that were analysed are the most representative joints for the considered module, being the most stressed and complex.

The analysed most representative joints are the following:

- Trunnion Joint (Sacs Joint PEA2): Joints used for lifting the topside structure
- TP-Topside Connection Joint (SACS Joints TU01, TU02, TU04): Joints where Topside is connected to TP

- TP-Topside Connection Joint (SACS Joint TU03): Joint where Topside structure is connected to TP and to Crane

These joints are highlighted in the following figures.

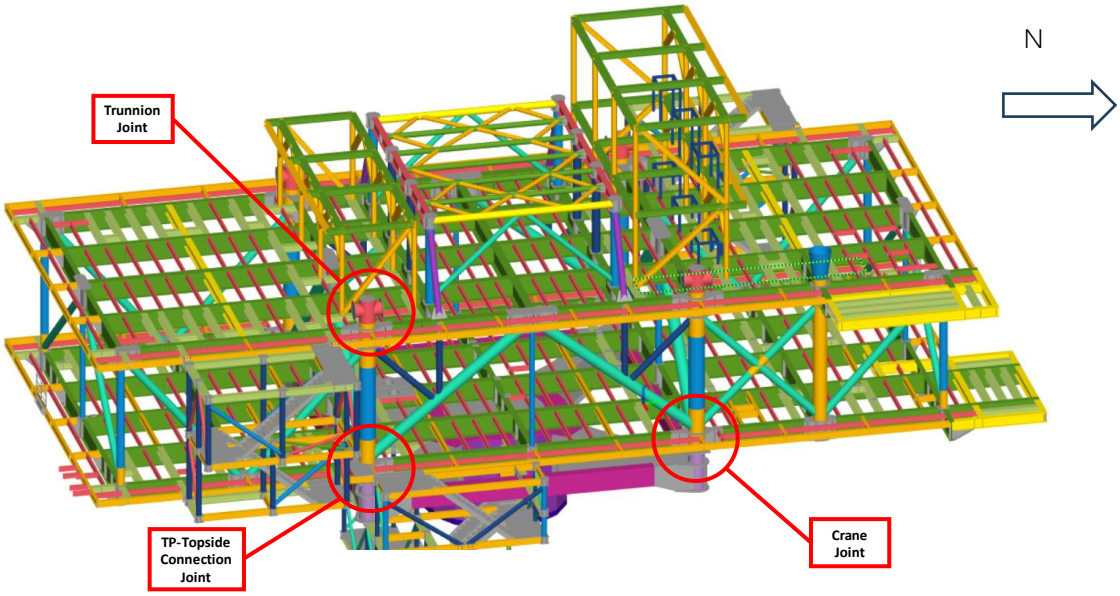


Figure 9.2 – Analyzed critical joints

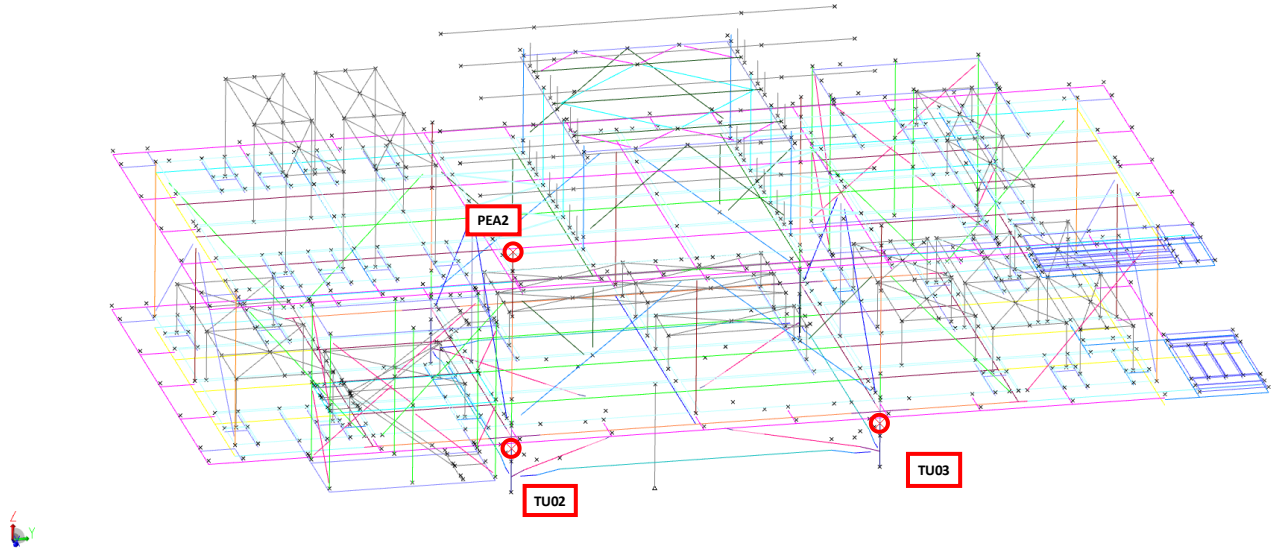


Figure 9.3 – Analyzed critical joints – SACS Model

Only joints TU02 and TU03 are fatigue sensitive joints for transportation, since the topside is sustained at the four joints TU01-TU0-TU03-TU04 over the lower grillage beams. There is no fatigue check for in-service conditions.

9.1 METHODOLOGY

The most critical load conditions from In-Service, Transportation and Lifting analyses 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.

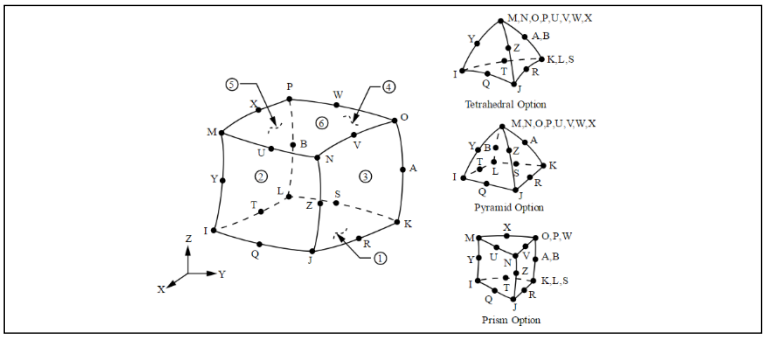


Figure 9.4 – Mesh elements – SOLID186

- 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.

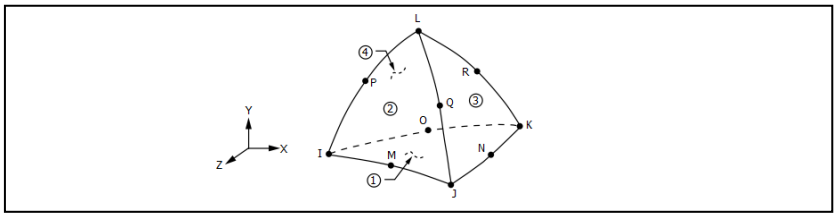


Figure 9.5 – Mesh elements – SOLID187

- 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.

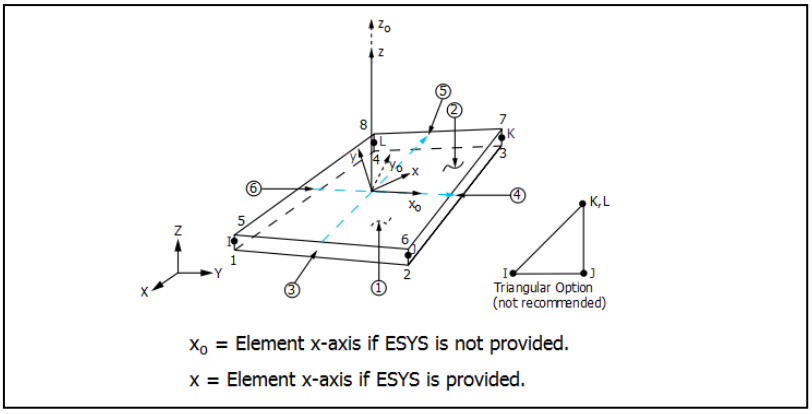


Figure 9.6 – Mesh elements – SHELL181

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.

9.2 BASIC LOAD CASES AND LOAD COMBINATIONS

The general basic load cases of BLPH Platform are defined in the In-Service, Transportation and Lifting analysis reports Ref./22/, Ref./23/ and Ref./24/.

The same sets of loads and combinations are maintained. Joints local stress states are computed for the most critical load combinations for the In-Service, Transportation and Lifting analyses, as per relative reports (Ref./22/, /23/, /24/).

Refer to the aforementioned reports for the definition of basic load cases and load combinations of the models for each analysis, as well as for the results about the considered joint in order to select the most critical load combination.

10 TRUNNION JOINT

This section presents the local analysis and results for the Trunnion Joint. Sacs Joint PEA2 (Joint A2 at El. +30.31m) has been modelled and analysed for the Lifting condition, being the most stressed one. In the following sections are reported:

- Description of the FE model of Trunnion joint
- Structural analysis results and checks

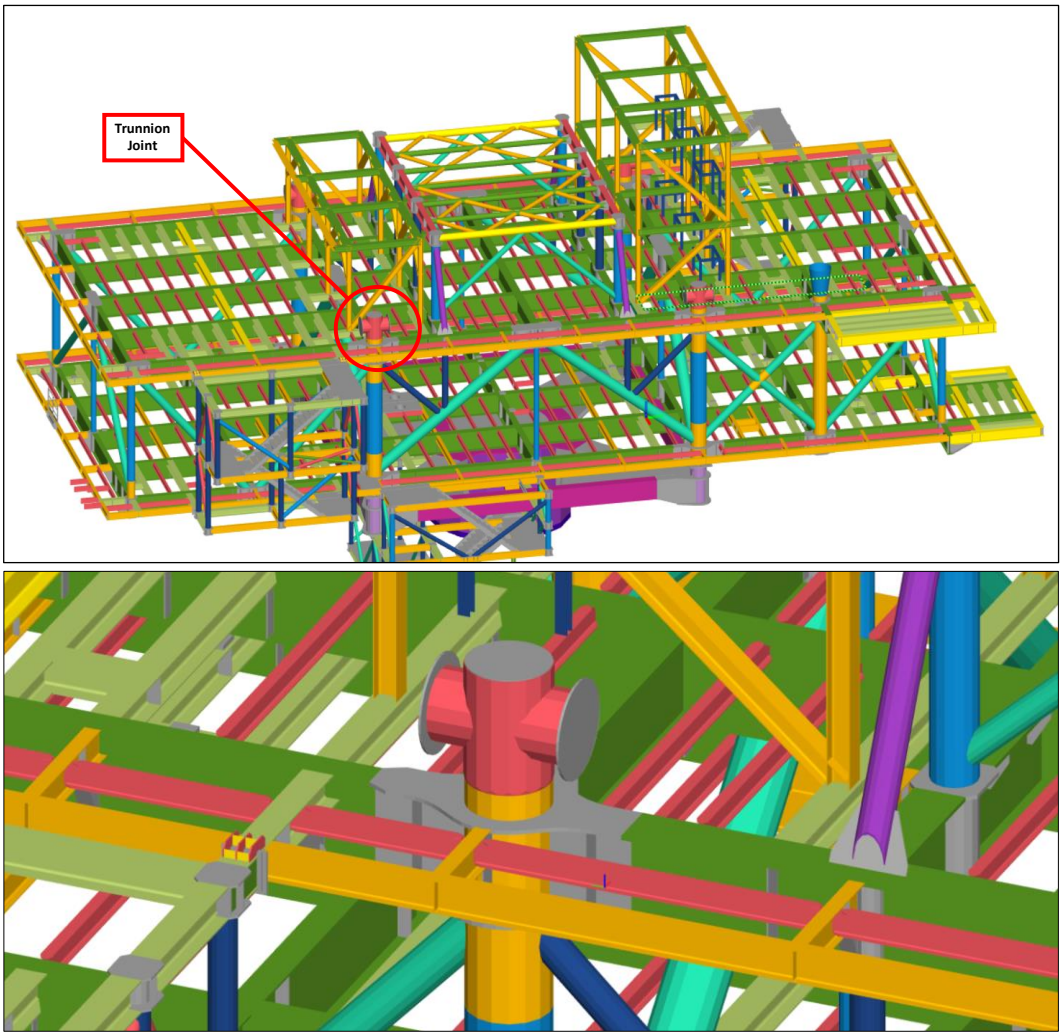


Figure 10.1 – Trunnion Joint – 3D View

This joint is not fatigue sensitive.

10.1 FE MODEL

The Trunnion Joint is analysed for the Lifting condition. It is a critical joint since the Topside structure is lifted at the 4 trunnion joints. The most stressed trunnion joint, that is SACS joint PEA2, is modelled in ANSYS environment; it is represented in Figure 10.2.

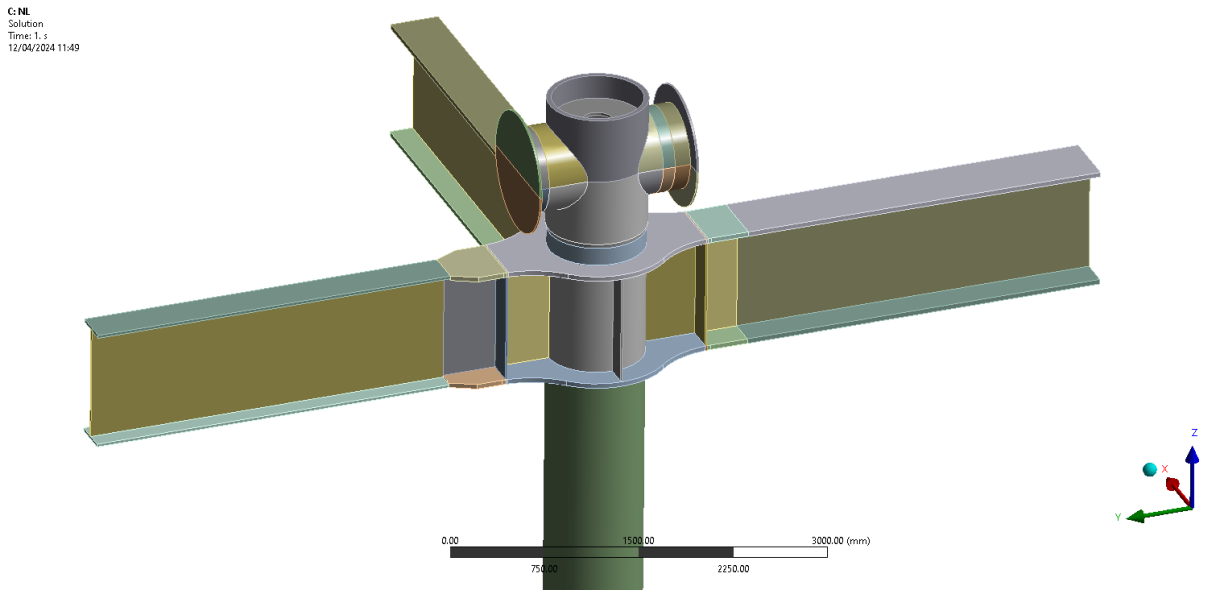


Figure 10.2 – Trunnion Joint FE Model

It is a Φ820x40 mm pipe connected to the top joint of the topside, made by the crossing of two longitudinal beams WPG1000x500x20x40 mm and WPG1000x300x20x25 with a transversal beam WPG1000x400x15x40, sustained by a passing tubular column Φ800x30.

The joint is reinforced with top and bottom 40mm plates and 15mm stiffeners. The trunnion pipe has two orthogonal pipes Φ700x35, closed by 25mm round plates, where grommets are anchored for lifting.

The joint is made by S420 steel.

The joint is modelled with the correct geometry and thickness of all the elements as per structural drawings Ref./32/, /33/, /34/.

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

10.1.1 Reference System

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

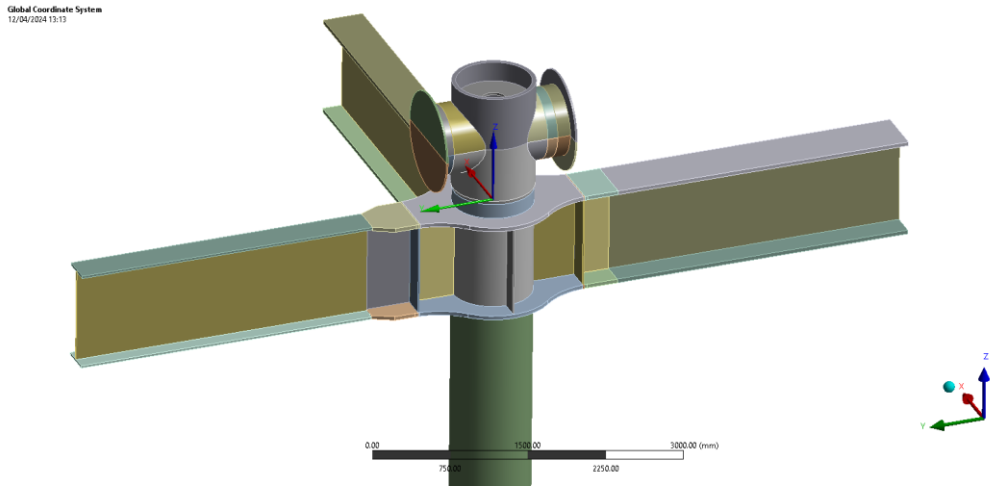


Figure 10.3 – Trunnion Joint FE Model – Reference system

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 $12 \cdot 10^{-6}$ / °C

The assumed constitutive law for the material 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 Table 10.1. The curve associated to the previous equation is represented in Figure 10.4.

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

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 10.1 – S420 steel constitutive law parameters

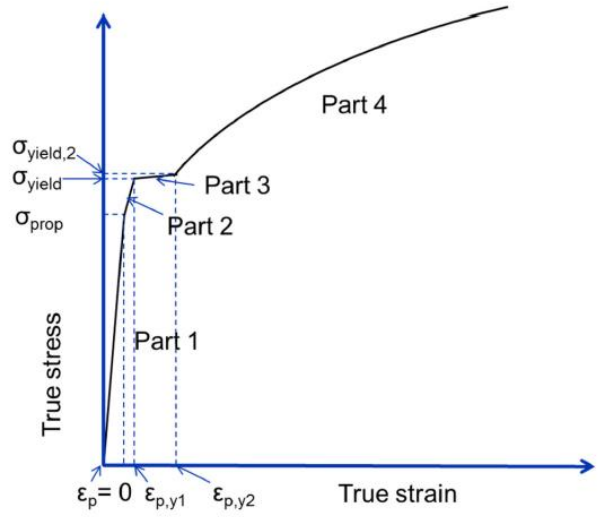


Figure 10.4 – Steel constitutive law according to DNV-RP-C208

10.1.3 Model Mesh

The FE model is meshed using hexahedra and tetrahedra elements, as reported in 9.1.1.

Mesh is refined in the trunnion tube and in the joint. Mesh statistics are reported in Table 10.2.

Mesh Part	Number
Nodes	1309673
Elements	678575

Table 10.2 – Trunnion Joint FE model – mesh statistics

In the following figures, FE model mesh for trunnion join is presented.

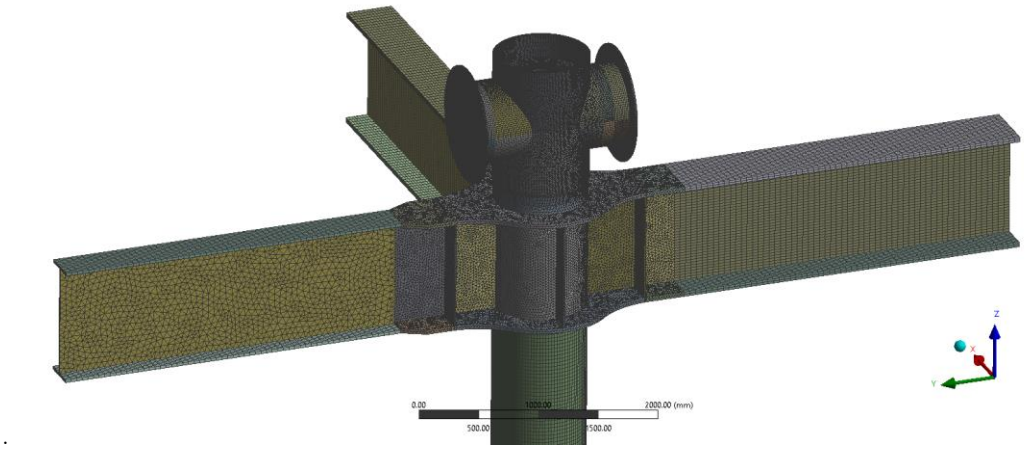


Figure 10.5 – Trunnion Joint FE Model mesh – 3D isometric view

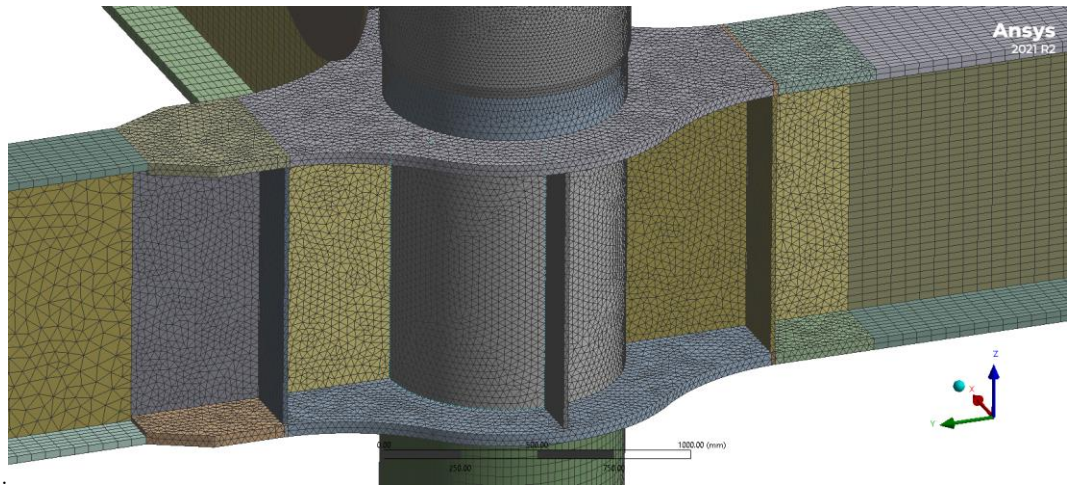


Figure 10.6 – Trunnion Joint FE Model mesh – joint detail view

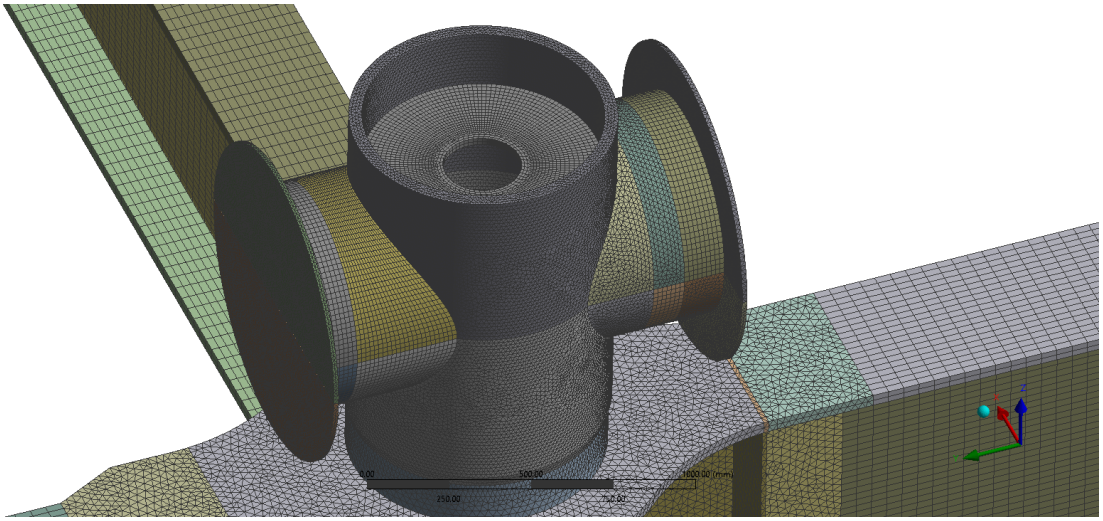


Figure 10.7 – Trunnion Joint FE Model mesh – trunnion detail view

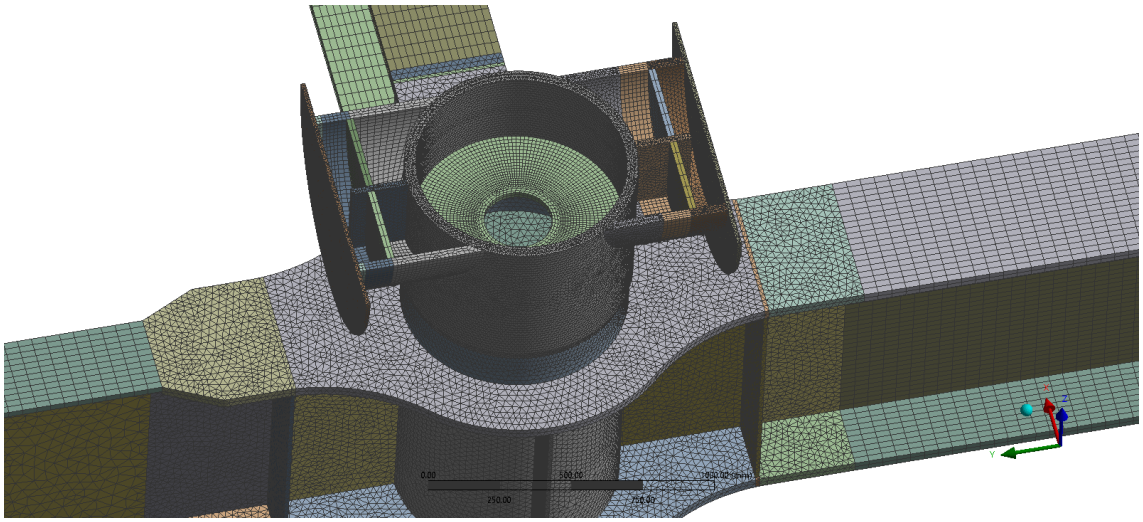


Figure 10.8 – Trunnion Joint FE Model mesh – trunnion section view

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.

