



Publication Year	2022
Acceptance in OA	2025-03-07T11:22:13Z
Title	Global to Local FEA validation for complex geometries: submodeling technique applied to mechanical structures for ELT class of instrumentation
Authors	CIANNIELLO, Vincenzo, DE CAPRIO, VINCENZO, D'AURIA, Domenico, EREDIA, Christian, CASCONE, Enrico
Publisher's version (DOI)	10.1117/12.2628332
Handle	http://hdl.handle.net/20.500.12386/36481
Serie	PROCEEDINGS OF SPIE
Volume	12184

Global to Local FEA validation for complex geometries: submodeling technique applied to mechanical structures for ELT class of instrumentation

Vincenzo Cianniello^{*a,b}, Vincenzo De Caprio^a, Domenico D'Auria^a, Christian Eredia^a, Enrico Cascone^a

^aIstituto Nazionale di Astrofisica (INAF) – Osservatorio Astronomico di Capodimonte, Salita Moiarriello 16, 80131 Napoli, Italy; ^bUniversità degli Studi di Napoli Federico II – Dipartimento di Ingegneria Industriale (DII), Via Claudio 21, 80125 Napoli, Italy

ABSTRACT

The following paper illustrates the submodeling technique applied to mechanical structures, that can be considered a trade-off between the burdensome, under a computational point of view, fully 3D models and the simpler defeatured model obtained from lines and surfaces. This approach allows to have results that are compliant to the actual behaviour of the real object and on the other hand a quick response by the calculation and analysis software. The case study refers to a main support structure of an optical instrument, and focuses in particular on the central joint between the main parts composing the full structure. The way to model truss-shaped structure is investigated both from a geometric point of view and with boundary conditions. The nomenclature and materials mostly refer to the ANSYS Workbench (R) software.

Keywords: ESO, ELT, MAORY, MORFEO, AO, FEM, FEA, CAD, CAE, Mechanical Structure, Submodeling.

*vincenzo.cianniello@inaf.it; phone +39 081 5575 111

1. INTRODUCTION

The Extremely Large Telescope (ELT) is an optical/near-infrared telescope. The telescope works with a system of 5 mirrors and incorporates others for the adaptive optics. Its main characteristic is its 39-meter primary mirror. Since it is not possible to build such a big mirror considering the current technological capabilities, it consists of 798 hexagonal active segments, each 1.4 meters wide.

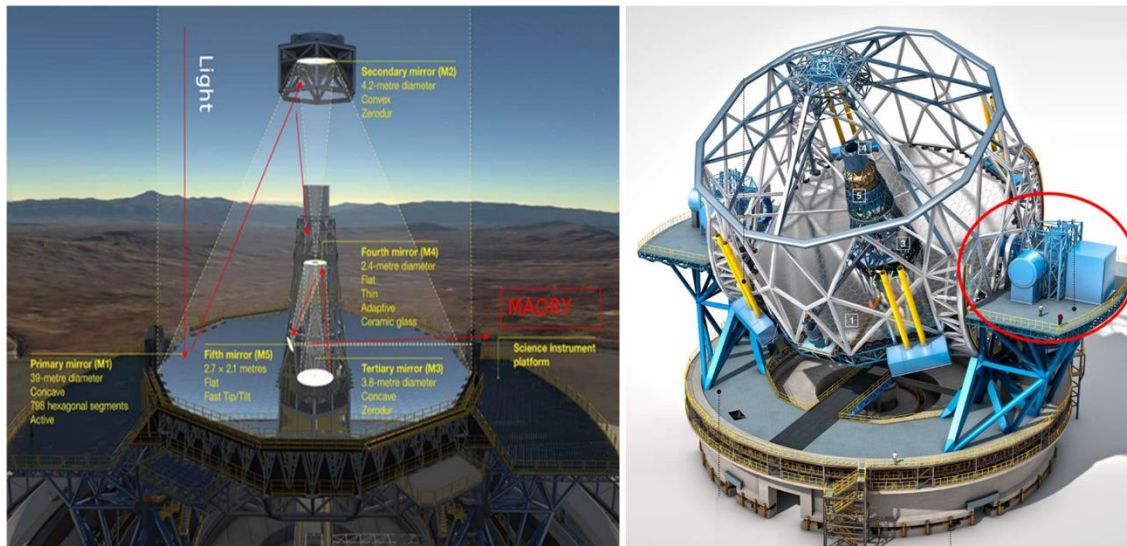


Figure 1. Layout of main optical path (left) and mechanical structure of ELT (right).

Due to its dimensions, the ELT will be the largest ground based optical telescope of the world, allowing to have high performances under a resolution point of view. This challenge found feedback in the high scientific level instruments of which it is provided. One of these instruments is MORFEO, formerly known as MAORY[1][2][3][4]. MORFEO belongs to "first light" class of instruments of ELT. It will be installed on the Nasmyth Platform. MORFEO will foresee a second port for a second instrument, beyond MICADO, still to be determined.

MORFEO is an Adaptive Optics (AO) module for ELT. The adaptive optics are all the technologies that reduce the distortion of the atmospheric turbulence using deformable mirrors. MORFEO offers two AO modes: multi-conjugate adaptive optics (MCAO), to achieve uniform adaptive optics compensation over the full MICADO field of view, and single-conjugate adaptive optics (SCAO), for peak performance over a smaller field of view. MICADO is a near-infrared camera and spectrograph.

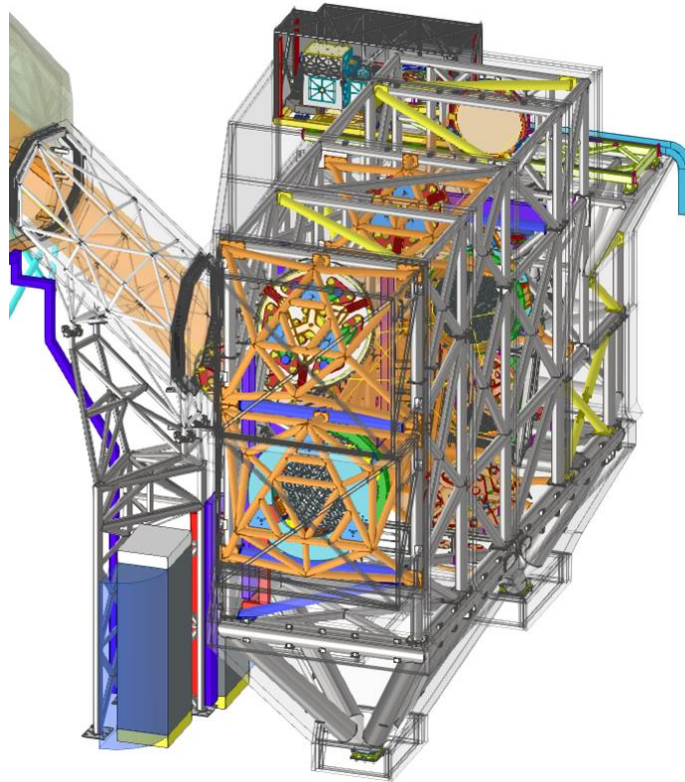


Figure 2. MORFEO Main Structure general Assembly.

The design of MORFEO project is in charge of the MORFEO Consortium, which includes the Istituto Nazionale di Astrofisica (INAF), the Institut de Planétologie et d'Astrophysique de Grenoble (IPAG) and the National University of Ireland, Galway (NUI-Galway). The European Southern Observatory is also involved in the development of the whole instrument.

The final mechanical design of the MORFEO MSS for the PDR was finalized in April 2021 (1Q21). Further details about the mechanical design are reported in [5][7].

To give an overview on this work, a quick shot about the paper is given. The paper is structured as follows: the next chapter is an overview of MORFEO, specially focusing on the description of the Main Support Structure (MSS), the optical bench of the instrument. So the methodology for submodeling is outlined. Then the hypotheses for FEA are listed and the results outcoming from the global FEA validation are shown.