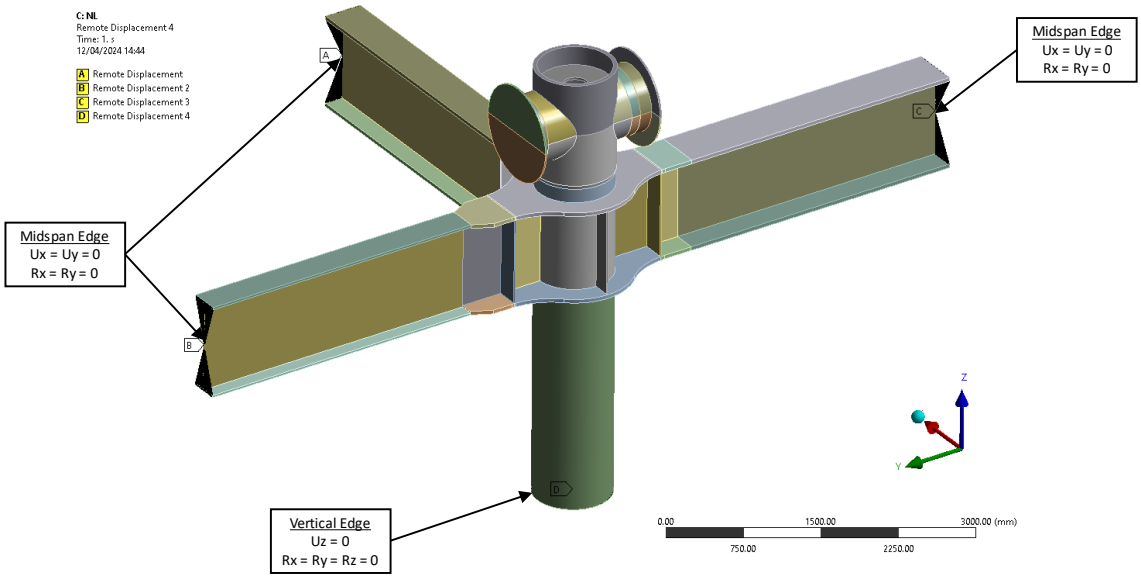


Figure 10.9 – Trunnion Joint FE Model – external constraints

10.1.5 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./24/).

JOINTS REACTION FORCES AND MOMENTS REPORT

JOINT	LOAD COND	FORCE(X) kN	FORCE(Y) kN	FORCE(Z) kN	MOMENT(X) kN-m	MOMENT(Y) kN-m	MOMENT(Z) kN-m
PEA2	LA20	0.000	0.000	5754.750	0.000	0.000	0.000
	LA21	0.000	0.000	12577.497	0.000	0.000	0.000
	LA22	0.000	0.000	11126.203	0.000	0.000	0.000
	LA23	0.000	0.000	9674.888	0.000	0.000	0.000
	LA40	0.000	0.000	4568.286	0.000	0.000	0.000
	LA41	0.000	0.000	9984.364	0.000	0.000	0.000
	LA42	0.000	0.000	8832.293	0.000	0.000	0.000
	LA43	0.000	0.000	7680.199	0.000	0.000	0.000
	LC20	0.000	0.000	5282.048	0.000	0.000	0.000
	LC21	0.000	0.000	11544.364	0.000	0.000	0.000
	LC22	0.000	0.000	10212.293	0.000	0.000	0.000
	LC23	0.000	0.000	8880.195	0.000	0.000	0.000
	LC40	0.000	0.000	4095.585	0.000	0.000	0.000
	LC41	0.000	0.000	8951.243	0.000	0.000	0.000
	LC42	0.000	0.000	7918.378	0.000	0.000	0.000
	LC43	0.000	0.000	6885.494	0.000	0.000	0.000
	LIF0	0.000	0.000	4935.393	0.000	0.000	0.000
	LIF1	0.000	0.000	10786.710	0.000	0.000	0.000
	LIF2	0.000	0.000	9542.058	0.000	0.000	0.000
	LIF3	0.000	0.000	8297.379	0.000	0.000	0.000
PEA4	LA20	0.000	0.000	4191.173	0.000	0.000	0.000
	LA21	0.000	0.000	9160.240	0.000	0.000	0.000
	LA22	0.000	0.000	8103.263	0.000	0.000	0.000
	LA23	0.000	0.000	7046.213	0.000	0.000	0.000
	LA40	0.000	0.000	5377.636	0.000	0.000	0.000

	LA41	0.000	0.000	11753.367	0.000	0.000	0.000
	LA42	0.000	0.000	10397.171	0.000	0.000	0.000
	LA43	0.000	0.000	9040.902	0.000	0.000	0.000
	LC20	0.000	0.000	3715.874	0.000	0.000	0.000
	LC21	0.000	0.000	8121.421	0.000	0.000	0.000
	LC22	0.000	0.000	7184.307	0.000	0.000	0.000
	LC23	0.000	0.000	6247.123	0.000	0.000	0.000
	LC40	0.000	0.000	4902.337	0.000	0.000	0.000
	LC41	0.000	0.000	10714.547	0.000	0.000	0.000
	LC42	0.000	0.000	9478.224	0.000	0.000	0.000
	LC43	0.000	0.000	8241.824	0.000	0.000	0.000
	LIF0	0.000	0.000	4720.529	0.000	0.000	0.000
	LIF1	0.000	0.000	10317.200	0.000	0.000	0.000
	LIF2	0.000	0.000	9126.717	0.000	0.000	0.000
	LIF3	0.000	0.000	7936.173	0.000	0.000	0.000
PEC2	LA20	0.000	0.000	5040.011	0.000	0.000	0.000
	LA21	0.000	0.000	11015.381	0.000	0.000	0.000
	LA22	0.000	0.000	9744.339	0.000	0.000	0.000
	LA23	0.000	0.000	8473.278	0.000	0.000	0.000
	LA40	0.000	0.000	3861.473	0.000	0.000	0.000
	LA41	0.000	0.000	8439.577	0.000	0.000	0.000
	LA42	0.000	0.000	7465.753	0.000	0.000	0.000
	LA43	0.000	0.000	6491.915	0.000	0.000	0.000
	LC20	0.000	0.000	5512.713	0.000	0.000	0.000
	LC21	0.000	0.000	12048.536	0.000	0.000	0.000
	LC22	0.000	0.000	10658.297	0.000	0.000	0.000
	LC23	0.000	0.000	9268.037	0.000	0.000	0.000
	LC40	0.000	0.000	4334.175	0.000	0.000	0.000
	LC41	0.000	0.000	9472.721	0.000	0.000	0.000
	LC42	0.000	0.000	8379.684	0.000	0.000	0.000
	LC43	0.000	0.000	7286.623	0.000	0.000	0.000
	LIF0	0.000	0.000	4514.367	0.000	0.000	0.000
	LIF1	0.000	0.000	9866.539	0.000	0.000	0.000
	LIF2	0.000	0.000	8728.061	0.000	0.000	0.000
	LIF3	0.000	0.000	7589.561	0.000	0.000	0.000
PEC4	LA20	0.000	0.000	3959.325	0.000	0.000	0.000
	LA21	0.000	0.000	8653.507	0.000	0.000	0.000
	LA22	0.000	0.000	7655.009	0.000	0.000	0.000
	LA23	0.000	0.000	6656.431	0.000	0.000	0.000
	LA40	0.000	0.000	5137.862	0.000	0.000	0.000
	LA41	0.000	0.000	11229.315	0.000	0.000	0.000
	LA42	0.000	0.000	9933.595	0.000	0.000	0.000
	LA43	0.000	0.000	8637.793	0.000	0.000	0.000
	LC20	0.000	0.000	4434.624	0.000	0.000	0.000
	LC21	0.000	0.000	9692.320	0.000	0.000	0.000
	LC22	0.000	0.000	8573.939	0.000	0.000	0.000
	LC23	0.000	0.000	7455.490	0.000	0.000	0.000
	LC40	0.000	0.000	5613.161	0.000	0.000	0.000
	LC41	0.000	0.000	12268.096	0.000	0.000	0.000
	LC42	0.000	0.000	10852.533	0.000	0.000	0.000
	LC43	0.000	0.000	9436.864	0.000	0.000	0.000
	LIF0	0.000	0.000	4774.968	0.000	0.000	0.000
	LIF1	0.000	0.000	10436.177	0.000	0.000	0.000
	LIF2	0.000	0.000	9231.975	0.000	0.000	0.000
	LIF3	0.000	0.000	8027.695	0.000	0.000	0.000

Table 10.3 – BLPH Platform – Lifting reactions at lifting points

From the lifting analysis report (Ref./24/), the maximum ULS vertical force acting on SACS joint PEA2 is extracted: F = 1282 tons = 12577 kN. This load takes into account all combination coefficients for lifting condition, including the consequence factor CF = 1.30.

According to DNV-ST-N001 Ref./10/, for sling load distribution in the case of doubled grommet over the upper shackle, the total lifting load shall be divided into 55%-45% to the two members of the trunnion, so that two forces are applied at each edge:

F_1 = 0.45*12577 kN = 5659.65 kN

F_2 = 0.55*12577 kN = 6917.35 kN

Each grommet edge is bent over the trunnion, so another 55%-45% distribution shall be considered for the grommet frictional losses over the bend.

This results in applying a torsional moment at the two trunnion ends according to the scheme in Figure 10.10:

$M_1 = (0.55-0.45)*F_2*d = 0.05*d*F_1 = 198.10$ kN

$M_2 = (0.55-0.45)*F_1*d = 0.05*d*F_2 = 242.11$ kN

Being d = 700mm the trunnion diameter round which grommet is bent.

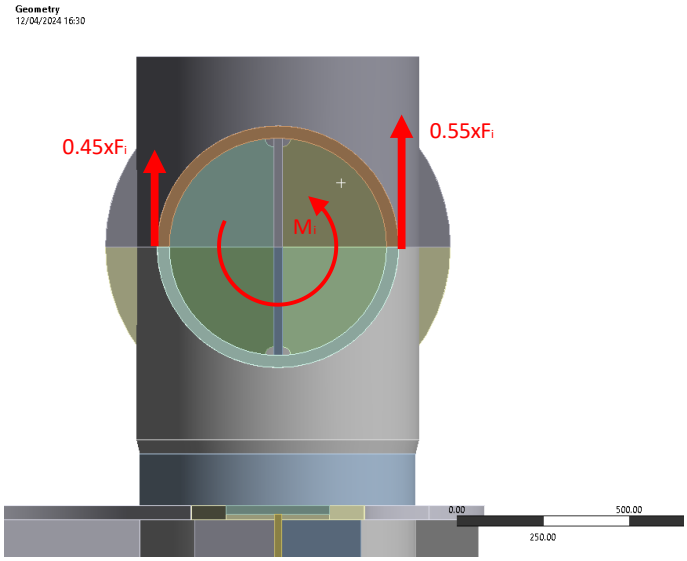


Figure 10.10 – Grommet distribution forces round trunnion

A 5% out-of-plane force is also included. A Resume of the applied forces is reported in Table 10.4.

Edge	Fx [kN]	Fy [kN]	Fz [kN]	My [kNm]
1	0	282.98	5659.65	198.10
2	0	345.87	6917.35	242.11

Table 10.4 – Trunnion Joint FE model – applied loads

These forces and moments are applied to the two ends of the trunnion as shown in Figure 10.11.

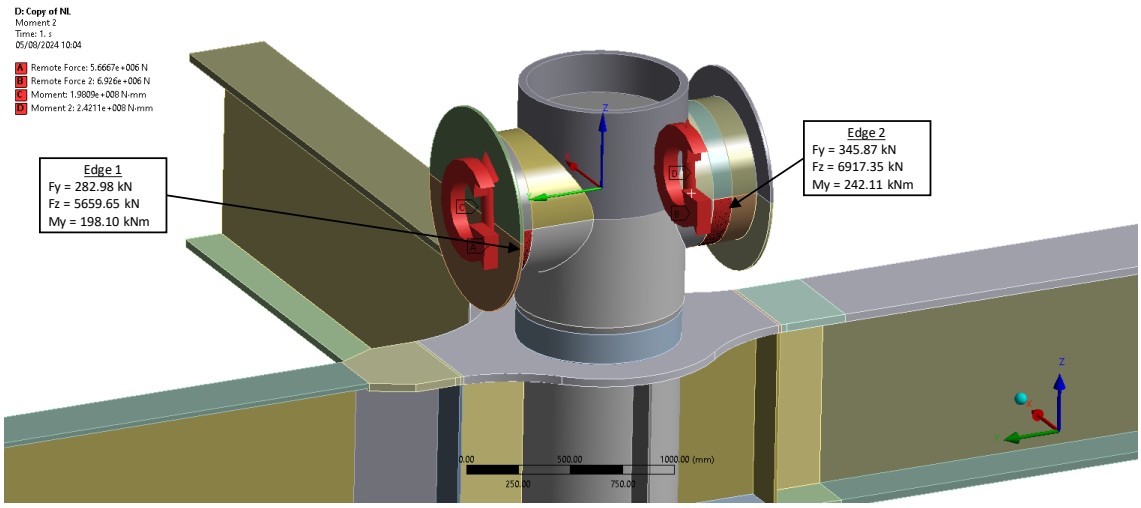
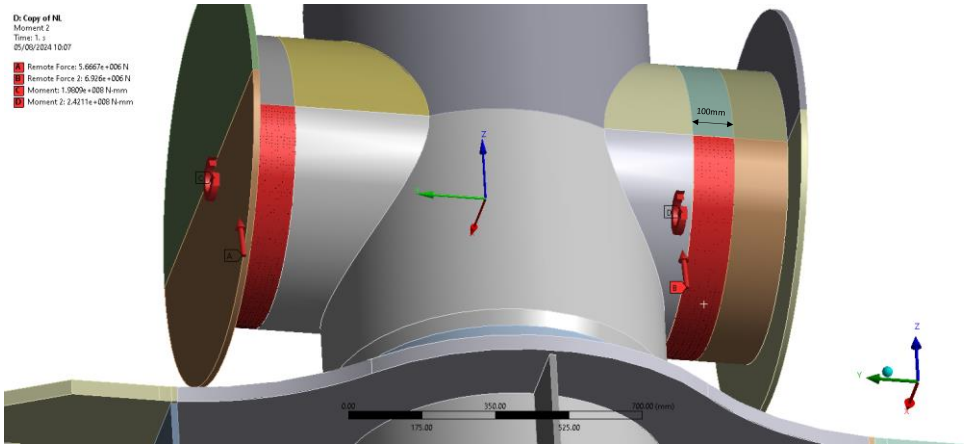


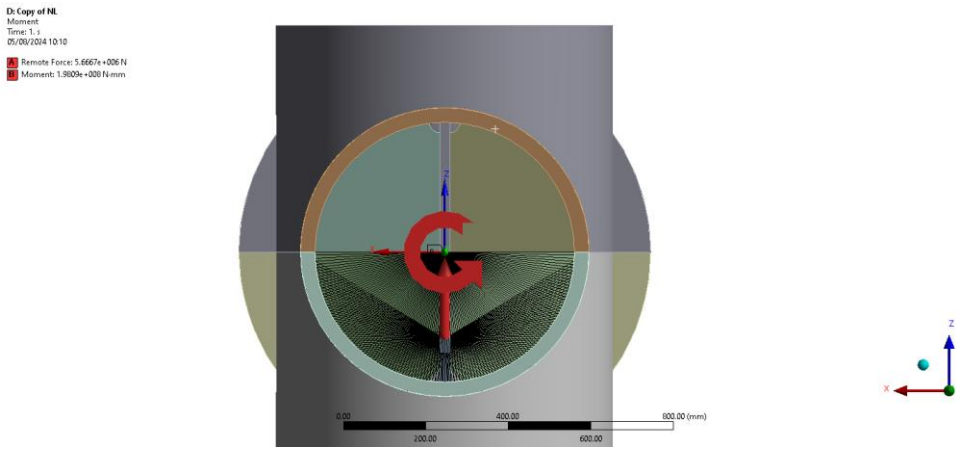
Figure 10.11 – Trunnion Joint FE model – applied loads

The loads are applied to a strip 100mm wide, which is the grommet minimum diameter to be used for lifting, and it takes the semicircular contact surface between the grommet and the trunnion. Loads are applied shifted to one extreme end of the trunnion to account for max. displacement.

The vertical load is applied to the surface at a remote point which is at 2/3R from the centre of the trunnion downwards (or 1/3R from the bottom vertex of the trunnion section) in order to simulate the sinusoidal load distribution of the load on the loaded strip (grommet force is more intense at the bottom of the trunnion, almost negligible at the centre points). The torsional moment is applied at the centre of the section.

Load application details are shown in Figures 10.12.





Figures 10.12 – Trunnion Joint FE model – load application area and point

10.2 FEM ANALYSIS AND RESULTS

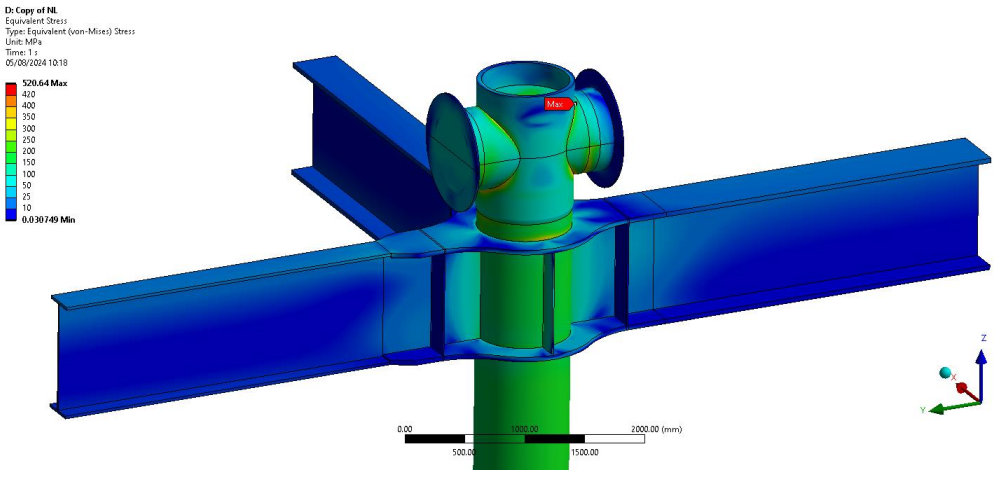
10.2.1 FEA Results

Trunnion joint is checked for Lifting conditions, the most critical ones for this joint. Loads are taken from the Lifting analysis report Ref./24/, 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.

For the Trunnion joint, 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 trunnion joint are plotted.