The analyses carried out are focusing on the worst case come out as feedback of the FE model at global level. As further specified in the next paragraphs the definition of "worst case" is strictly related to the mechanical element under study. In particular, the Central Joint case has been taken under examination. This research path is developed in order to demonstrate compliance with the requirements provided by ESO.

2. MODELING AND SUBMODELING STRATEGY

The mechanical analyses carried out for the whole instrument are based on the following assumptions, in order to perform the engineering calculations through the FEA approach:

- a. The global reference coordinate system is directly imported from CAD to FE software. It is aligned with the MORFEO standard coordinate system (MORFEO C-SYS). Of course, all the results are carried out in this coordinate system. To have a reference, Z is perpendicular to Nasmyth platform against gravity, and the other axes are aligned to the NP axes.
- b. The strategy in order to evaluate the earthquake effects is performed like a quasi-static analysis with particularized set of accelerations.
- c. The results of all the analyses with their measure units (SI) are:
 - Von-Mises stresses [MPa]
 - Total deformations [mm]
 - The rotations [°]
 - The reaction forces [N]
 - The reaction moments [N*mm]

To carry out the structural analysis and in order to have a lighter models and shorter computational times, simpler FE models than the 3D CADs have been developed. All the solid parts composing the actual objects are substituted by lines, surfaces and point according to their shape. In this way, we have virtual FE objects with a comparable behaviour to the realistic trusses, joints or panels, but a very light and flexible model to perform computes with. In these modified model, each 1-D beam element is aligned with the geometrical axis of the solid (3-D) CAD beam. In the same way the plane elements also are placed in order to have the 2-D surface on the middle symmetrical plane, referring to the thickness, of the solid (3-D) CAD plate. Another family of 1-D constraint elements are also used to simulate rigid objects to connect the different parts each other or to transfer the optomechanical payload (or other kind of payloads) to the support structure.

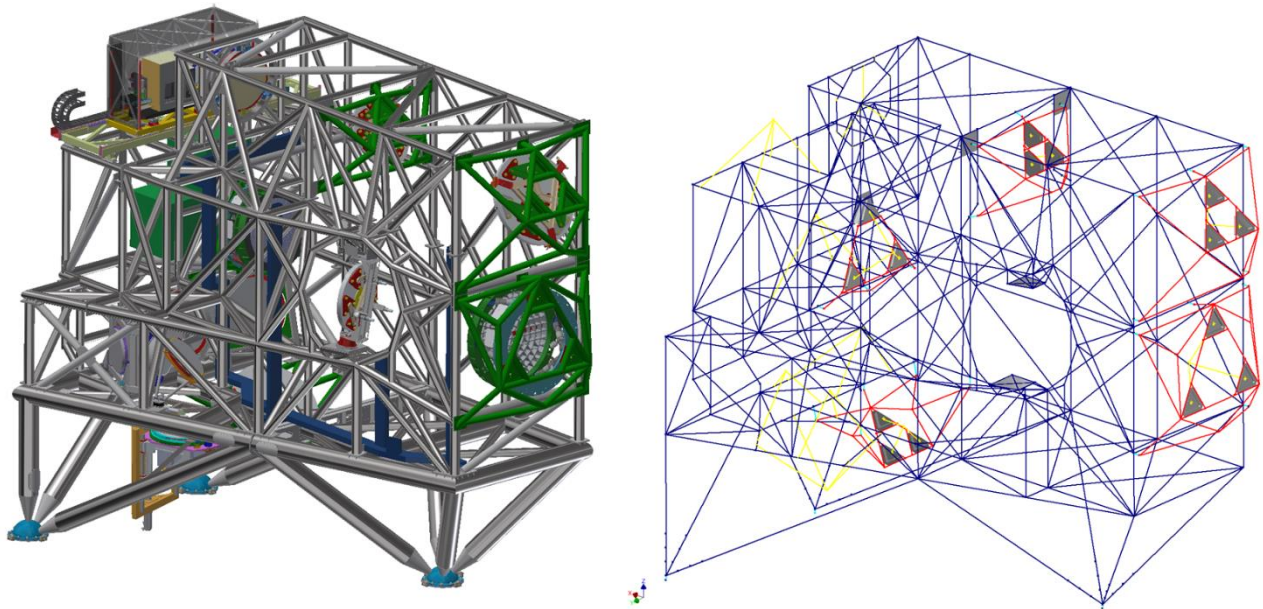


Figure 3. Comparison between a 3D CAD model (left) and a CAD model for the FE analysis (right).

In order to simulate plate and beam elements in the FE model, the CAD model used for pre-processor FEA is made of surfaces, 3D lines, and points; the missing geometrical information, like thickness, have been assigned in Ansys Mechanical afterward.

Anyway, for local analysis, due to the complexity of the geometry under study, some detailed analyses of 3-D submodeling have been carried out. For local analysis (submodeling verifications) some parts/assy of the whole structure are simulated as 3-D components. They are usually parts of a more complex geometry where the differences between the 3-D and the 1-D/2-D models previously described are very significant. In fact, in this case the CAD model is slightly modified, as usual of CAD model for FEA, to simulate the local submodeling in all its boundary conditions, directly imported from the global (1-D/2-D) FE model. For example, the screws of a junction can be substituted by small cylinders that emulate them in the behaviour, but do not have all the geometrical details of an actual screw. The 3D model of each item under study was exported directly from the CAD software (Autodesk Inventor) to Ansys WB 2020 R1 CAE software.

In the global FE model of the MSS, the structural elements are modelled as BEAM188 (1D) or as SHELL181 elements (2D), as well as in the local FE model are used the SHELL181 elements for all structural elements except to screws and bolts that are modelled as BEAM188.

2.1 MORFEO MSS design description

The Main Support Structure (Figure 4) is made of more separate parts, both for computational and for AIV reasons and jointed via some interface flange. The MSS is made of different welded sub-assy then bolted to make the complete structure. Other important parts of the MSS are the OSS (Optomechanical support structure), which are the holders for all the optomechanical elements. Moreover, on the MSS are also installed: LGS, MORFEO Calibration Unit [18], Deformable Mirrors, MORFEO Thermal Enclosure [12], electronic cabling [16][17][19][20], thermal routing and other support structures provided by PF0 [11].

The MSS is provided with 3 mounting flanges to install the structure directly on the Nasmyth Platform. All the parts which compose the MSS comply with the same design philosophy: they are truss lattice structure shaped in order to optimize the global mass of the structures.