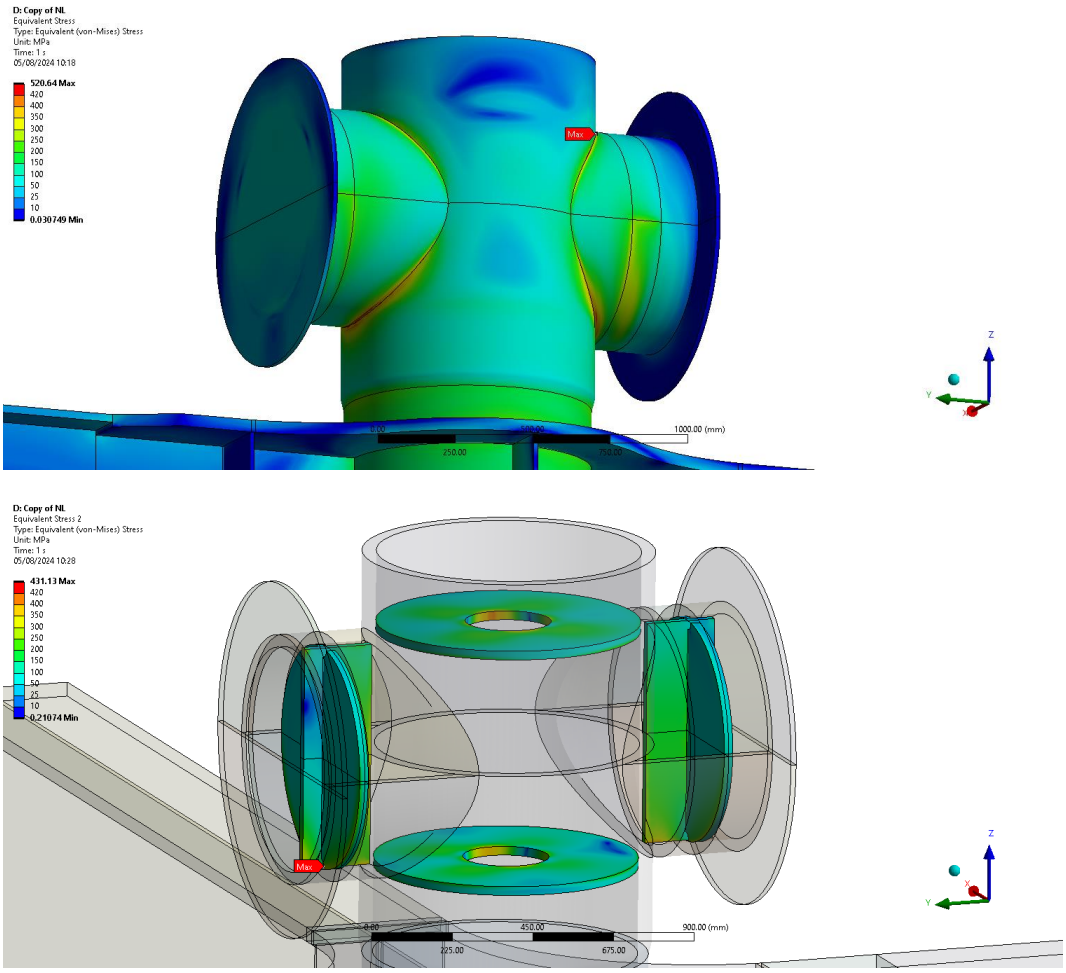


Figure 10.13 – Trunnion Joint FE model – VM Stresses

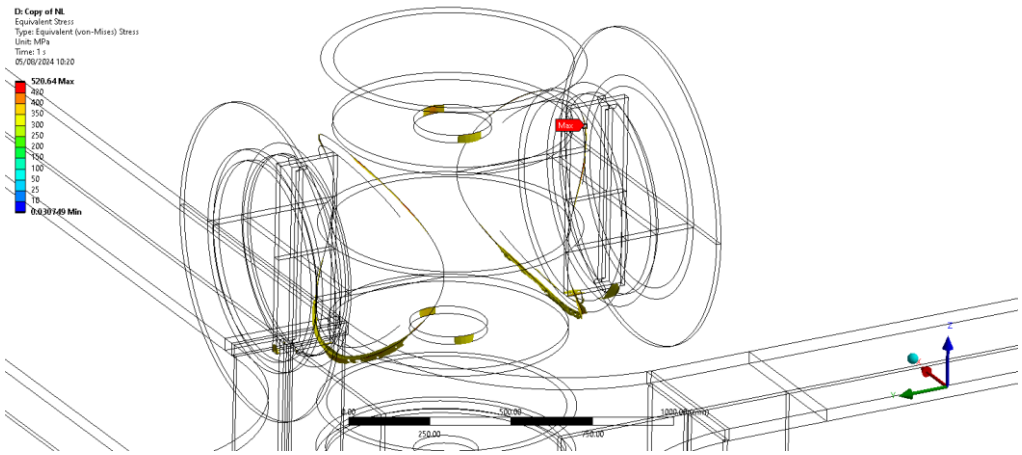


Figure 10.14 – Trunnion Joint FE model – VM Stresses > 340 MPa

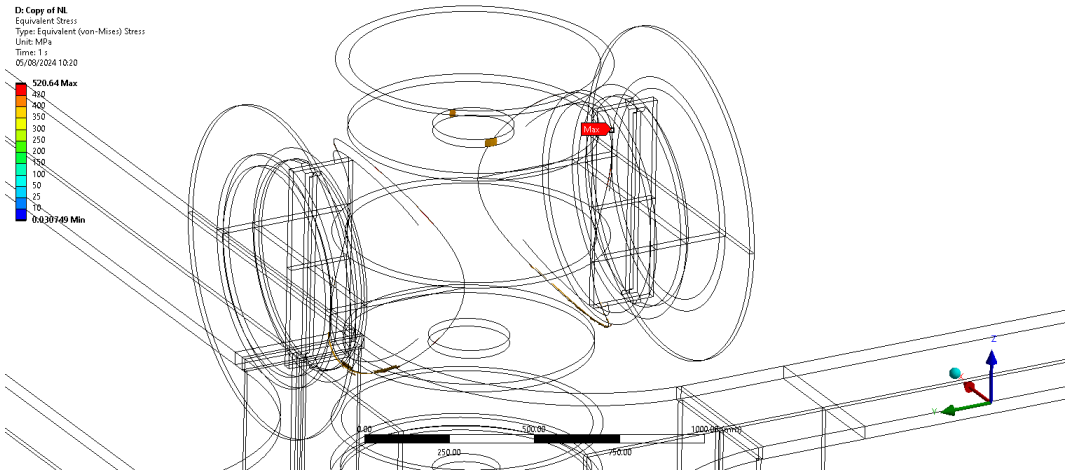


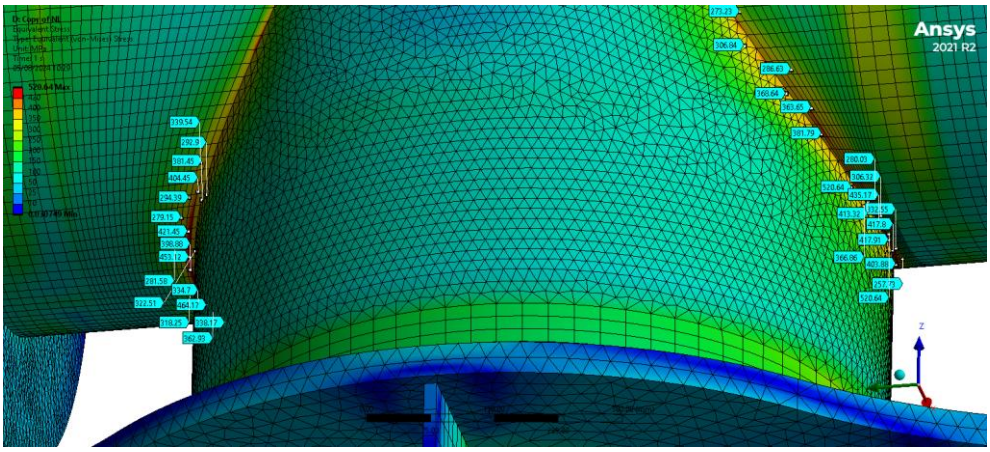
Figure 10.15 – Trunnion Joint FE model – VM Stresses > 400 Mpa

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:

$\sigma_{adm} = \Phi * F_y = 0.85*400 = 340$ Mpa

It can be observed that the VM stresses are largely lower than the admissible stress σ_{adm} = 340 Mpa for global checks, and they are exceeded in few local portions that are the load application surface, the intersection between the vertical Φ820x40 and the horizontals Φ700x35 and the internal surfaces of the rings; these portions are confined in the limit area 2tx2t, not interesting the whole thickness of the affected plates. The cross sections of the elements are then considered have the required capacity to bear the lifting stresses.

The F_y value is exceeded in very limited portion areas, with a peak max. value of σ_{VM} = 521 MPa, so plastic strains in these portions are investigated as well.



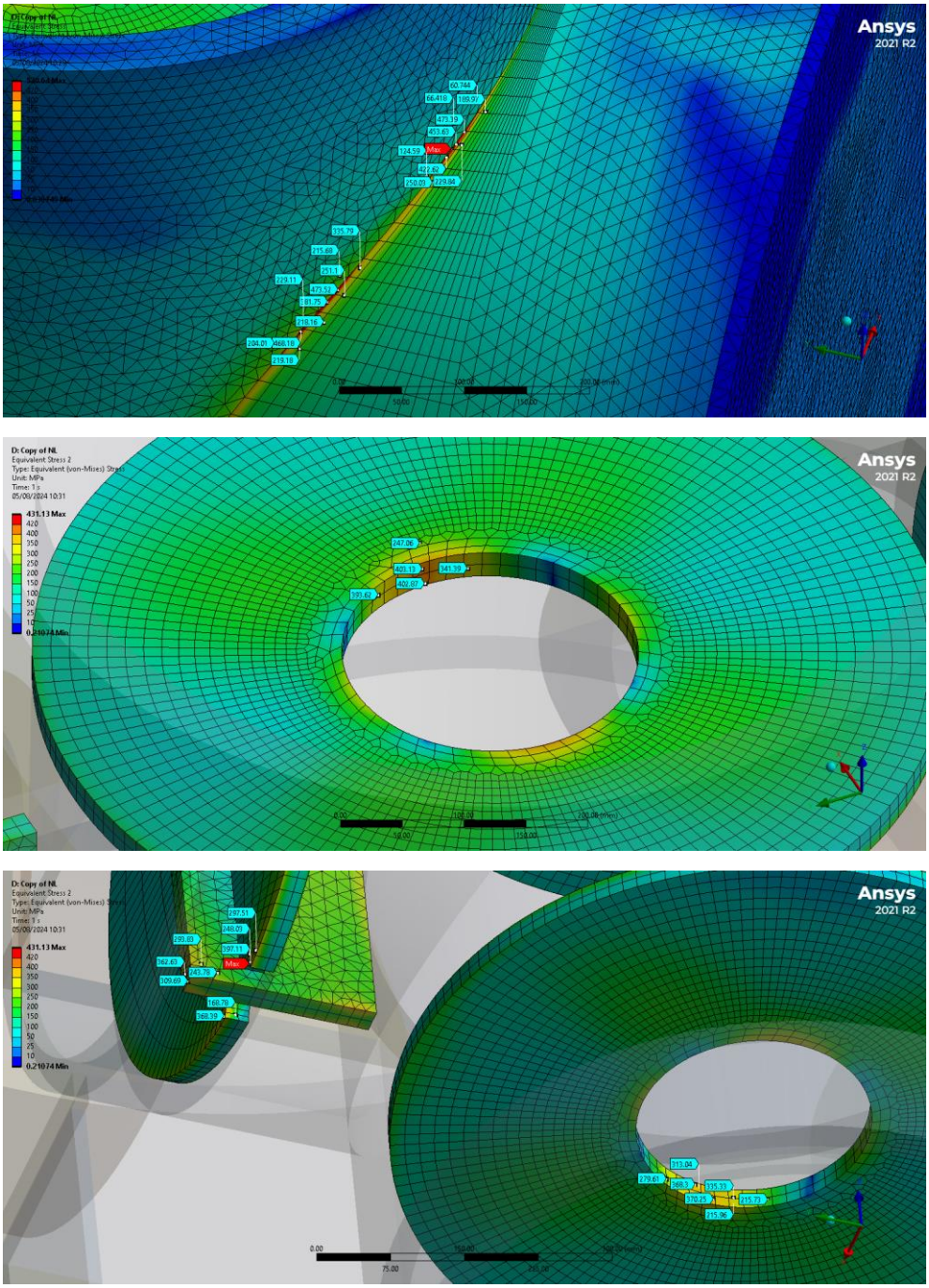


Figure 10.16 – Trunnion Joint FE model – Peak VM Stresses at hot spot areas

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 10.17) and total strains (Figure 10.18) are plotted.

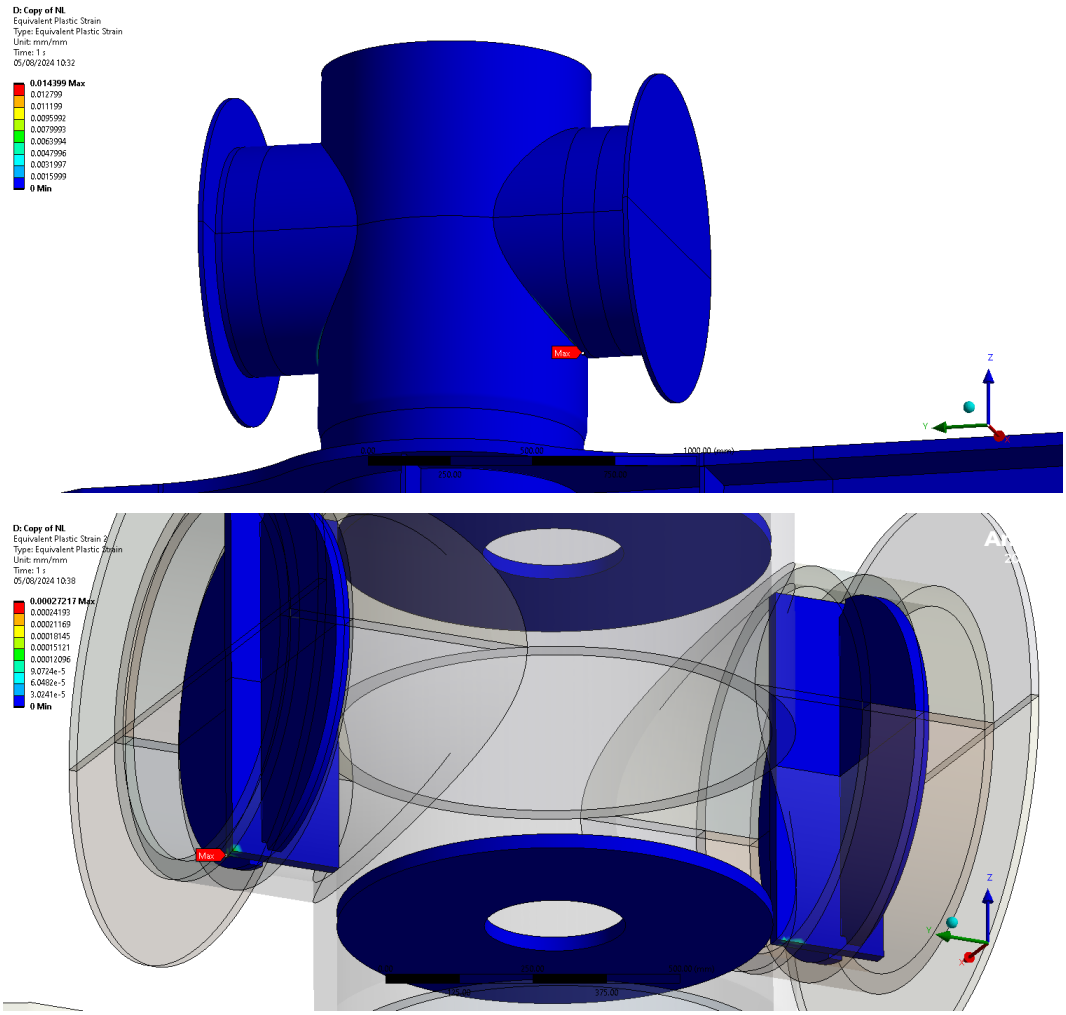


Figure 10.17 – Trunnion Joint FE model – Plastic strains

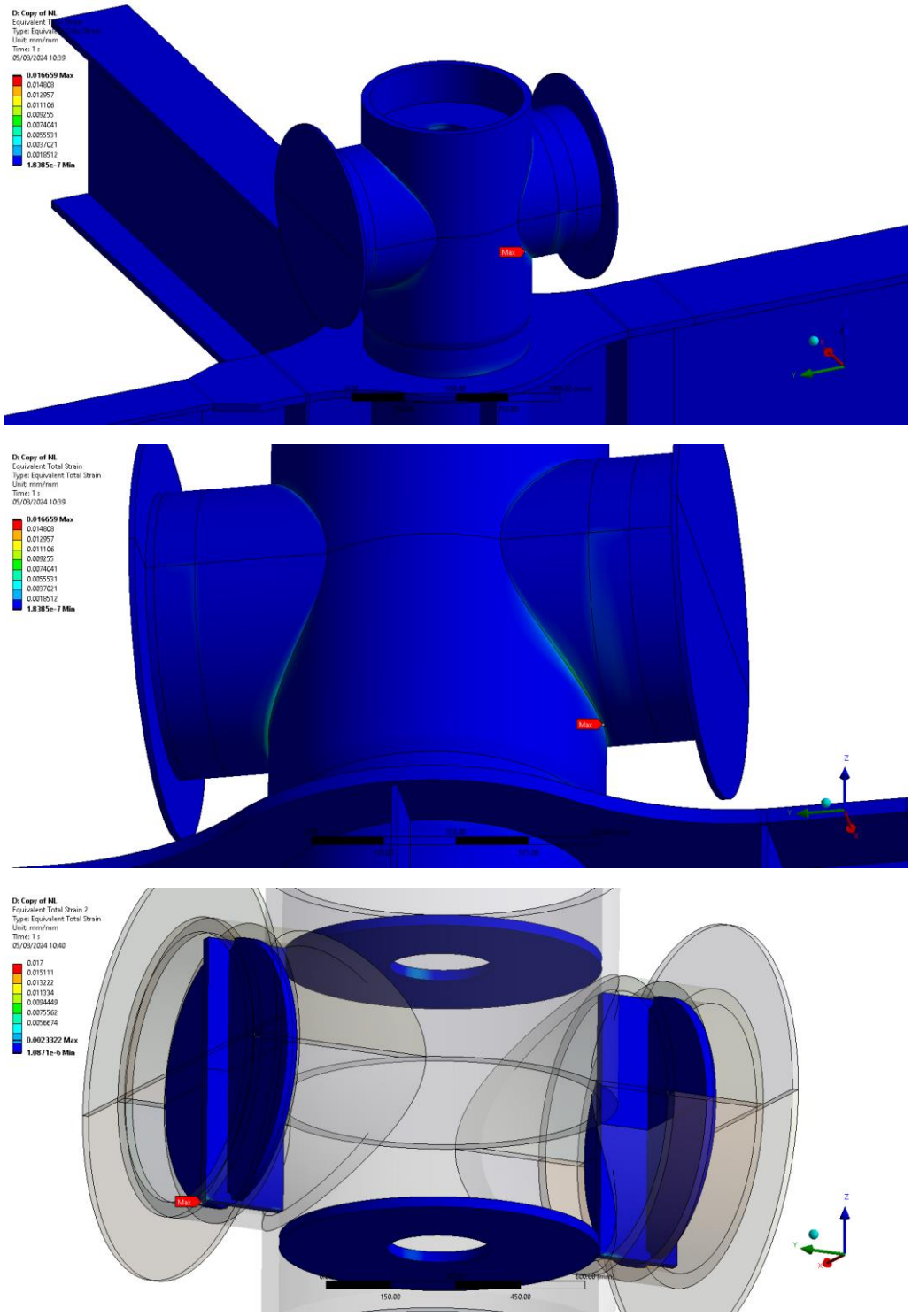


Figure 10.18 – Trunnion Joint FE model – Total strains

It is observed that plastic strains occur in a narrow portion at the intersections between the horizontal and vertical tubulars of the trunnion and at the bottom corners of the vertical stiffeners.

The max. value of total strains is ε_t = 0.017 mm/mm < ε_{adm} =0.05 mm/mm (admissible value for not fatigue sensitive joints). Strains are within the admissible value.

Checks are fulfilled.

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

10.2.2 Local checks for out-of-plane force

Local checks are performed for out-of-plane force. In sec.10.2.1 out-of-plane forces are used to investigate the central column and the trunnions stress state due to the moment they generate. In this section, local checks of the application points of these forces are performed. Stopper plate having thickness 20mm and the internal ring

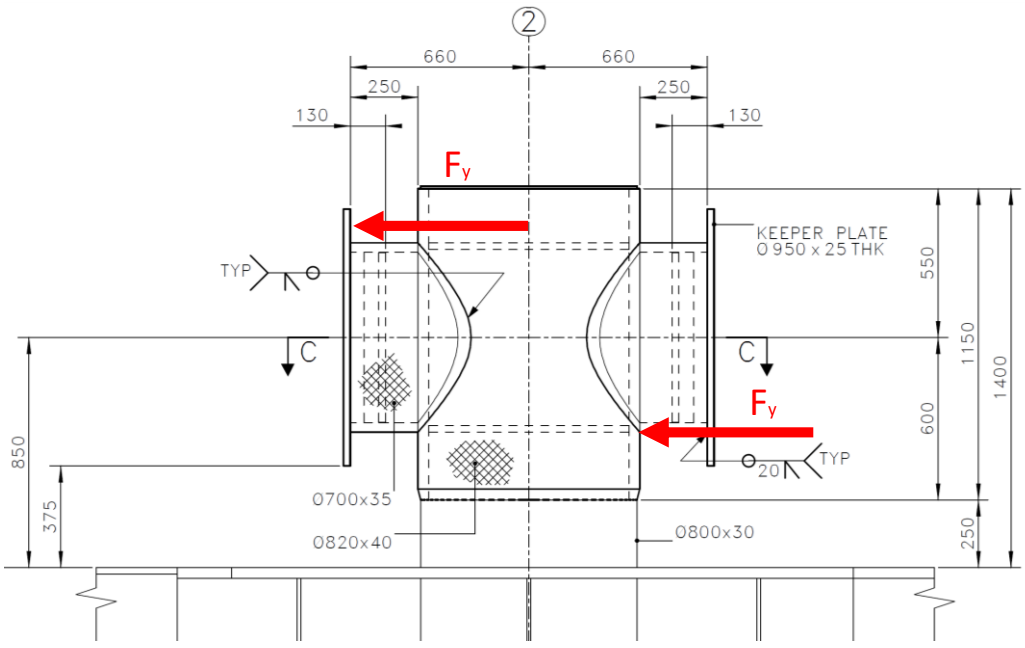


Figure 10.19 – Out-of-plane forces – local checks

The max. out-of-plane force is considered (sec.10.1.5): F_y = 350 kN.

Stopper plate check

The stopper plate is verified considering the ULS force applied to the midspan of its width, at 75mm.

The resisting section is made by the upper semicircular section of the stopper plate having dimensions 1100x25mm (see Figure 10.20).

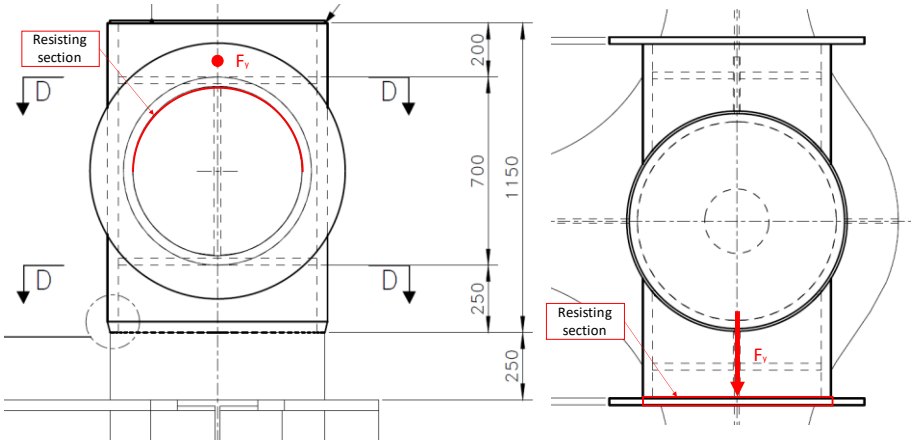


Figure 10.20 – Stopper plate resisting cross section

Checks are performed as follows:

Shear force: V = 350 kN

Bending moment: M = 350x0.075 = 26.25 kNm

Resisting Area: A = 1100x25 = 27500 mm^2

Plastic module: $W_p = 1100x25^2/4 = 171875\ mm^3$

Shear check: $\tau = V/A = 12.75\ MPa < f_v = 345/\sqrt{3} = 200\ MPa$ OK

Bending check: $\sigma = M/W = 151\ MPa < f_b = 345\ MPa$ OK

Ring check

Internal ring is checked considering the ULS force applied in a resisting section given by a quarter of the ring (45°diffusion), so 581x25mm (see Figure 10.21).

Checks are performed as follows:

Axial force: N = 350 kN

Resisting Area: A = 580x25 = 14500 mm^2

Axial check: $\sigma = N/A = 24\ MPa < f_a = 345\ MPa$ OK

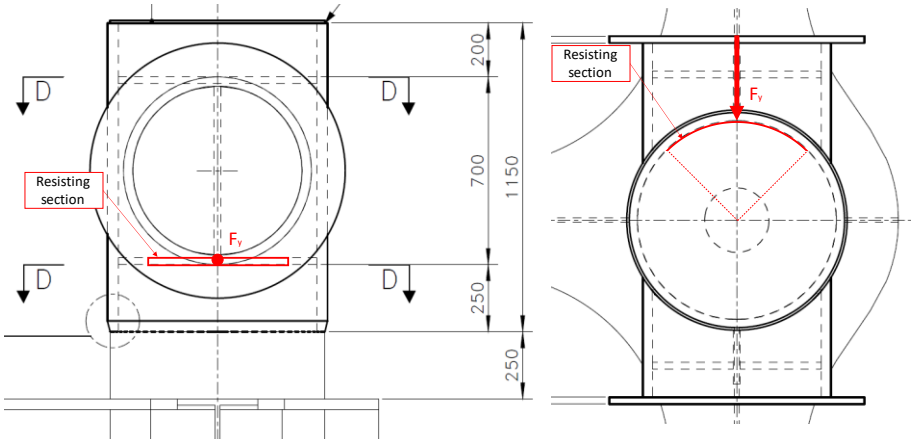


Figure 10.21 – Internal Ring resisting cross section

11 TP-TOPSIDE CONNECTION JOINTS

This section presents the local analysis and results for the Sacs Joint TU02 (Joint A2 at El. +23.31m), which is the most critical joint of the lower level of the Topside structure and is the connection joint between the Topside and the Transition Piece (TP). The results of the analysis performed on this model are valid also for joint TU04 and TU03 which share the same geometry (TU03 is mirrored), because the envelope of the max. loads for the three joints is used for the analysis.

This joint has been analysed for the In-Service and Transportation conditions, being the most critical ones. In the following sections are reported:

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

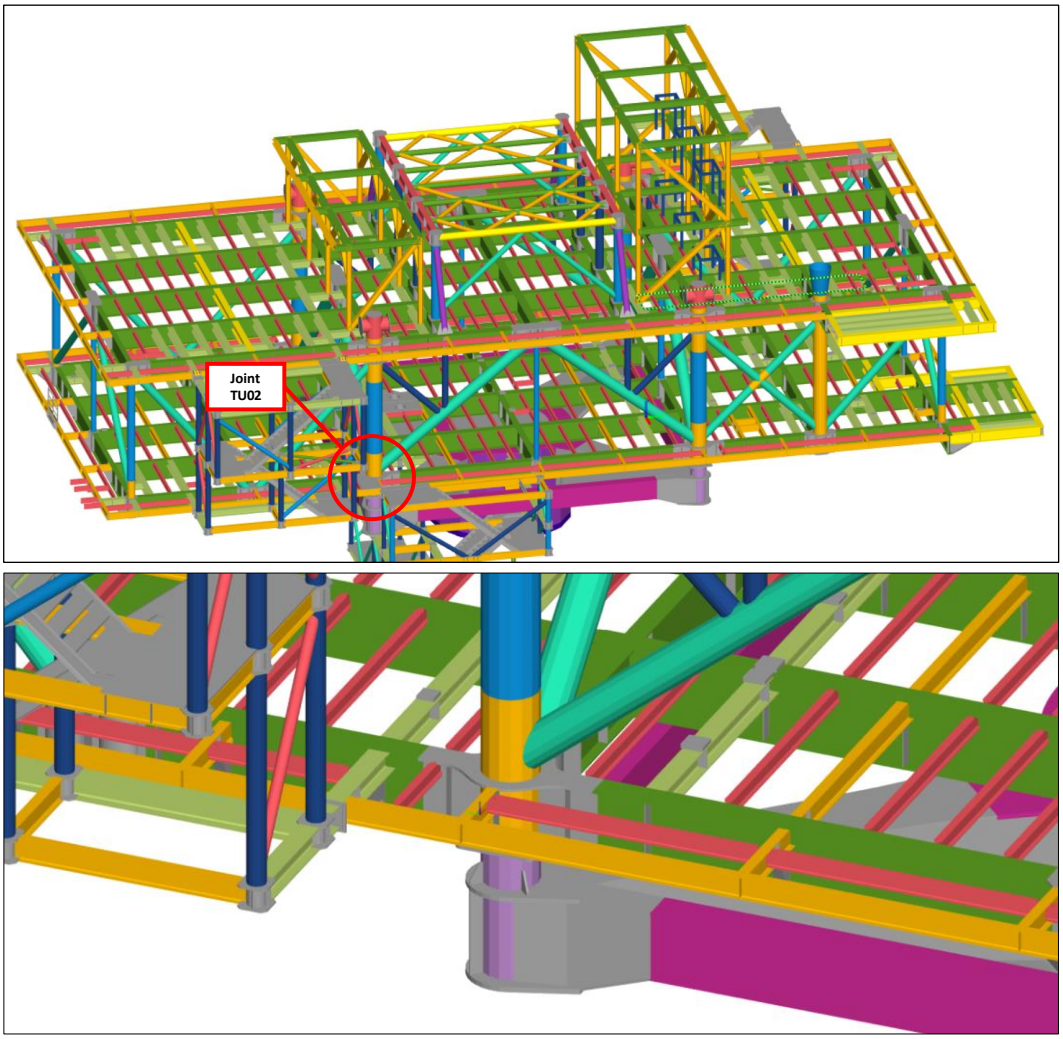


Figure 11.1 – Joint TU02 – 3D view

11.1 FE MODEL

The Joint is analysed for the In-Service and Transportation condition. It is a critical joint since the Topside structure lays on the Transition Piece (in-Place) and on the barge deck at these joints. The most stressed joint, that is SACS joint TU02, is modelled in ANSYS environment; it is represented in Figure 11.2 .

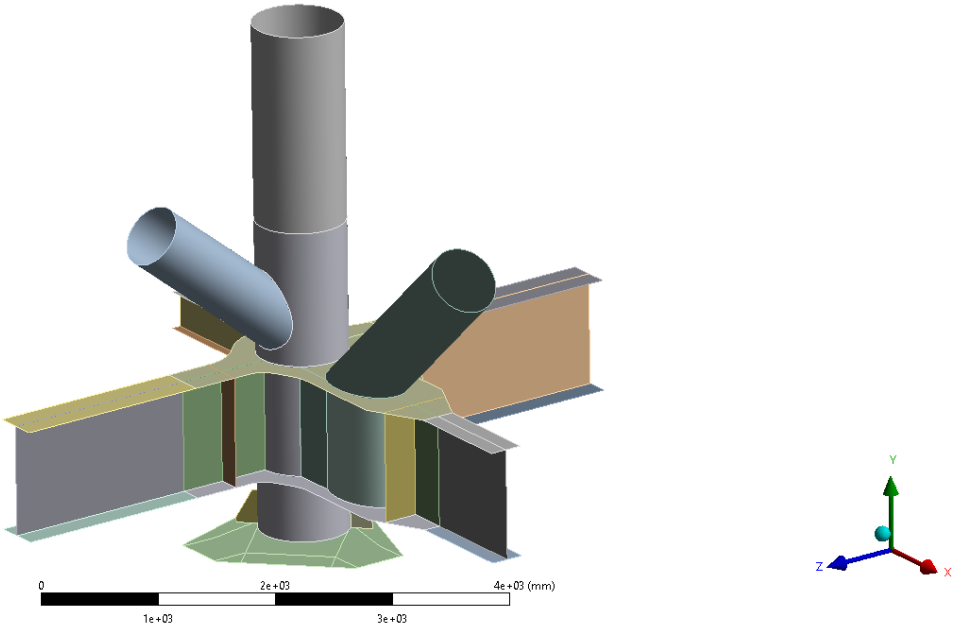


Figure 11.2 – Joint TU02 FE Model

It is a Φ800x40 mm column passing through to the lower level of the topside, made by the crossing of two longitudinal beams WPG1000x400x20x25 mm with a transversal beam WPG1000x300x15x25. The column is intersected by a diagonal brace Φ550x20 and a diagonal Φ600x20 ends on the upper flange of the node on the WPG1000x300x15x25 side. The joint is reinforced with top and bottom 40mm plates and 15mm stiffeners and shells.

The joint is made by S420 steel.

The joint is modelled with the correct geometry and thickness of all the elements as per structural drawings Ref./29/, /31/.

The column is based on transition piece, which is not modelled because not scope of this document (Refer to BLPH Platform – FE Analysis Ref. /25/); only the TP top plate and supporting column of the connection, is modelled, and checked. The vertical stub of TP, that supports the topside main columns, is made by 800x40mm pipe. To correctly model the bevel and offshore partial penetration weld, the FE model has been prepared with 25mm thk plates at the connection with topside bottom flange.

The results of the analyses are also valid for joints TU04 and TU01, which are geometrically identical (TU01 is mirrored) since the maximum envelope of the loads for the three joints is used in the analysis.

11.1.1 Reference System

The global reference system of the FE model is represented in Figure 11.3.

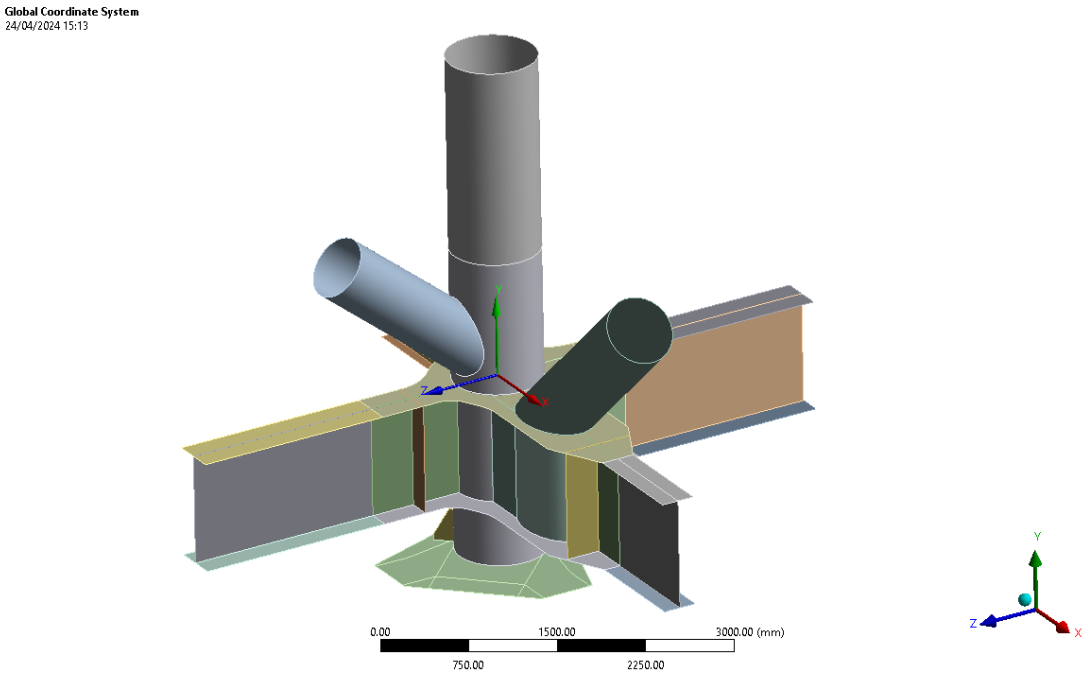


Figure 11.3 – Joint TU02 FE Model – Reference system

The local reference systems of the members framing into the joint, according to the ones of the In-Service SACS model (Ref./22/), are represented in Figure 11.4.

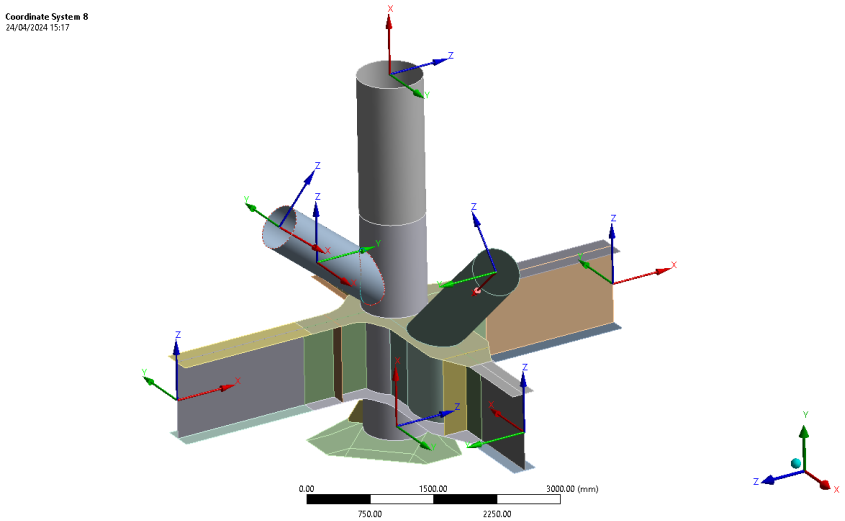


Figure 11.4 – Joint TU02 FE Model –Members Local Reference systems

11.1.2 Material

The material used for all the elements of the node is 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 $12 \cdot 10^{-6}$ / °C

The assumed constitutive law for the material 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 Table 11.1. The curve associated to the previous equation is represented in Figure 11.5.

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

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 11.1 – S420 steel constitutive law parameters

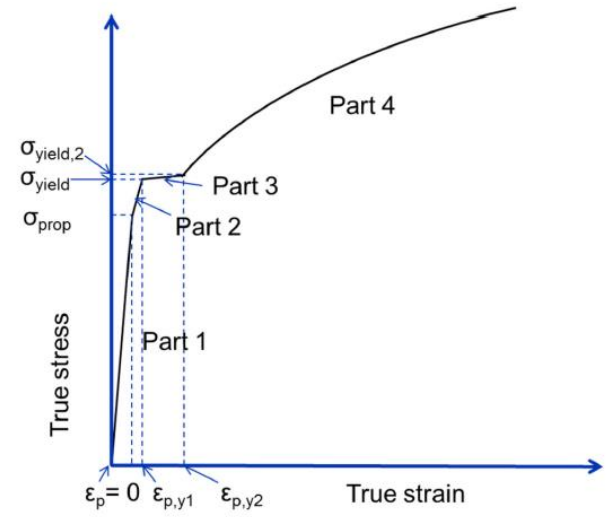


Figure 11.5 – Steel constitutive law according to DNV-RP-C208

11.1.3 Model Mesh

The FE model is meshed using quadrilateral shell elements, as reported in 9.1.1.

Mesh is refined inside the joint. Mesh statistics are reported in Table 10.2.

Mesh Part	Number
Nodes	48869
Elements	49129

Table 11.2 – Joint TU02-TU04 FE model – mesh statistics

In the following figures, FE model mesh for joint is presented.

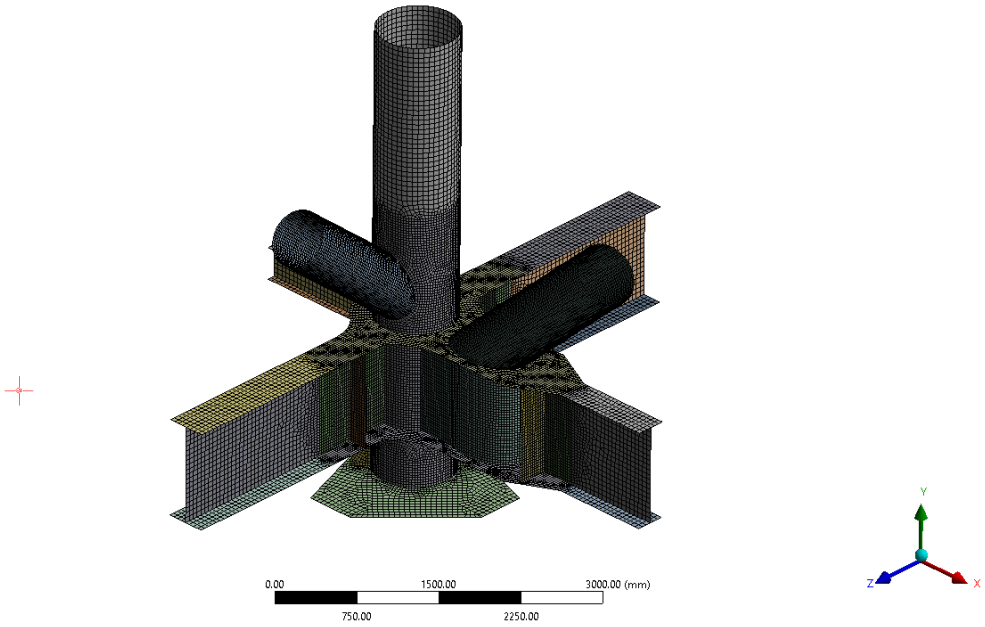


Figure 11.6 – Joint TU02 FE Model mesh – 3D isometric view

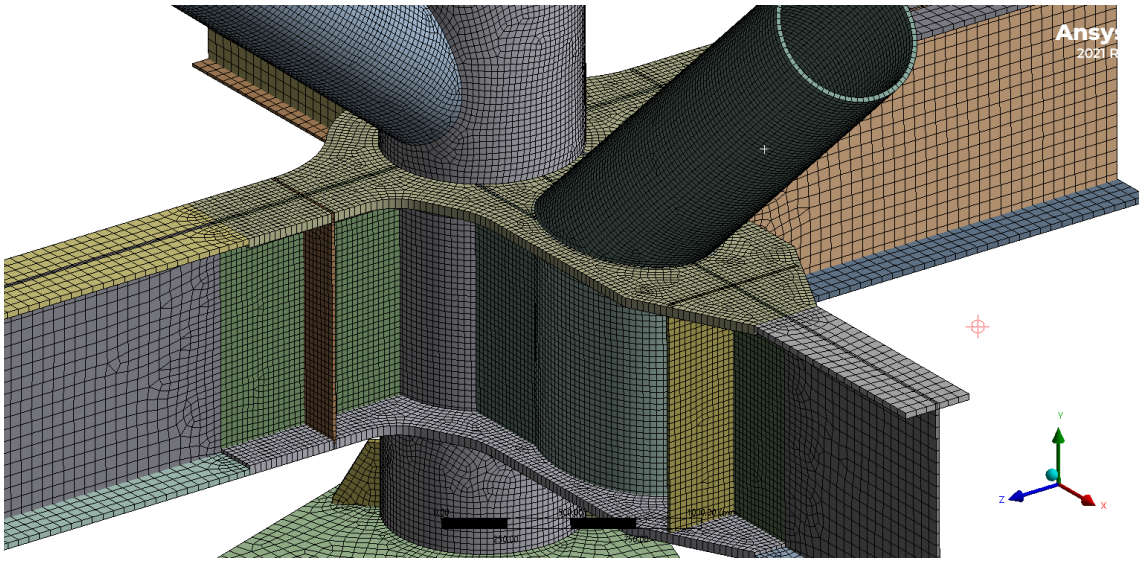


Figure 11.7 – Joint TU02 FE Model mesh – back detail view

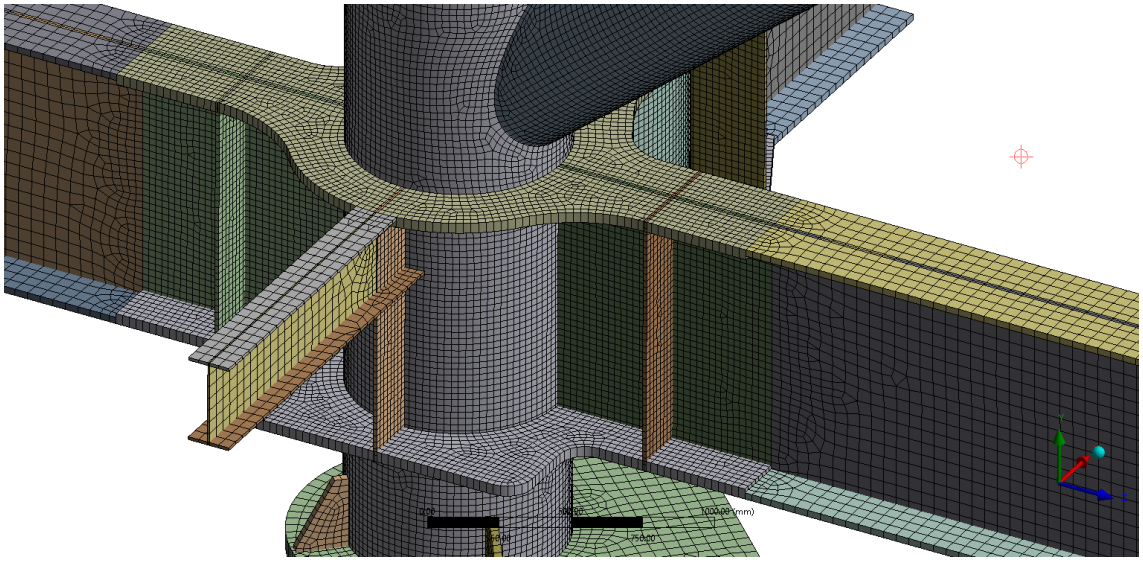


Figure 11.8 – Joint TU02 FE Model mesh – front detail view

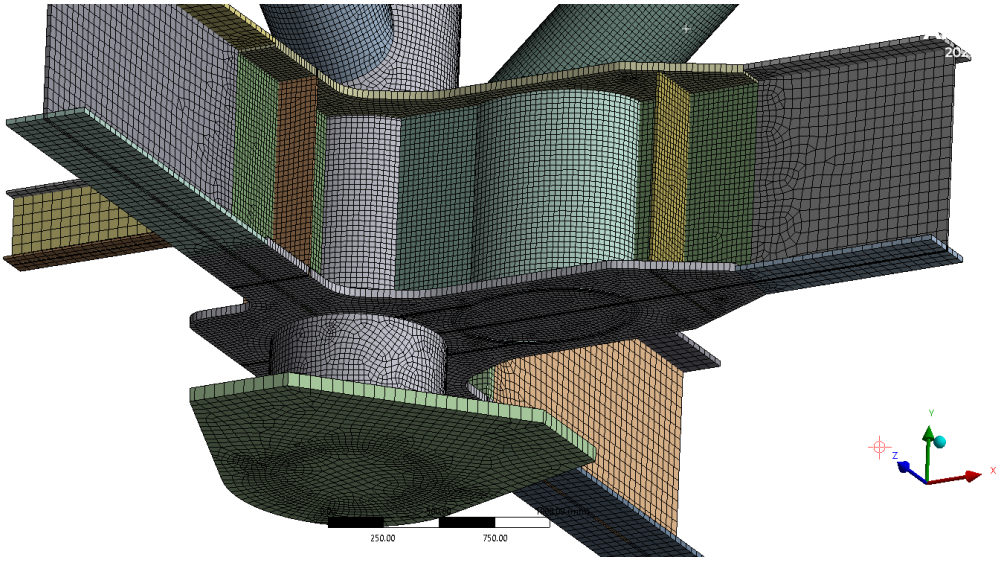


Figure 11.9 – Joint TU02 FE Model mesh – bottom detail view

11.1.4 Constraints

External constraints are applied on the edges of the modelled members that converge into the node in order to restore the actual boundary conditions of the node. Applied constraints are displayed in Figure 11.10.