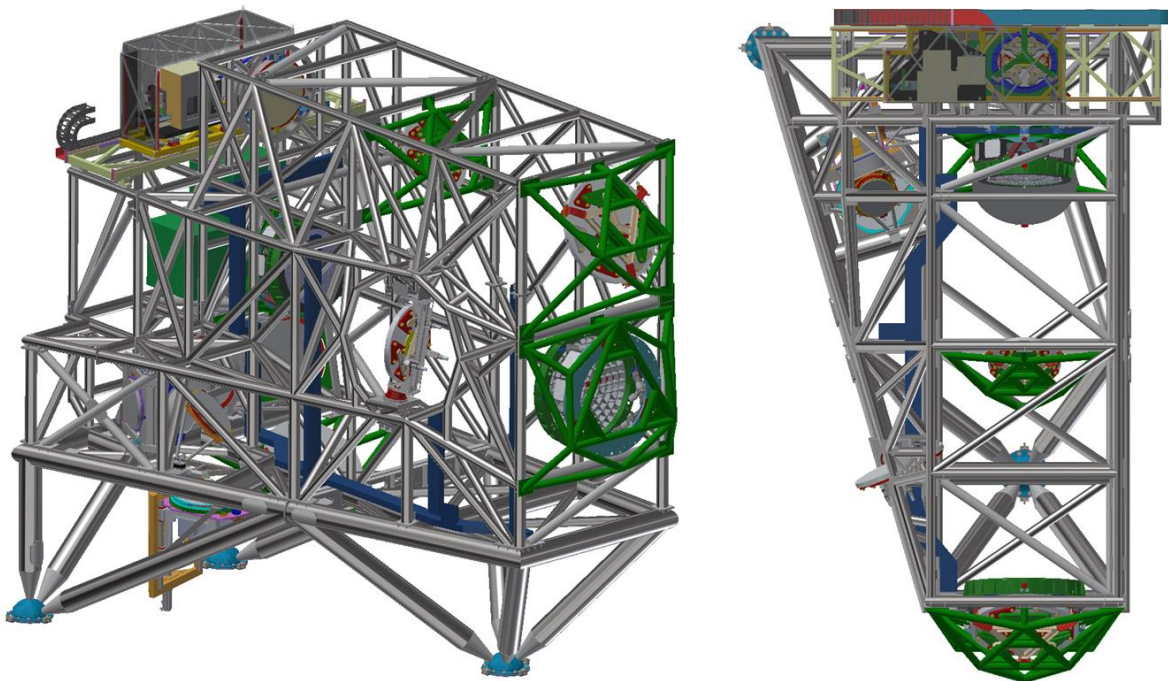


Figure 4. 3D CAD model of MORFEO MSS: lateral view (left) upper view (right).

In order to optimize the ratio between the global stiffness and the global mass of the MORFEO MSS, the circular tube profiles have been chosen and the structure is made of truss-beam shaped elements [6]. In fact, the circular pipe profiles optimize the inertia and minimize the mass compared to other types of common profiles. Because the MSS is a high weight support structure, it was verified that using different profiles (for example square or rectangular) the increase of total mass is significant. In the same way, even to contain the global mass of the system, it was decided to use different profiles for

the main support structure. The whole assembly is made of standard profiles in ANSYS Structural Steel (see [7] article analyses for the physical properties of used common materials in the FE analyses).

The CAD model used for FEA at global level is represented in Figure 5.

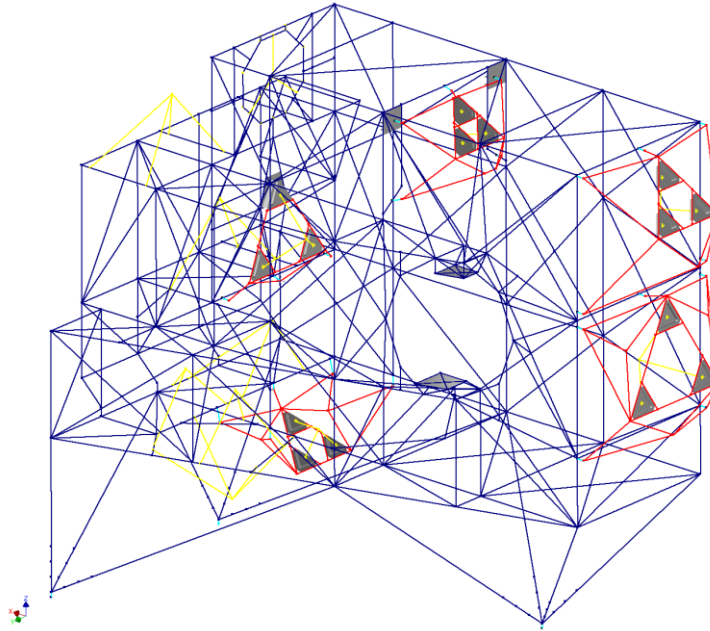


Figure 5. CAD model for FEA of MORFEO_MSS_Type_6.

Taking in consideration the complexity of the object under study, the finite elements are represented and highlighted with different colors in the Figure 19:

- Line elements in **blue** are used for the **MSS profiles**,
- Line elements in **red** are used for the **OSS profile**,
- Surface elements in **light grey** are used in the **OSS** and to **support M11**,
- Line elements in **pale blue** are used to connect the different parts of the MSS,
- Line elements in **yellow** are used **to transfer the load** from mass points to nodes linked to the MSS or the OSS,
- Mass elements, not highlighted, are used to simulate the CoG mass of optical elements. In fact, they are practically the vertex of the yellow line elements illustrated above.

3. CASE STUDY: CENTRAL JOINT

The submodeling analysis is centered on the local structural validation of one of the bolted joints of the Main Support Structure. In particular, the object under analysis in this paper is the **central joint between the MSS structures “A1” and the “A2”**, respectively the main parts composing the bottom part/first floor of the MSS, with two main support points and one main support point. The mechanical design in the figure below is illustrated.

As illustrated in the next picture but better visible in the CAD model of the central joint the geometry is very complex. In order to give an accurate overview on the geometry the main characteristics are here listed:

- **8 beams for each main part** of the MSS structures converge in the interface flat beams (2) for a total of 18.
- **Other 10 beams converge** in this joint from the above elements.

- **The profiles used are very different** under dimensions, inertia sections and thicknesses.
- **The two main parts** are jointed via bolts.
- **The difference** between a simplified model and a detailed model **are huge**.

For this reason, is necessary to perform a submodeling analysis, in order to estimate properly the actual behaviour of the central joint of the MSS.

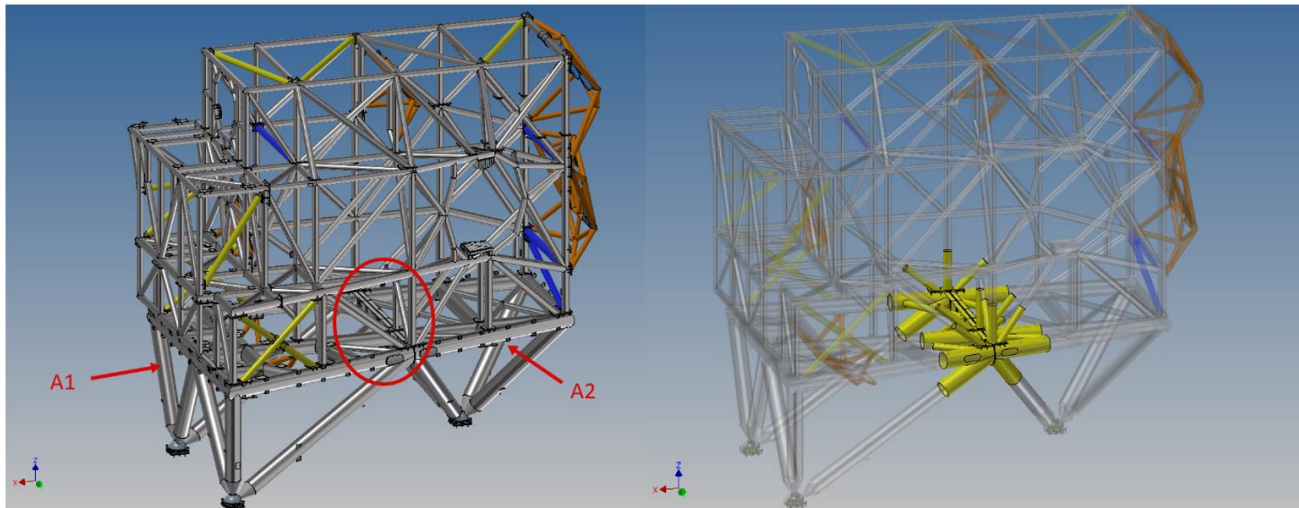


Figure 6. Central joint of the MSS in the red circle (left) and highlighted alone (right).

In order to verify the central joint as 3-D submodeling, a very large part of the Main Support Structure has been taken into account, highlighted in yellow in the figure below (see Figure 6). Since the thickness of the tubes is very small relatively to the other 2 dimensions, a **surface model** was chosen for the structural analysis.

The 2-D surface CAD model, derived from 3-D model using the relatively middle surface, with all its particulars, is shown in the figure below (Figure 7). The surfaces of the model have been discretized and using 2-D elements SHELL181, while the bolts are modelled as beam elements using BEAM188.