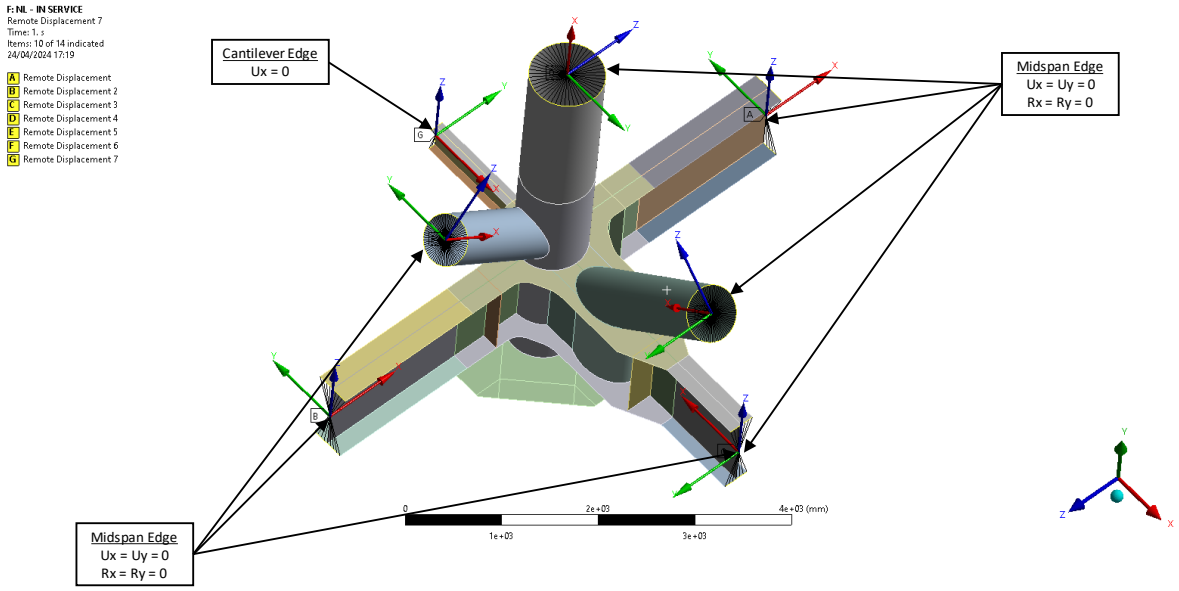


Figure 11.10 – Joint TU02 FE Model – external constraints

11.1.5 Loads

The worst load conditions for this joint are the in-service and transportation, not lifting, hence the joint is analysed for these conditions.

From the in-service analysis report (Ref./24/) and transportation analysis report (Ref./23/), the maximum ULS loads acting on SACS joints TU01, TU02 and TU04 are extracted, and reported in the following tables.

	MEMBER	MEMBER END	GROUP	LOAD CASE	Fx [kN]	Fy [kN]	Fz [kN]	Fyz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
ALL LC	TC44-TU04	TU04	TPS	E24H	**-1535**	-2176	2755	3511	-84	-1374	1743
	TC44-TU04	TU04	TPS	O32D	**-8980**	-1223	2590	2864	-206	-1392	3300
	TA24-TU02	TU02	TPS	E43E	-4279	**2671**	-2316	3535	-61	1176	-2238
	TC44-TU04	TU04	TPS	E33A	-5035	**-2887**	2361	3729	-60	-1098	2535
	TC44-TU04	TU04	TPS	E43G	-3709	-1412	**3648**	3911	-205	-1658	2196
	TA24-TU02	TU02	TPS	E13C	-3614	1266	**-3770**	3977	-250	1839	-1969
	TC44-TU04	TU04	TPS	E24G	-2085	-1149	3095	3302	-162	-1476	1711
	TC44-TU04	TU04	TPS	E24A	-1972	-2434	1813	3035	-23	-1164	1772
	TC44-TU04	TU04	TPS	E33H	-4598	-2629	3303	**4222**	-120	-1308	2506
	TA24-TU02	TU02	TPS	E44H	-6541	-81	-620	**626**	-113	768	-1868
	TC24-TU01	TU01	TPS	E33D	-4520	-256	-3105	3116	**121**	1683	1997
	TA24-TU02	TU02	TPS	E13B	-4861	192	-3264	3270	**-285**	1786	-1950
	TA24-TU02	TU02	TPS	O12C	-5786	1460	-3698	3976	-262	**2155**	-2610
	TC44-TU04	TU04	TPS	O42G	-5998	-1611	3536	3885	-222	**-1899**	2932
	TC44-TU04	TU04	TPS	O32B	-8269	-2113	2644	3385	-158	-1377	**3321**
	TA24-TU02	TU02	TPS	O22F	-8035	1972	-2807	3430	-188	1611	**-2993**
	TC24-TU01	TU01	TPS	E24G	-5898	-726	-335	799	-17	569	1988
	TA24-TU02	TU02	TPS	E34A	-4833	-519	-1565	1648	-191	1225	-1594
	TC44-TU04	TU04	TPS	O32G	-7869	-1858	3530	3989	-214	-1576	3293
	TA24-TU02	TU02	TPS	E34G	-4651	693	-311	759	-37	862	-1649

Table 11.3 – Member loads – In-Service condition

JOINT	Rx [kN]				Ry [kN]				Rz [kN]			
	MAX	LCOND	MIN	LCOND	MAX	LCOND	MIN	LCOND	MAX	LCOND	MIN	LCOND
TA24	3909	C214	-4277	C209	2275	C202	-2513	C205	11176	C209	-946	C316
TA44	3852	C214	-4234	C209	2838	C202	-2603	C205	10345	C209	-631	C316
TC24	4260	C214	-3890	C209	2244	C202	-2456	C205	10548	C214	-1129	C311
TC44	4241	C214	-3862	C209	2844	C202	-2629	C205	10453	C214	-603	C311

Table 11.4 – Member loads – Transportation condition

The max. envelope of these loads is applied to the FE model of the joint, separately for in-service and transportation condition.

Load Condition	Fx	Fy	Fz	Mx	My	Mz
	kN	kN	kN	kNm	kNm	kNm
In-Service	-8989	-2889	3649	-254	-4021	4744
Transportation	-11176	-4277	-2513			

Table 11.5 – Joint TU02 FE Model – applied loads

These forces and moments are applied to the remote point properly defined at the CoG of the member section of the connecting member between the joint and the TP, and according to its local reference system, as shown in sec. 11.1.1 (local reference system of the members are the same of the global model used for In-Service and Transportation analyses). The application of these forces and moments is displayed in the following figures.

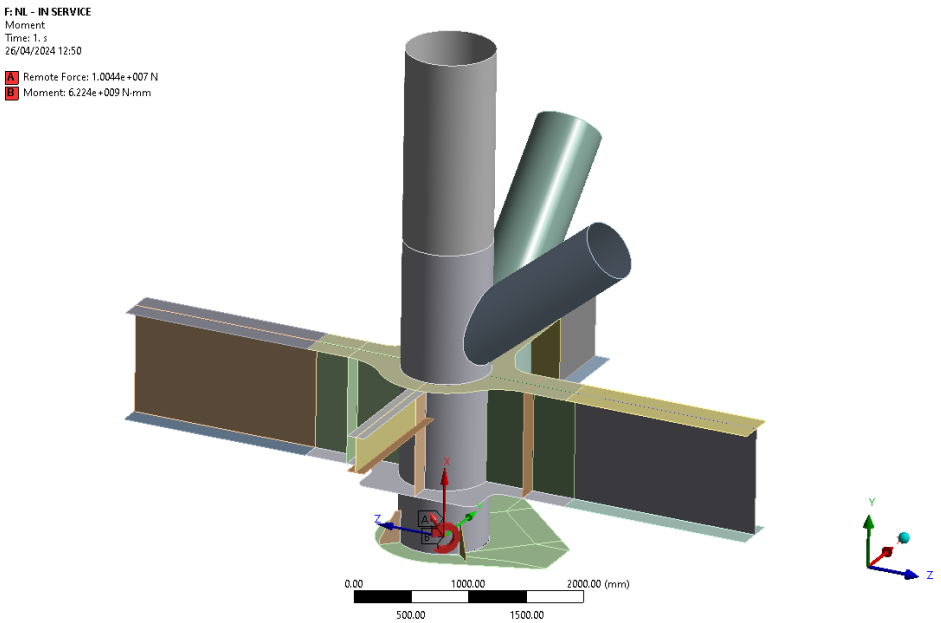


Figure 11.11 – Joint TU02 FE Model – applied loads – In-Service condition

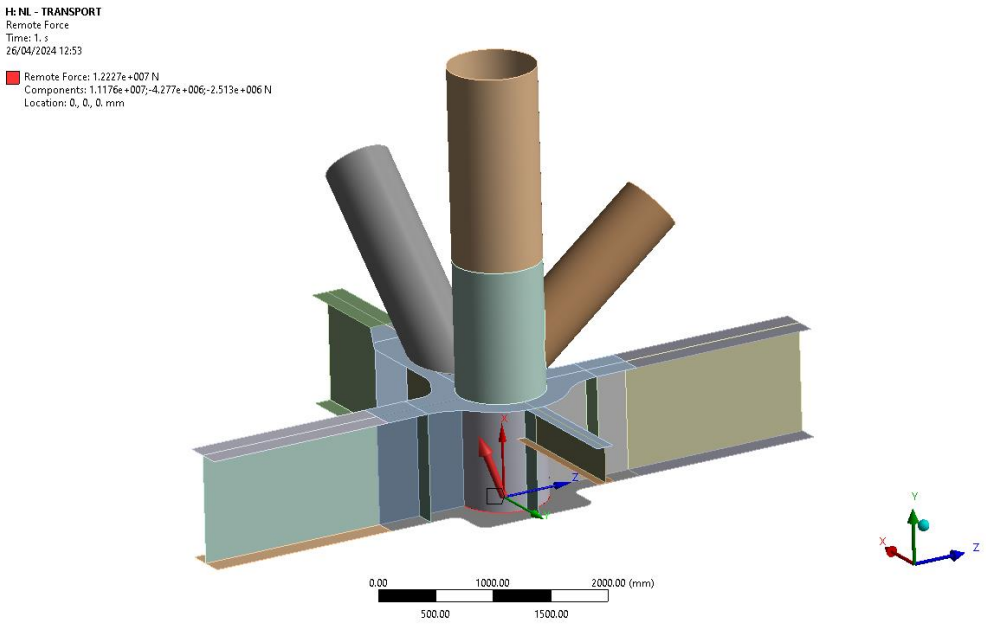


Figure 11.12 – Joint TU02 FE Model – applied loads – Transportation condition

Note: the FE model of the Transportation condition is modified, removing the member connecting the Joint with the TP, since during transportation the joint is supported by the grillage. Loads are directly applied to the node bottom side in contact with the grillage.

11.2 FEM ANALYSIS AND RESULTS

Joint TU02 is checked for both In-Service and Transportation conditions. It is the most critical joint of the lower elevation of the topside structure since it is the connection joint between topside and TP.

Loads are taken from the In-Service and Transportation analysis reports (Ref./22/,/23/), according to the relative ULS load combinations. Applied loads as envelope of the whole set of combinations for joints TU01, TU02 and TU04 are reported in sec. 11.1.5.

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.

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

11.2.1 In-Service Condition

Here are reported FEA results and checks for the In-Service condition.

In the following figures, the Von Mises stresses for the TU02 joint are plotted.

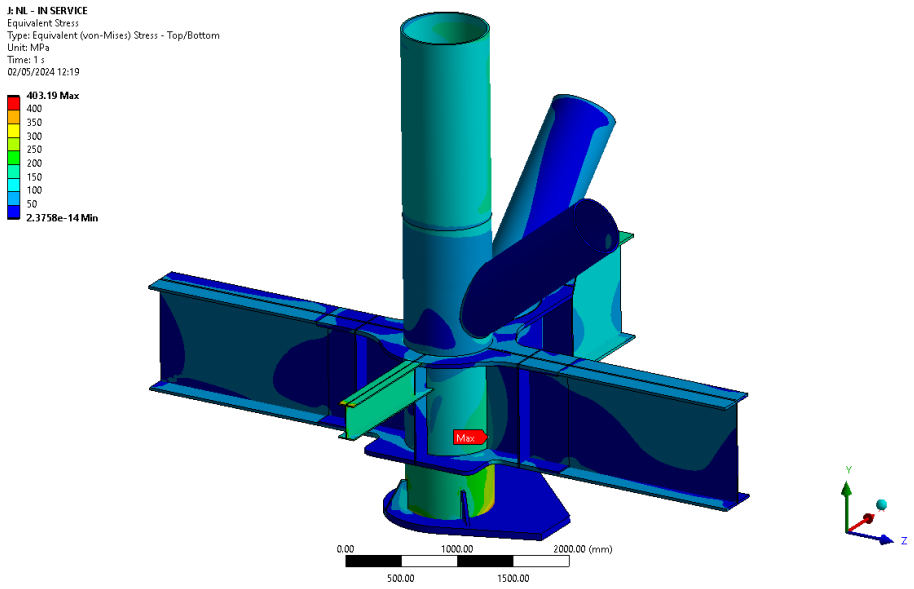


Figure 11.13 – Joint TU02 FE model – VM Stresses (front view)

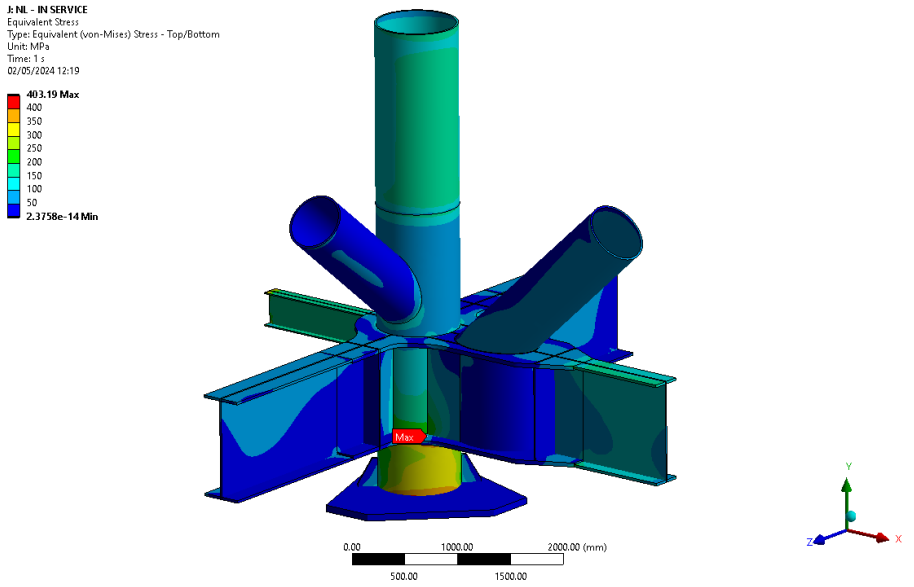


Figure 11.14 – Joint TU02 FE model – VM Stresses (back view)

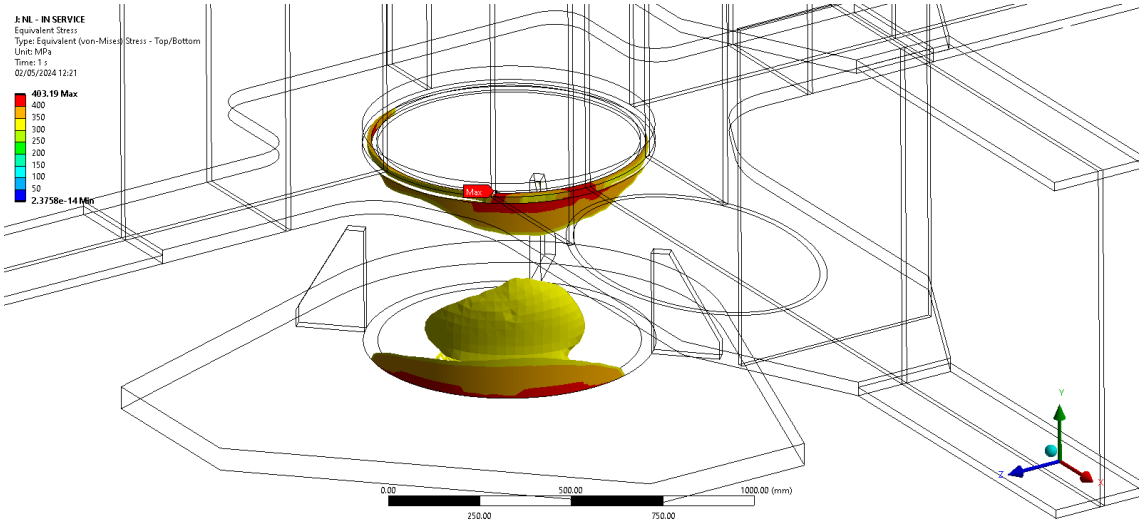


Figure 11.15 – Joint TU02 FE model – VM Stresses > 340 MPa

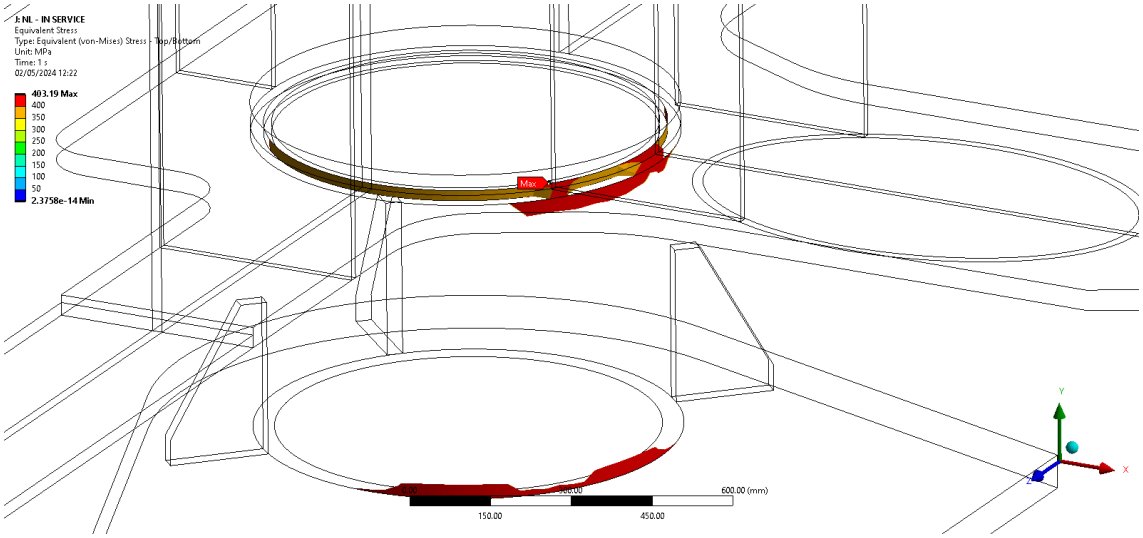


Figure 11.16 – Joint TU02 FE model – VM Stresses > 400 MPa

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:

$\sigma_{adm} = \Phi * F_y = 0.85*400 = 340$ MPa

It can be observed that the VM stresses are largely lower than the admissible stress σ_{adm} = 340 MPa for global checks, and they are exceeded in areas on the vertical Φ800x40 connecting the joint to the TP, at the intersection with the TP 65mm flange and at the upper part welded to the 40mm bottom flange of the joint; these portions are larger than the limit area 2tx2t, not interesting the whole thickness of the plates. The value of Fy is reached in these areas.

Since the stress checks are not fulfilled due to the extension of the areas, the deformations are checked.

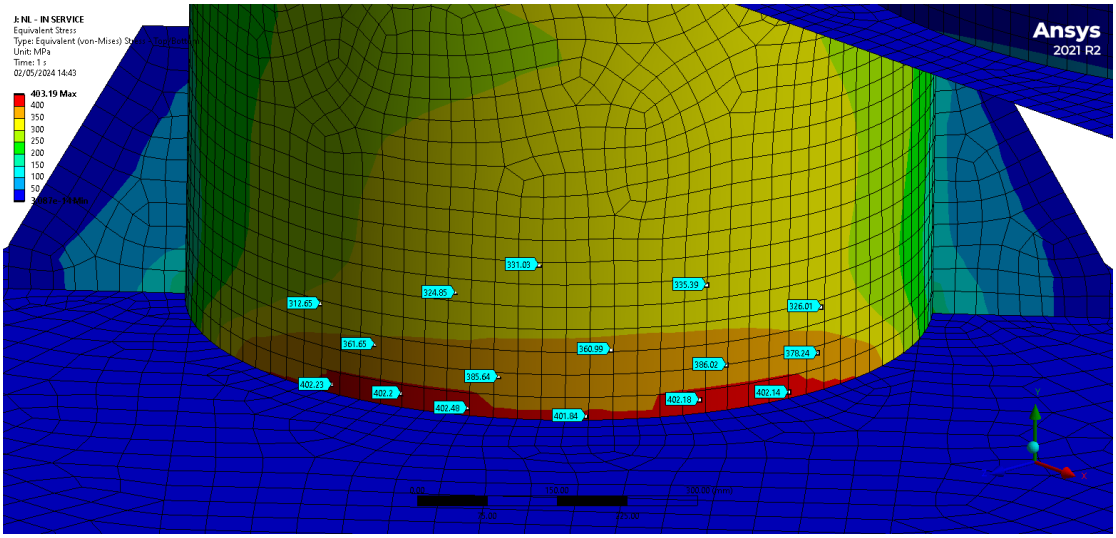


Figure 11.17 – Joint TU02 FE model – Peak VM stresses at hot spot area BOTTOM

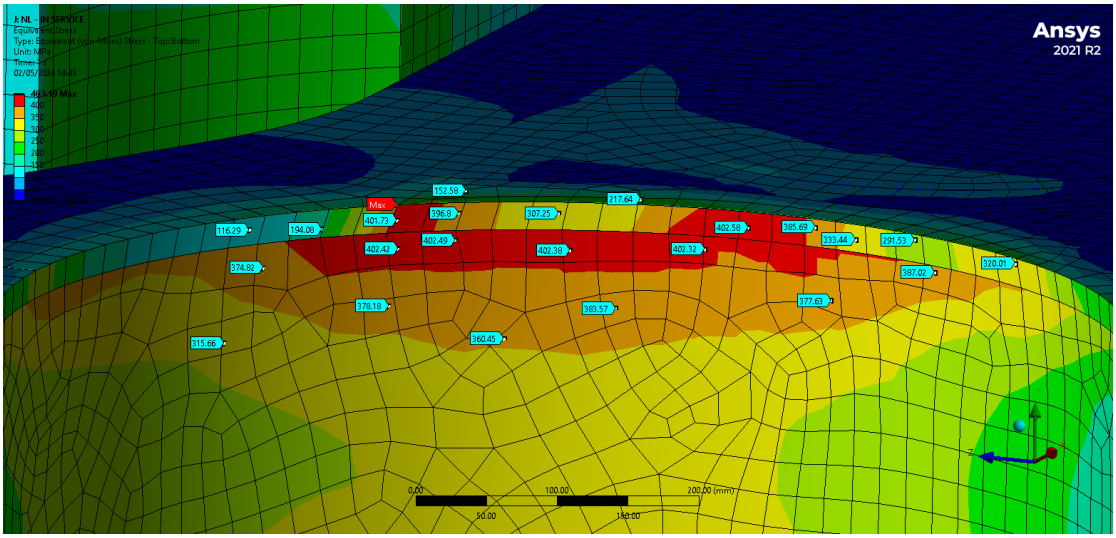


Figure 11.18 – Joint TU02 FE model – Peak VM stresses at hot spot area TOP

For the hot spot areas, elasto-plastic strains are measured according to the performed non-linear analysis. They shall be less than 1%, as per design criteria defined in Sec. 8.3, since the joint is fatigue sensitive during transportation.

In the following figures, plastic strains (Figure 10.17) and total strains (Figure 10.18) are plotted.

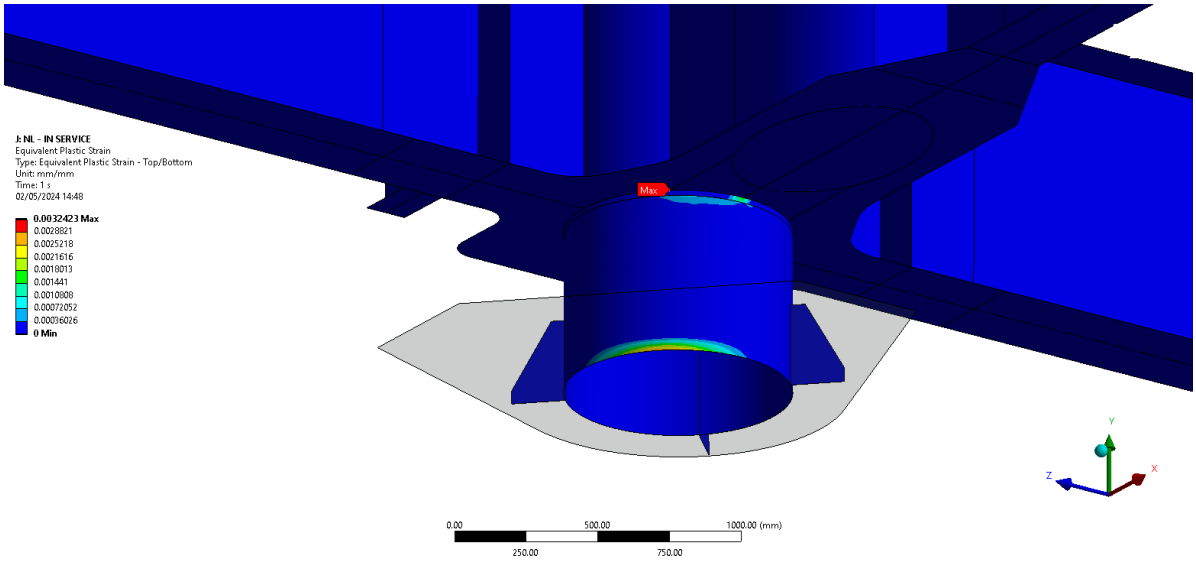


Figure 11.19 – Joint TU02 FE model – Plastic strains

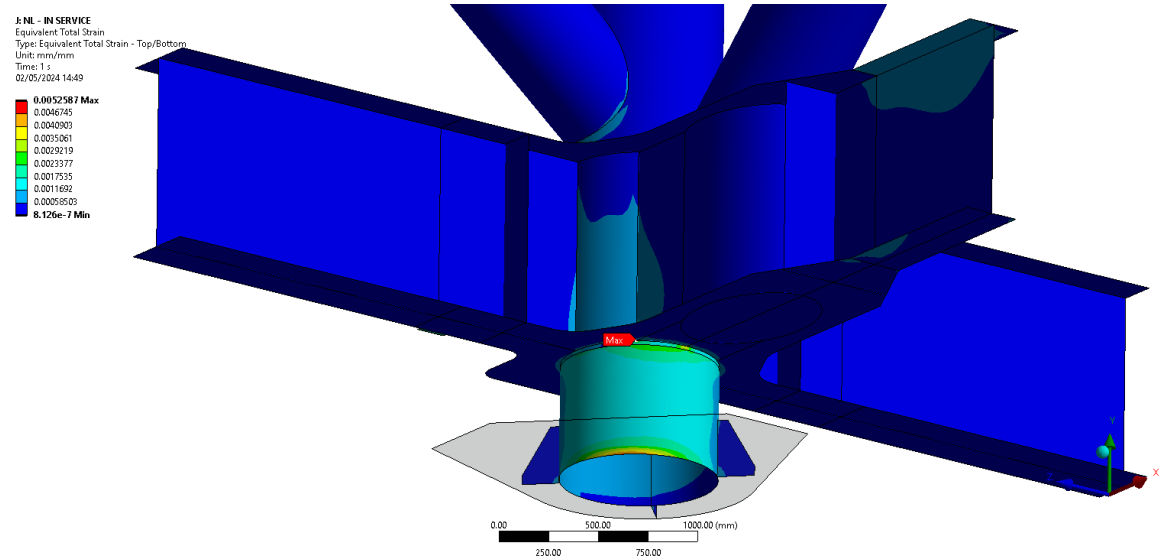


Figure 11.20 – Joint TU02 FE model – Total strains

It is observed that plastic strains occur in a narrow portion at the intersections between the vertical tubular member and the bottom 65mm plate.

The max. value of total strains is ε_t = 0.005 mm/mm < ε_{adm} =0.01 mm/mm. Strains are within the admissible value.

Checks are fulfilled, he joint has enough resistance capacity to withstand the loads for In-Service conditions.

11.2.2 Transportation Condition

Here are reported FEA results and checks for the Transportation condition. In the following figures, the Von Mises stresses for the TU02 joint are plotted.

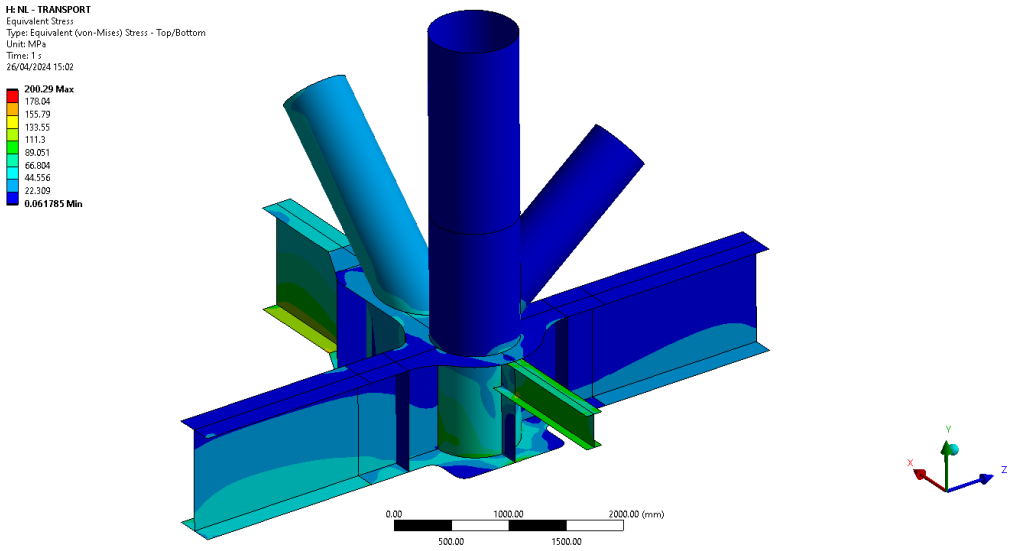


Figure 11.21 – Joint TU02 FE model – VM Stresses (front view)

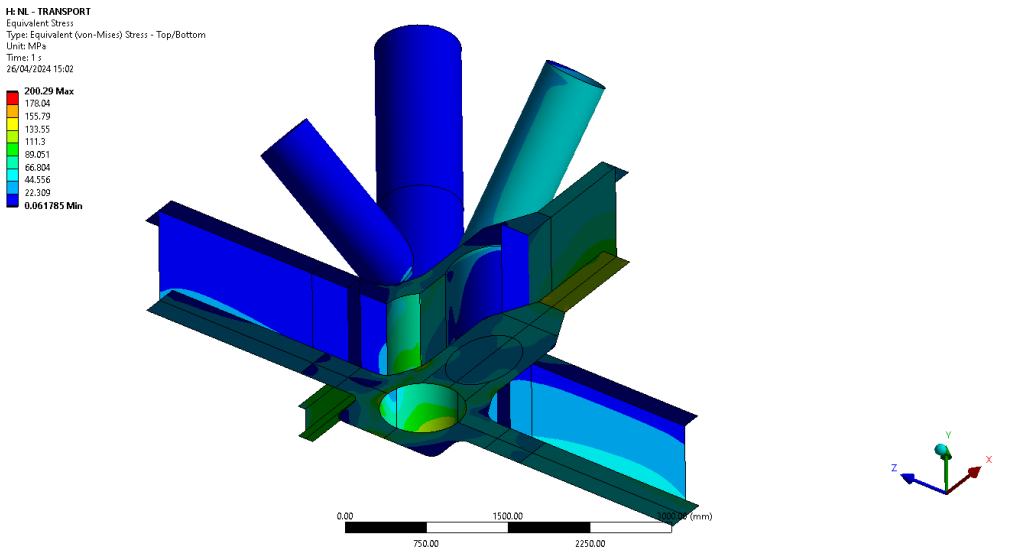


Figure 11.22 – Joint TU02 FE model – VM Stresses (back view)

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:

$\sigma_{adm} = \Phi * F_y = 0.85*400 = 340$ MPa

All VM stresses are largely lower than the admissible stress $\sigma_{adm} = 340$ MPa for global checks. The max value registered is $\sigma_{VM} = 200$ MPa < $\sigma_{adm} = 340$ MPa.

Stress checks are fulfilled; therefore, the joint has enough resistance capacity to withstand the loads for Transportation conditions.

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

12 TP-TOPSIDE CONNECTION JOINT TU03

This section presents the local analysis and results for the Sacs Joint TU03 (Joint A4 at El. +23.31m), which is a critical joint of the lower level of the Topside structure because it is connected to the crane column through a brace. This joint has been analysed for the In-Service conditions, being the most critical due to the working crane. In the following sections are reported:

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

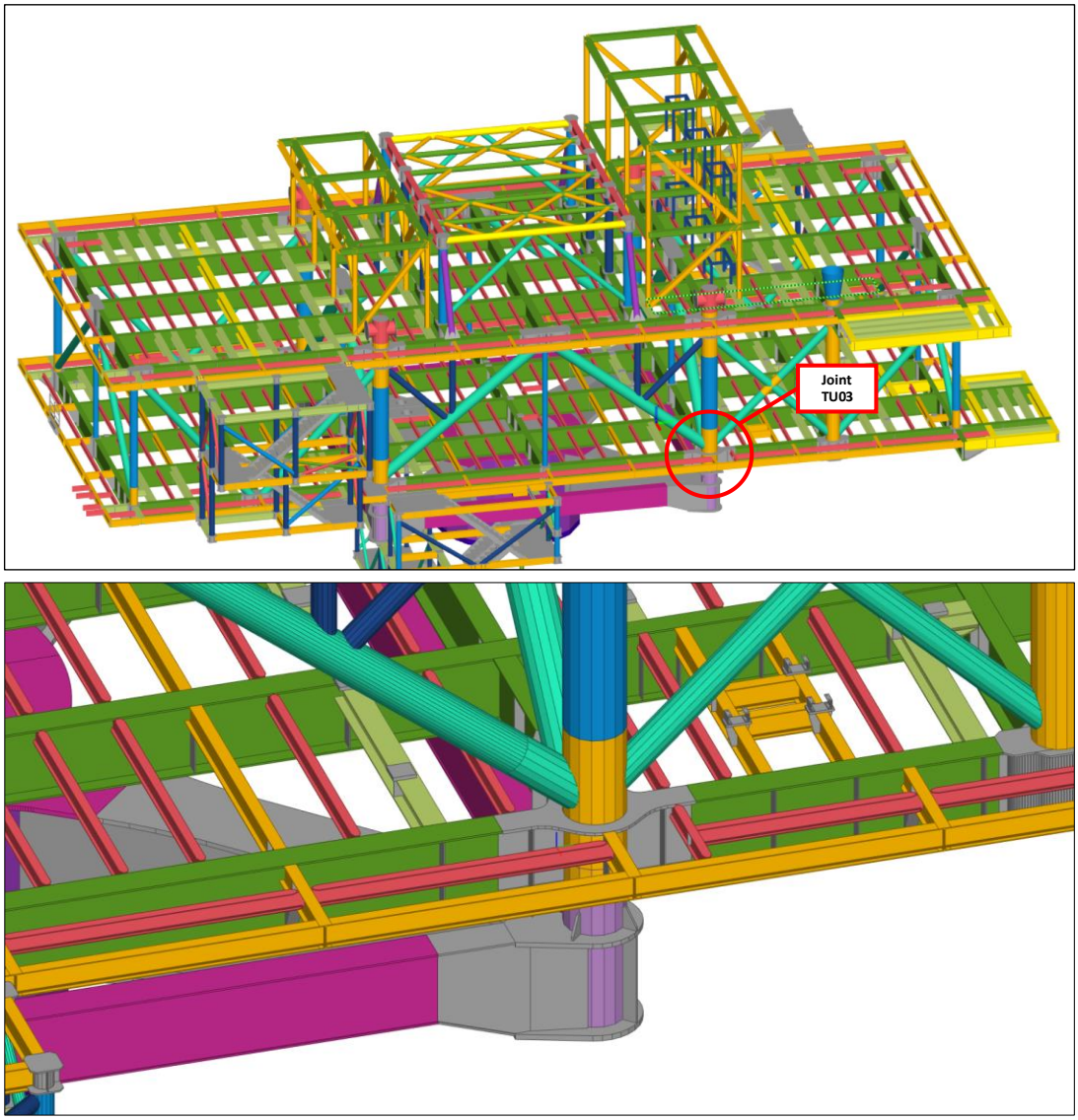


Figure 12.1 – Joint TU02 – 3D view

12.1 FE MODEL

The Joint is analysed for the In-Service condition. It is a critical joint since the Topside structure lays on the Transition Piece (in-Place) at this joint, and because it is connected to the crane that transmit forces to the joint when it is operating. This joint, which corresponds to SACS joint TU03 is modelled in ANSYS environment; it is represented in Figure 12.2.

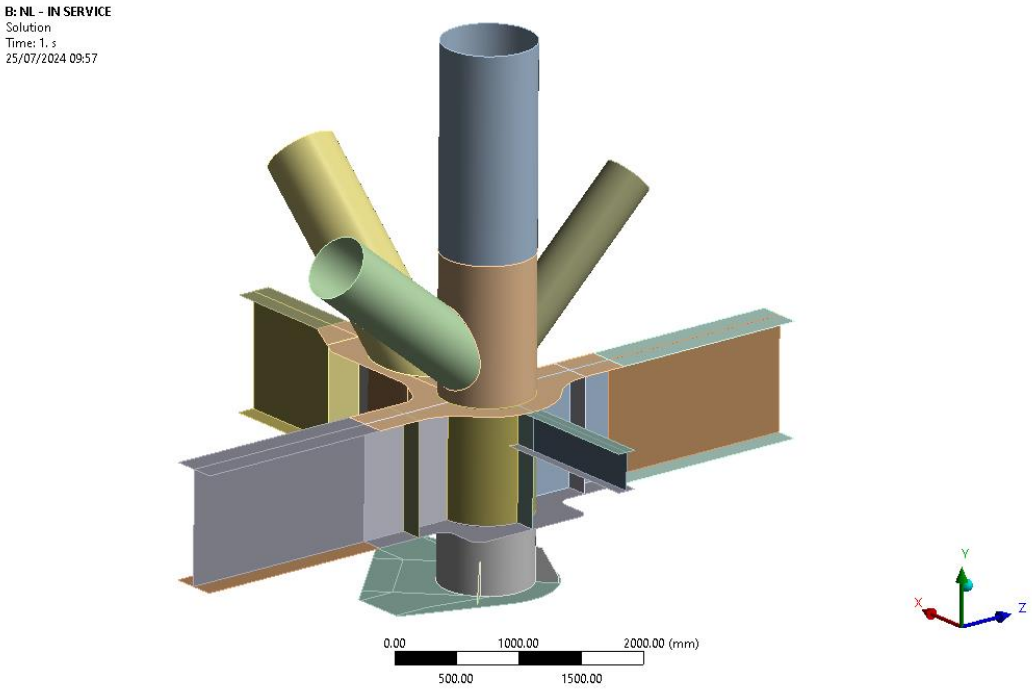


Figure 12.2 – Joint TU03 FE Model

It is a Φ800x40 mm column passing through to the lower level of the topside, made by the crossing of two longitudinal beams WPG1000x400x20x25 mm with a transversal beam WPG1000x300x15x25. The column is intersected by a diagonal brace Φ550x20 and a diagonal Φ600x20 ends on the upper flange of the node on the WPG1000x300x15x25 side. The brace Φ406x19 that converges to the joint connects it to the crane. The joint is reinforced with top and bottom 40mm plates and 15mm stiffeners and shells.

The joint is made by S420 steel.

The joint is modelled with the correct geometry and thickness of all the elements as per structural drawings Ref./27/, /29/, /31/.

The column is based on transition piece, which is not modelled because not scope of this document (Refer to BLPH Platform – FE Analysis Ref. /25/); only the TP top plate and supporting column of the connection, is modelled, and checked. The vertical stub of TP, that supports the topside main columns, is made by 800x40mm pipe. To correctly model the bevel and offshore partial penetration weld, the FE model has been prepared with 25mm thk plates at the connection with topside bottom flange.

12.1.1 Reference System

The global reference system of the FE model is represented in Figure 12.3.

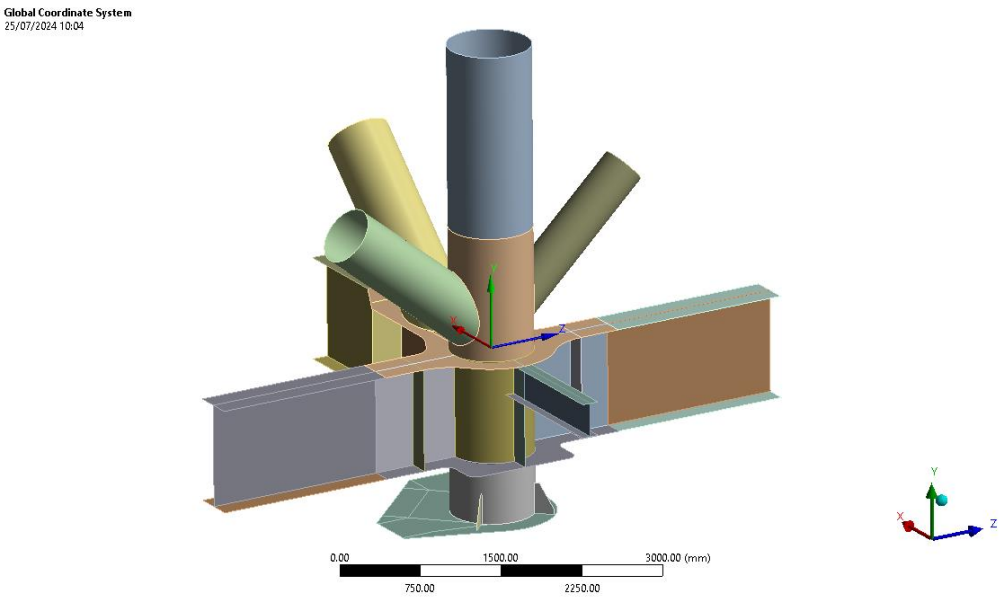


Figure 12.3 – Joint TU03 FE Model – Reference system

The local reference systems of the members framing into the joint, according to the ones of the In-Service SACS model (Ref./22/), are represented in Figure 12.4.

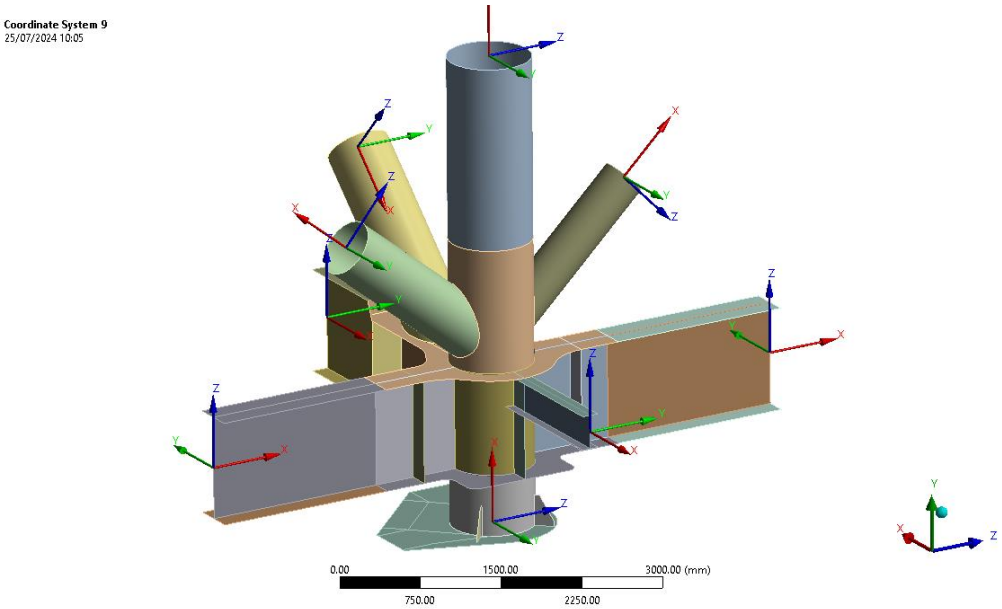


Figure 12.4 – Joint TU03 FE Model –Members Local Reference systems

12.1.2 Material

The material used for all the elements of the node is 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 $12 \cdot 10^{-6}$ / °C

The assumed constitutive law for the material 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 Table 11.1. The curve associated to the previous equation is represented in Figure 11.5.

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

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 12.1 – S420 steel constitutive law parameters

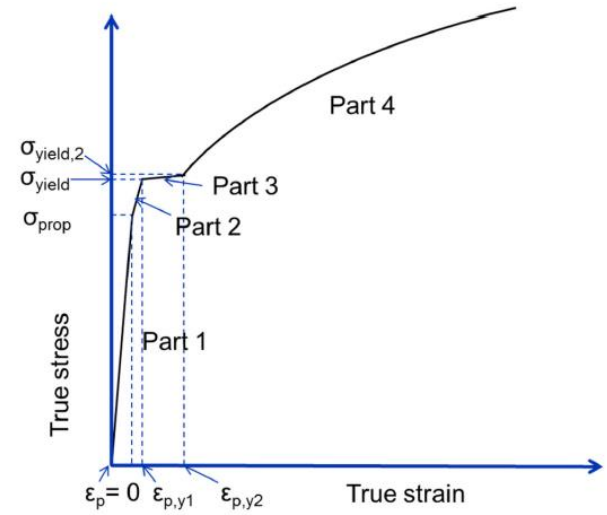


Figure 12.5 – Steel constitutive law according to DNV-RP-C208

12.1.3 Model Mesh

The FE model is meshed using quadrilateral shell elements, as reported in 9.1.1.

Mesh is refined inside the joint. Mesh statistics are reported in Table 12.2.

Mesh Part	Number
Nodes	52162
Elements	52445

Table 12.2 – Joint TU03 FE model – mesh statistics

In the following figures, FE model mesh for joint is presented.

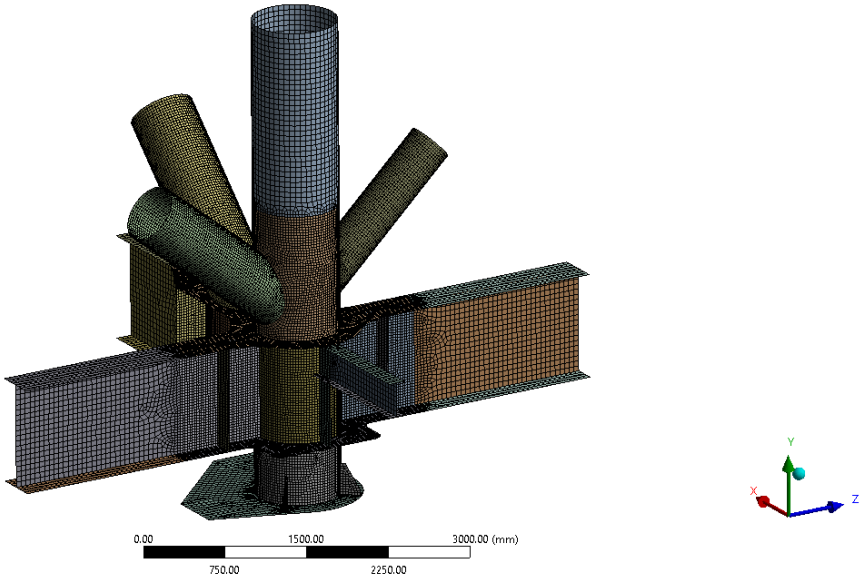


Figure 12.6 – Joint TU03 FE Model mesh – 3D isometric view

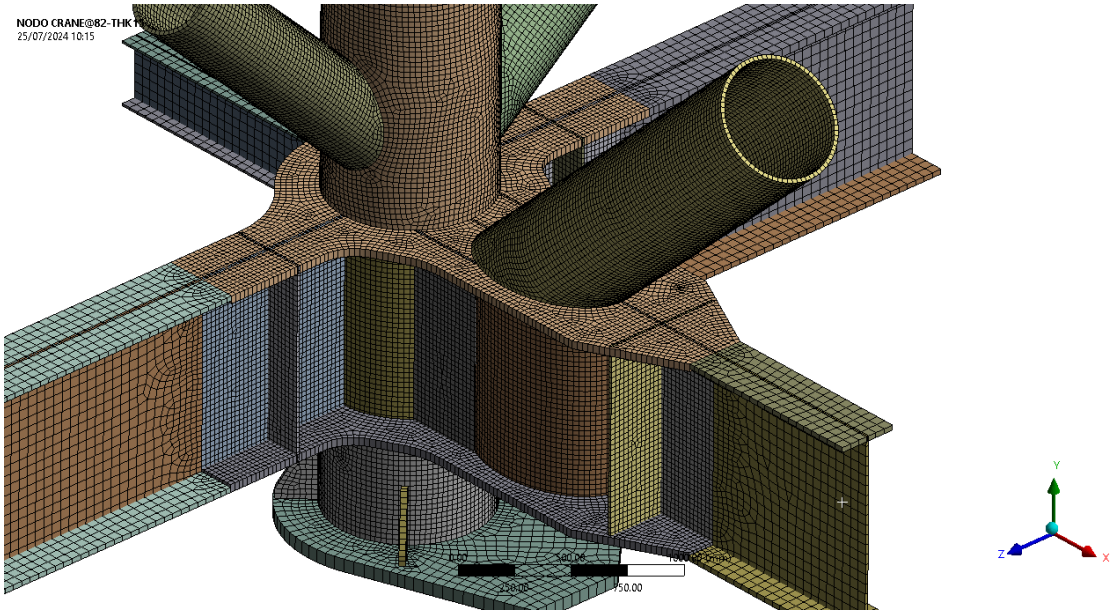


Figure 12.7 – Joint TU03 FE Model mesh – back detail view

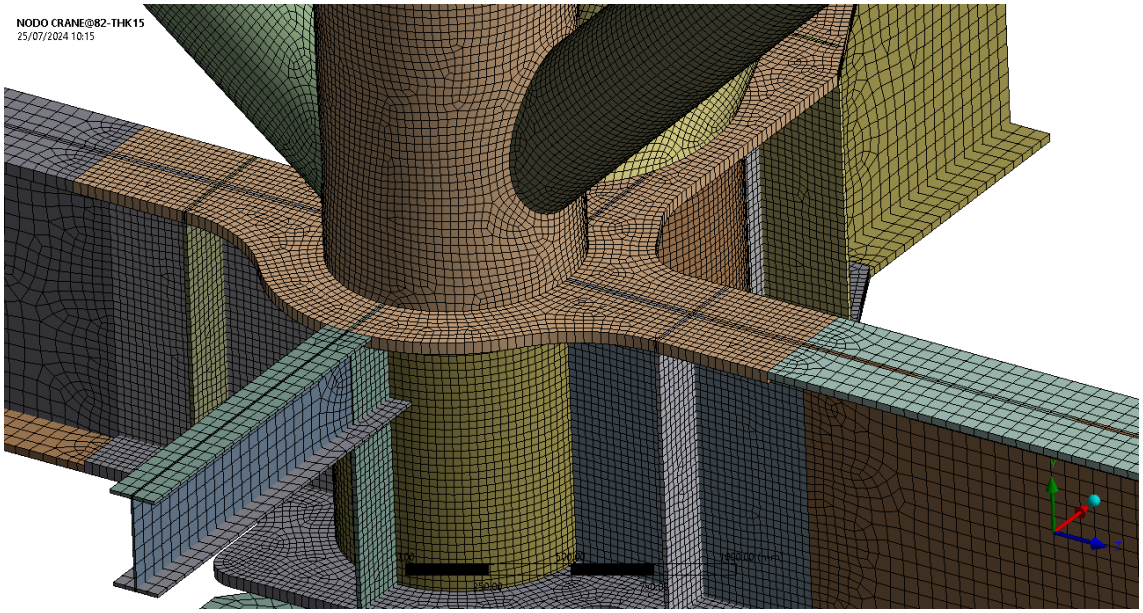


Figure 12.8 – Joint TU03 FE Model mesh – front detail view

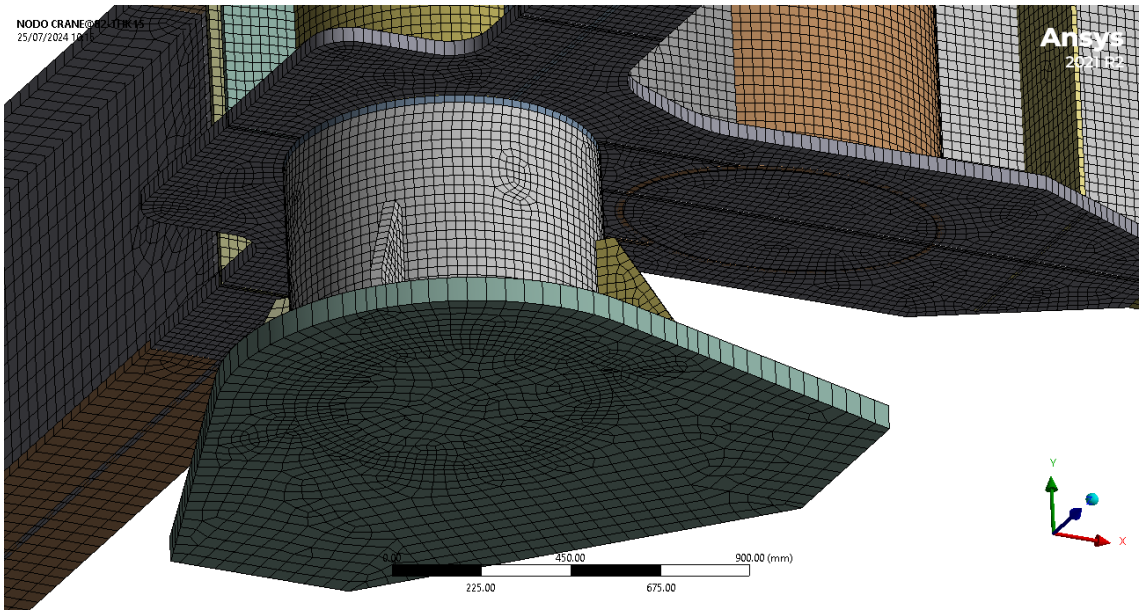


Figure 12.9 – Joint TU03 FE Model mesh – bottom detail view

12.1.4 Constraints

External constraints are applied on the edges of the modelled members that converge into the node in order to restore the actual boundary conditions of the node. Applied constraints are displayed in Figure 11.10.

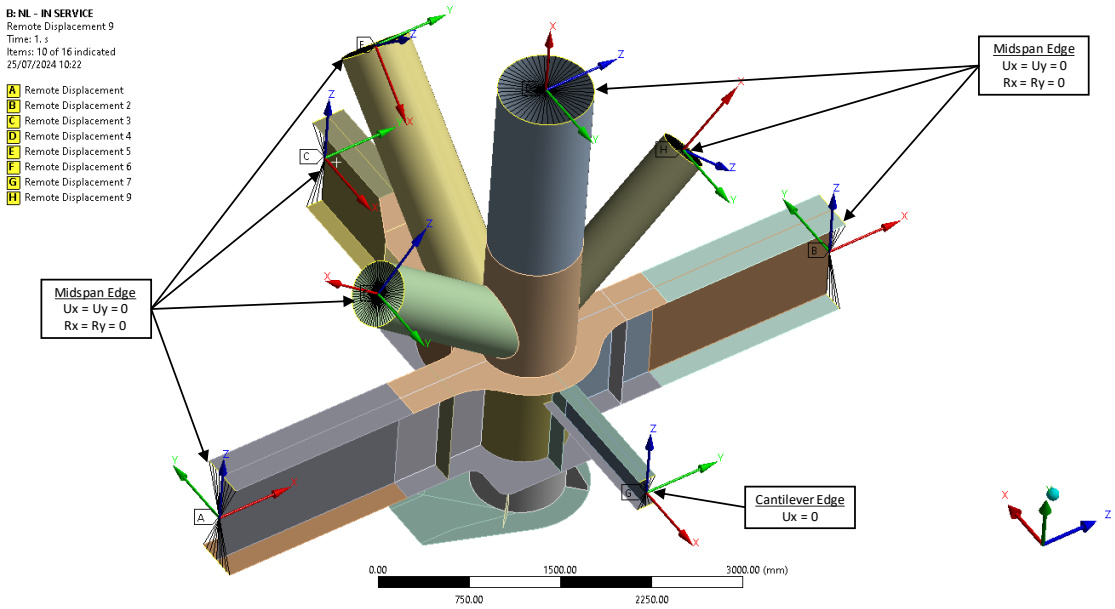


Figure 12.10 – Joint TU02 FE Model – external constraints

12.1.5 Loads

The worst load conditions for this joint are the in-service, when the crane is operating; hence the joint is analysed for these conditions.

From the in-service analysis report (Ref./24/) the maximum ULS loads acting on SACS joint TU03 are extracted and reported in the following table.

	MEMBER	MEMBER END	GROUP	LOAD CASE	Fx [kN]	Fy [kN]	Fz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
ALL LC	TA44-TU03	TA44	TPS	E44F	**-1672**	2267	2777	30	-2952	-2986
	TA44-TU03	TA44	TPS	O12B	**-10312**	1772	3173	135	-3571	-4397
	TA44-TU03	TA44	TPS	E33E	-5038	**3177**	2395	-45	-2556	-4301
	TA44-TU03	TA44	TPS	E44A	-4642	**-613**	1677	163	-2200	**-1226**
	TA44-TU03	TA44	TPS	O22G	-7389	2047	**4074**	167	**-4561**	-4195
	TA44-TU03	TA44	TPS	E44C	-4514	740	**273**	-14	-1030	-2080
	TA44-TU03	TA44	TPS	E44G	-2234	1213	3170	126	-3308	-2331
	TA44-TU03	TA44	TPS	E44E	-2106	2566	1765	-51	-2138	-3184
	TA44-TU03	TA44	TPS	O12F	-8666	2694	3890	88	-4127	-4884
	TA44-TU03	TA44	TPS	E34B	-6328	-127	682	70	-1230	-1780
	TA44-TU03	TA44	TPS	E23H	-5612	431	3538	**223**	-3804	-2393
	TA44-TU03	TA44	TPS	E34D	-4535	2120	744	**-82**	-1186	-3164
	TA44-TU03	TA44	TPS	E34C	-5765	927	288	-26	**-875**	-2436
	TA44-TU03	TA44	TPS	O12E	-8859	2822	3513	43	-3777	**-4968**
	TA44-TU03	TA44	TPS	E44B	-5076	-314	666	82	-1386	-1425

Table 12.3 – Member TA44-TU03 loads – In-Service condition

The max. envelope of these loads is applied to the FE model of the joint.

Load Condition	Fx	Fy	Fz	Mx	My	Mz
	kN	kN	kN	kNm	kNm	kNm
In-Service	-10312	3177	4074	-223	-4561	4968

Table 12.4 – Joint TU03 FE Model – applied loads

These forces and moments are applied to the remote point properly defined at the CoG of the member section of the connecting member between the joint and the TP, and according to its local reference system, as shown in sec. 12.1.1. The application of these forces and moments is displayed in Figure 12.11.

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

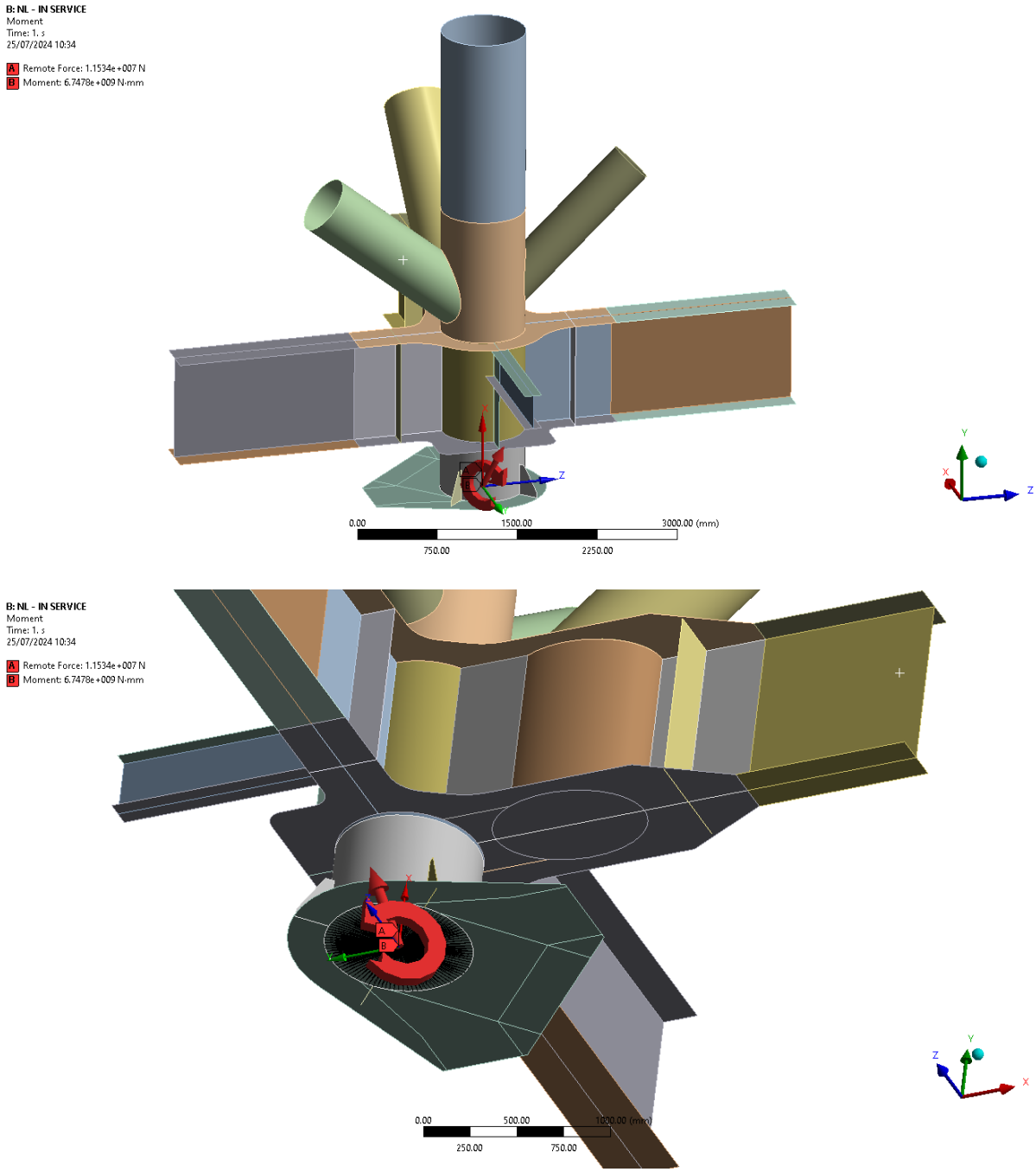


Figure 12.11 – Joint TU03 FE Model – applied loads – In-Service condition

12.2 FEM ANALYSIS AND RESULTS

Joint TU03 is checked for In-Service conditions. It is a critical joint of the lower elevation of the topside structure since it is the connection joint between topside and TP and it is directly connected to the crane.

Loads are taken from the In-Service analysis report (Ref./22/), according to the relative ULS load combinations. Applied loads as envelope of the whole set of combinations for joint TU03 are reported in sec.12.1.5.

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.

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

12.2.1 In-Service Condition

Here are reported FEA results and checks for the In-Service condition.

In the following figures, the Von Mises stresses for the TU03 joint are plotted.

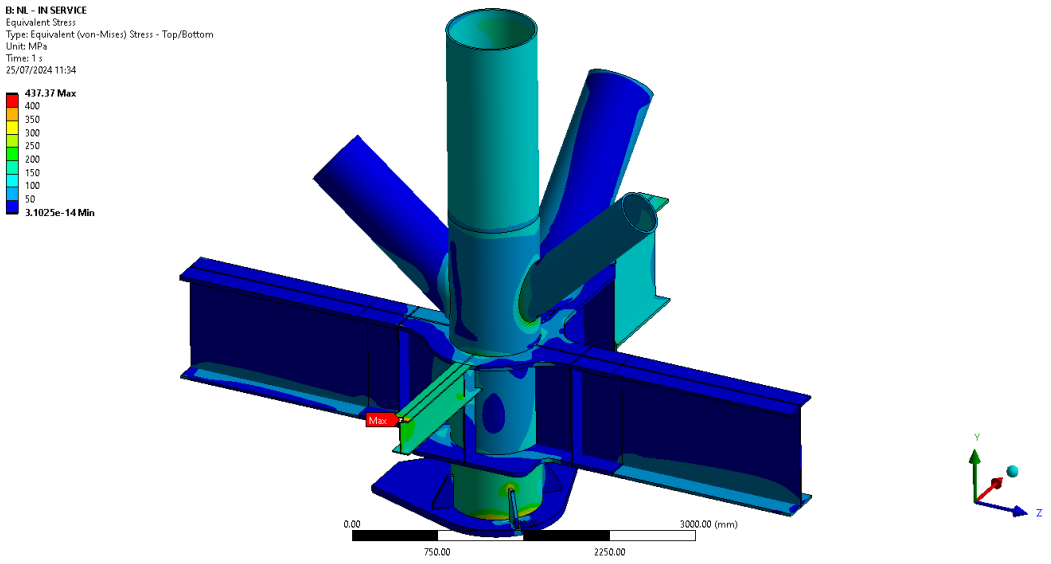


Figure 12.12 – Joint TU03 FE model – VM Stresses (front view)

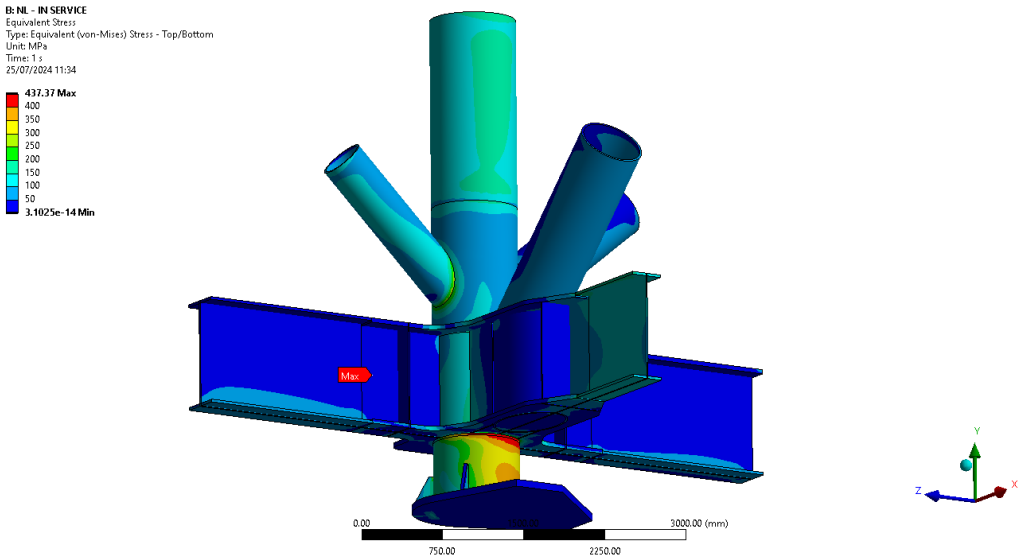


Figure 12.13 – Joint TU03 FE model – VM Stresses (back view)

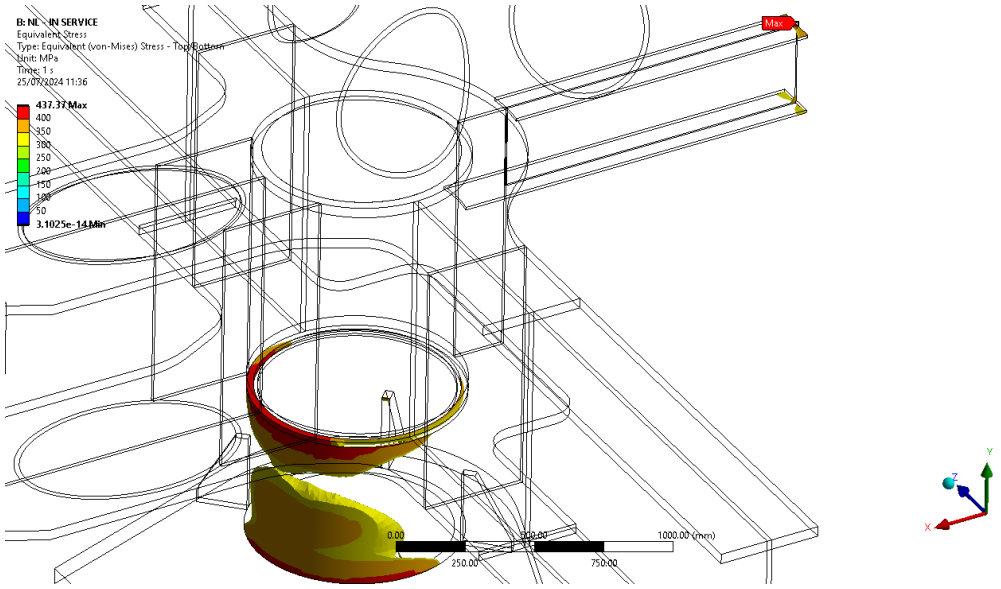


Figure 12.14 – Joint TU03 FE model – VM Stresses > 340 MPa

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:

$\sigma_{adm} = \Phi * F_y = 0.85*400 = 340$ MPa

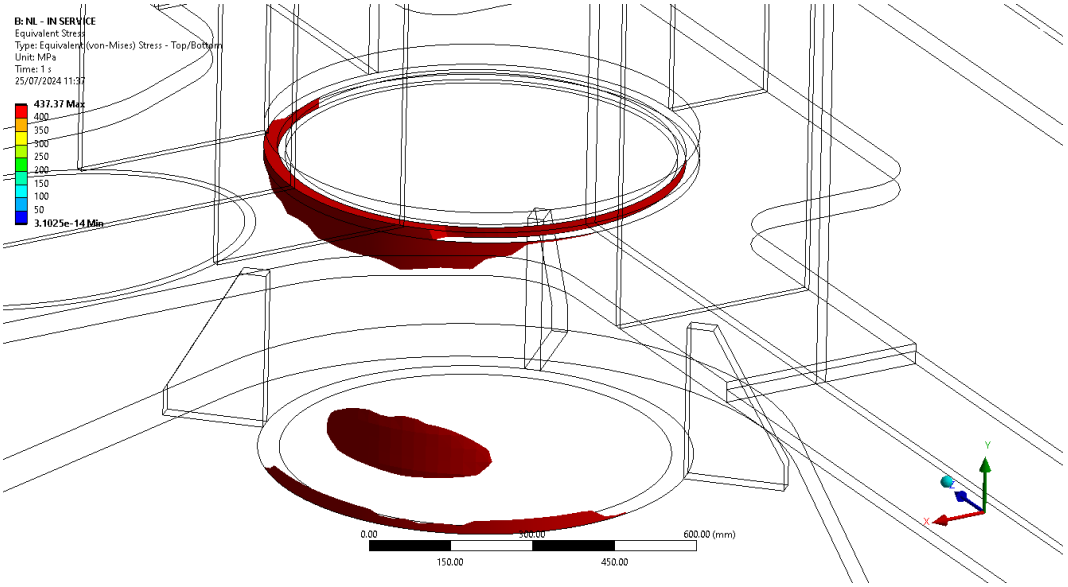


Figure 12.15 – Joint TU03 FE model – VM Stresses > 400 MPa

It can be observed that the VM stresses are largely lower than the admissible stress σ_{adm} = 340 MPa for global checks, and they are exceeded in areas on the vertical Φ800x40 connecting the joint to the TP, at the intersection with the TP 65mm flange and at the upper part welded to the 40mm bottom flange of the joint; the final edge of the IPE330 beam is also interested of some hot spot stress areas. These portions are larger than the limit area 2tx2t on the vertical column, but they do not interest the whole thickness of the plates, only superficial layers are involved; at the edge of the beam, the limit area of 2tx2t is respected. The value of Fy is reached in these areas.

Since the stress checks are not fulfilled due to the extension of the areas, the deformations are checked.

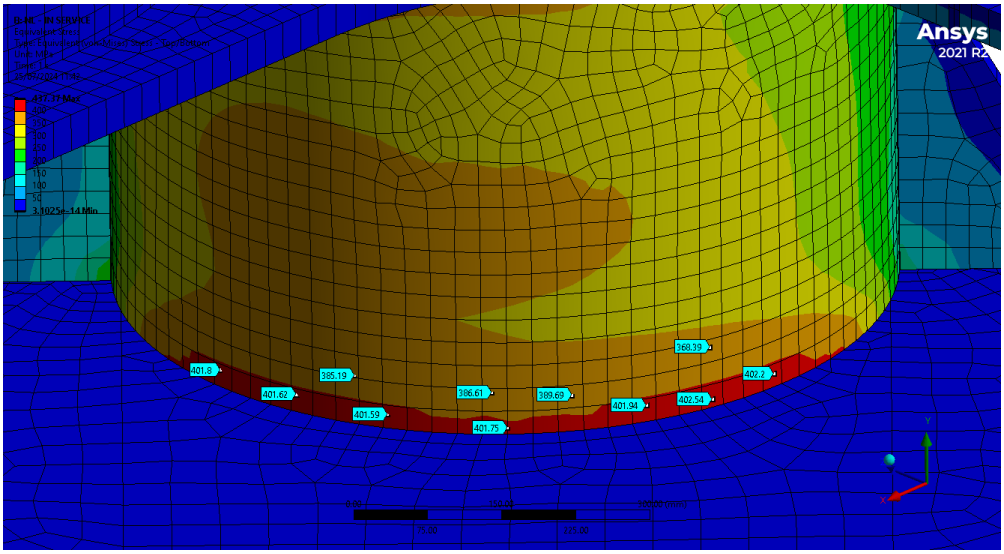


Figure 12.16 – Joint TU03 FE model – Peak VM stresses at hot spot area BOTTOM

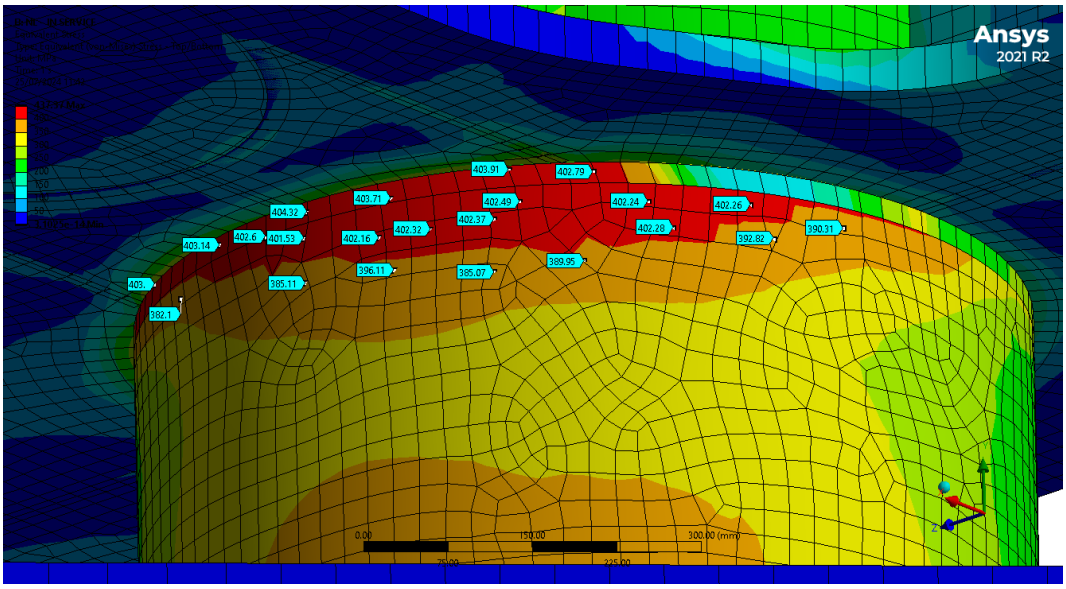


Figure 12.17 – Joint TU03 FE model – Peak VM stresses at hot spot area TOP

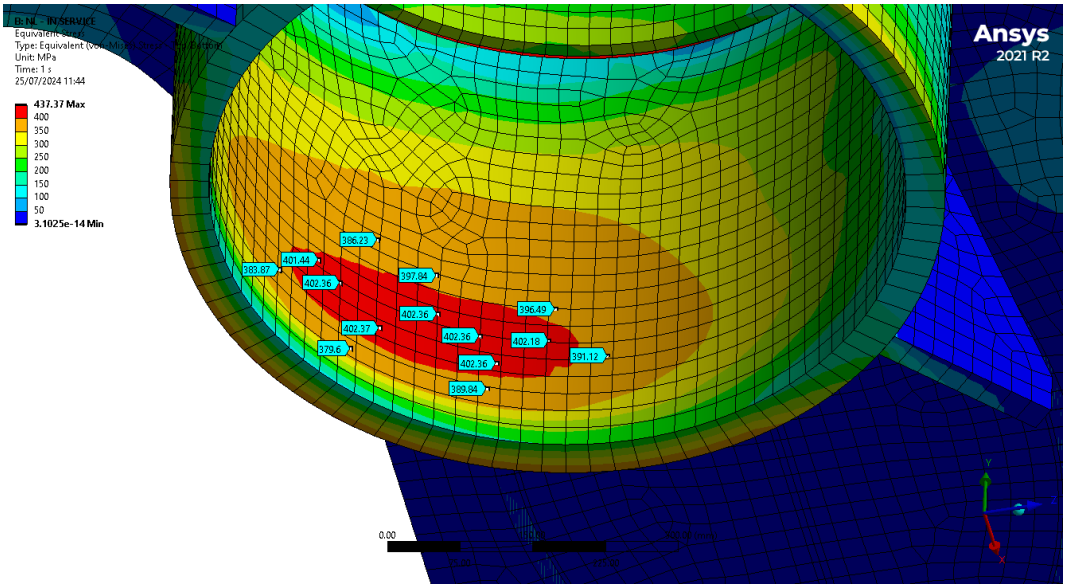


Figure 12.18 – Joint TU03 FE model – Peak VM stresses at hot spot area internal

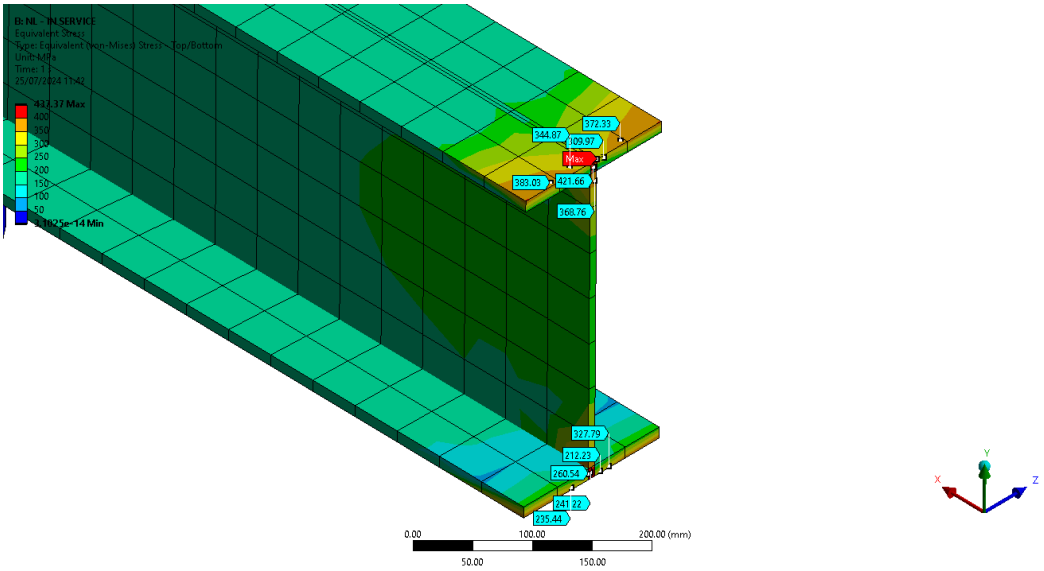
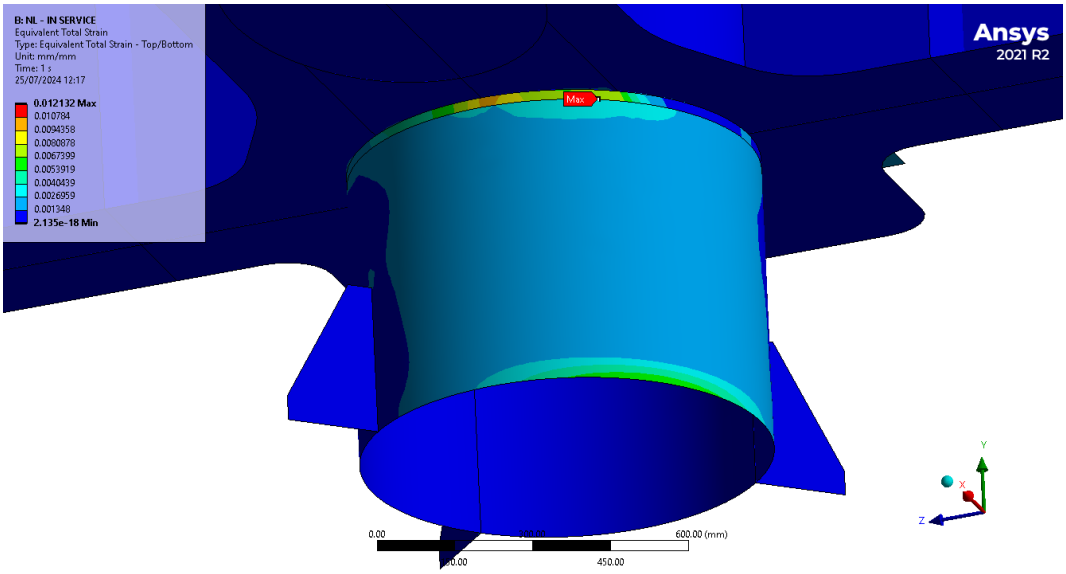


Figure 12.19 – Joint TU03 FE model – Peak VM stresses at IPE330 edge

For the hot spot areas, elasto-plastic strains are measured according to the performed non-linear analysis. They shall be less than 5% in the topside joint and less than 1% in the lower stub and welding, as per design criteria defined in Sec. 8.3: the joint above the 40mm bottom plate is fatigue sensitive during transportation, while the vertical stub Φ800x40 connecting the topside joint to the TP is not fatigue sensitive since the connection is made in place and during transportation the column has a free top edge (no loads).

In the following figures, plastic strains (Figure 10.17) and total strains (Figure 10.18) are plotted.



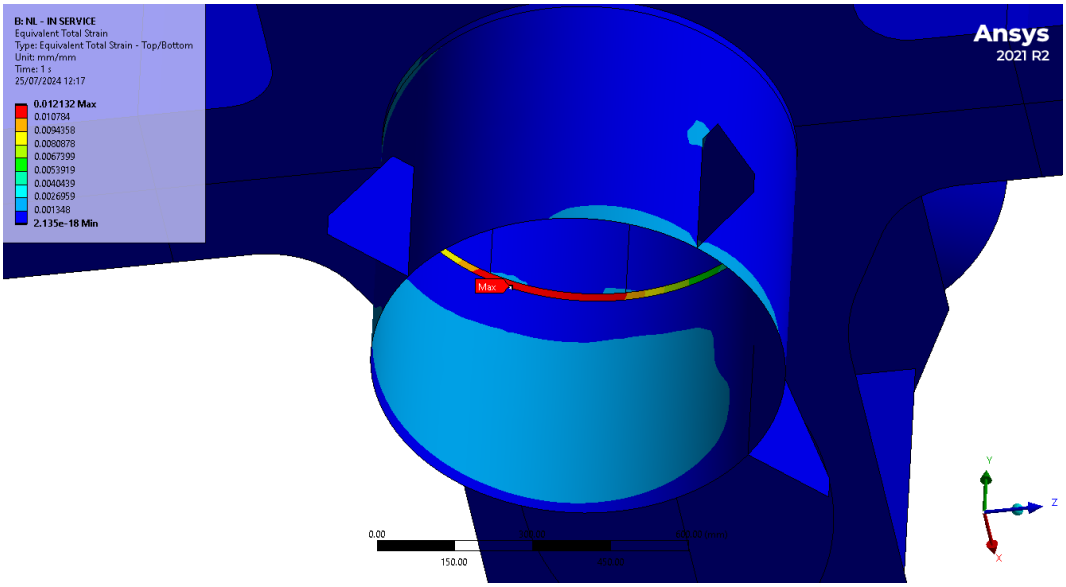
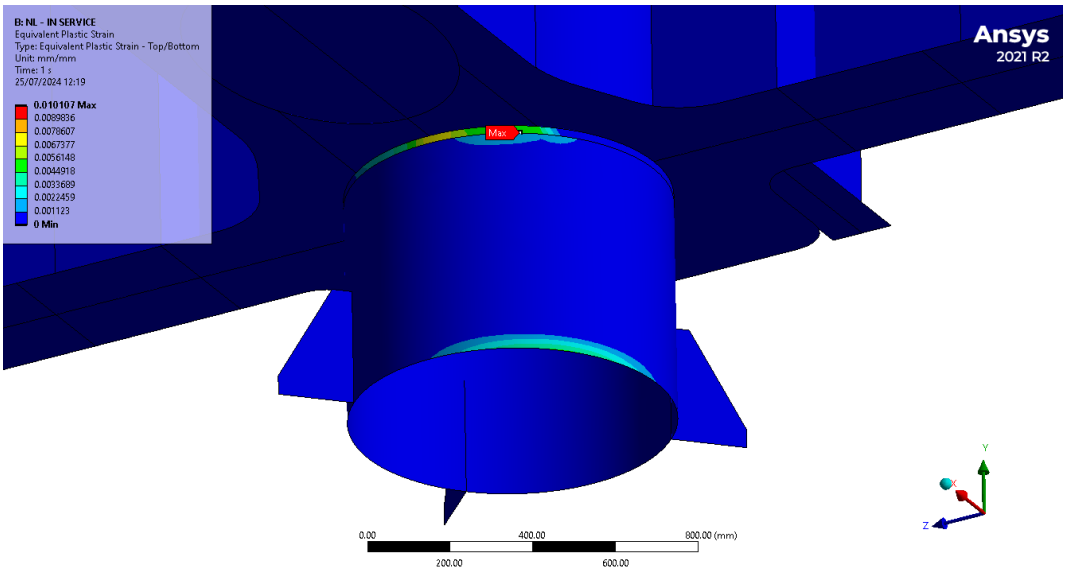


Figure 12.20 – Joint TU03 FE model – Plastic strains



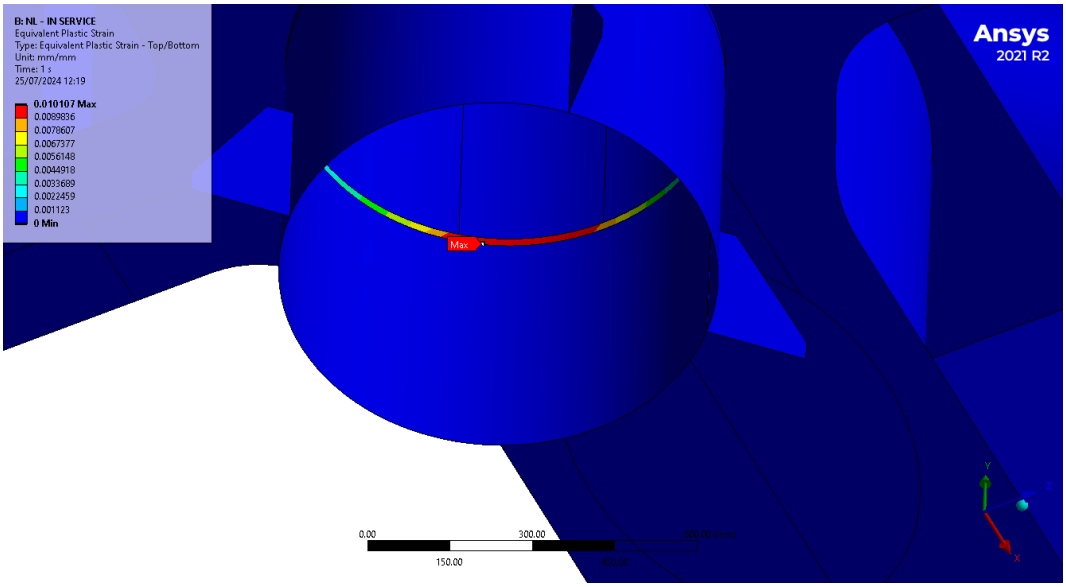


Figure 12.21 – Joint TU03 FE model – Total strains

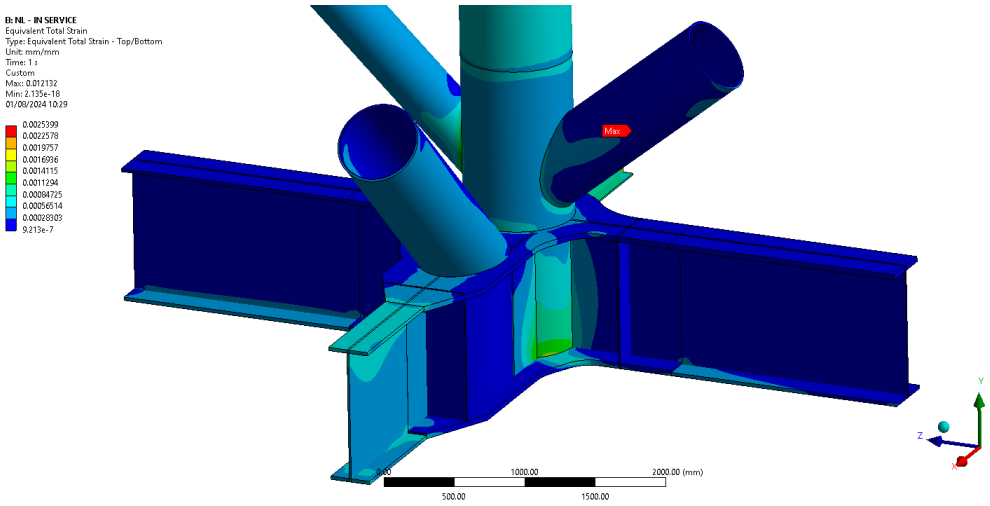


Figure 12.22 – Joint TU03 FE model – Total strains on upper joint (TOPSIDE part)

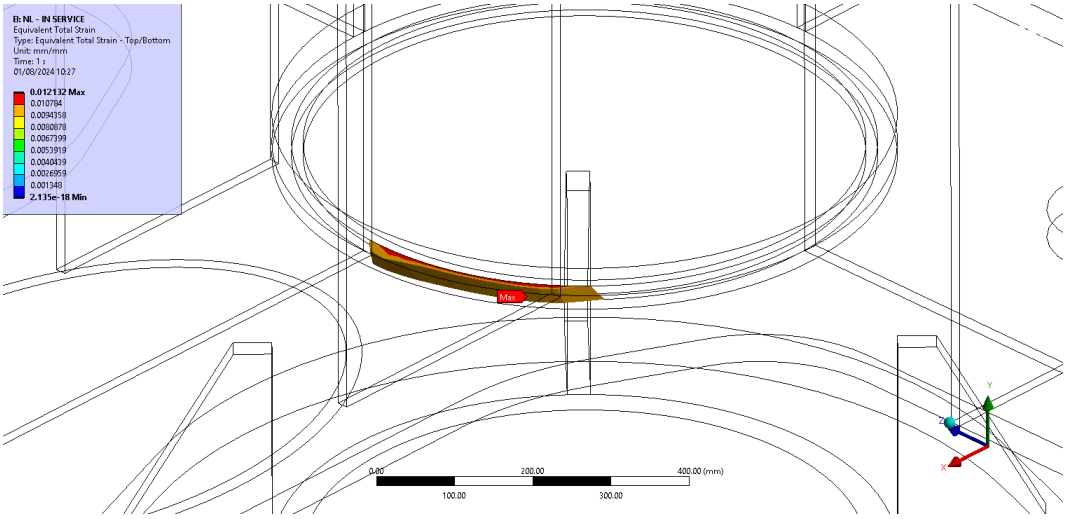


Figure 12.23 – Joint TU03 FE model – Total strains > 1%

It is observed that plastic strains occur in a narrow portion at the intersection between the vertical tubular member and the bottom 40mm plate of the joint, and at the intersection with the 60mm bottom plate.

The max. value of total strains is ε_t = 1.2% < ε_{adm} =5% at the lower stub connection (NOT fatigue sensitive part).

The max. value of total strains is ε_t = 0.3% < ε_{adm} =1% at the upper topside joint (fatigue sensitive part).

Strains are within the admissible value.

Checks are fulfilled, the crane joint has enough resistance capacity to withstand the loads for In-Service conditions.