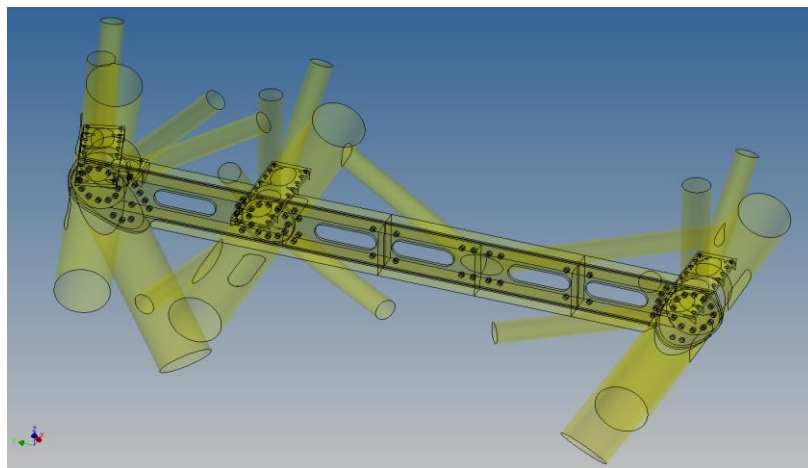


Figure 7. CAD model of the central joint on the MSS, particular

The element size imposed to mesh the model is **16 mm**. A detailed of the central joint meshed is shown in the figure below (Figure 8).

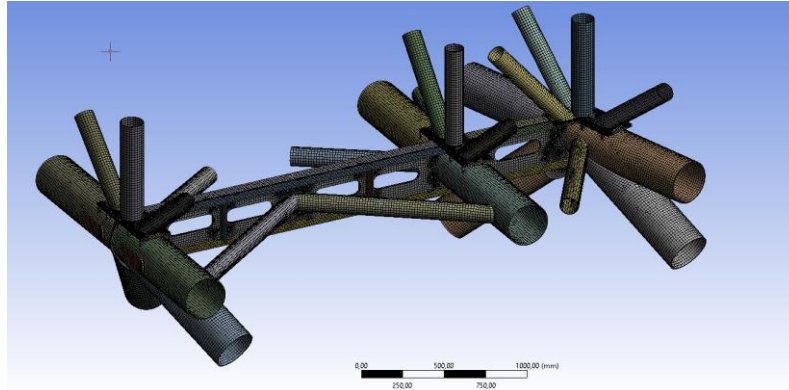


Figure 8. Meshed central joint FE model of the MSS

The loads were imported directly from the global FE model of the MSS through the technique of the sub-modeling. They have been created cut zones in order to import the boundary conditions of the global model. In the following picture (Figure 9) these boundary conditions can be seen. Displacements are automatically imported from the nearest nodes of the MSS FE global model and imposed at the cut-edges of the sub-model, highlighted in red in the following picture (Figure 9). The sub-model is constrained by those imported remote displacements, while the only other external load is the acceleration field (Figure 10).

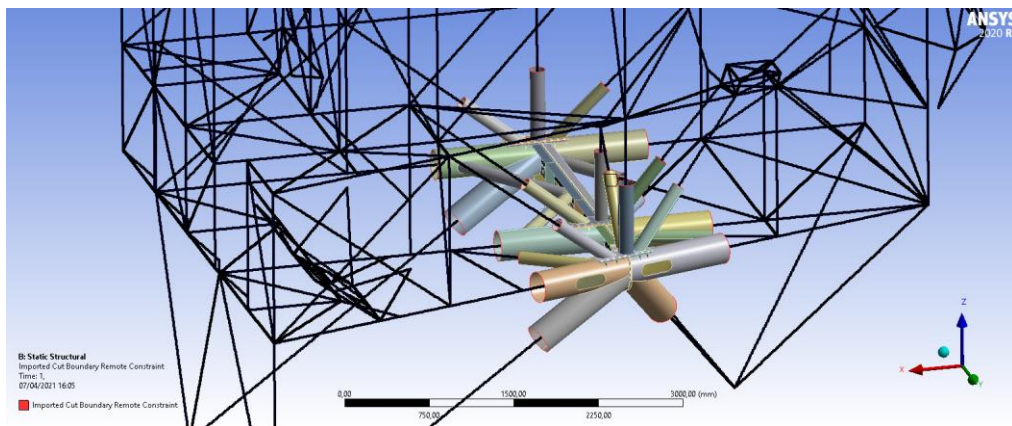


Figure 9. Imported boundary conditions of the MSS

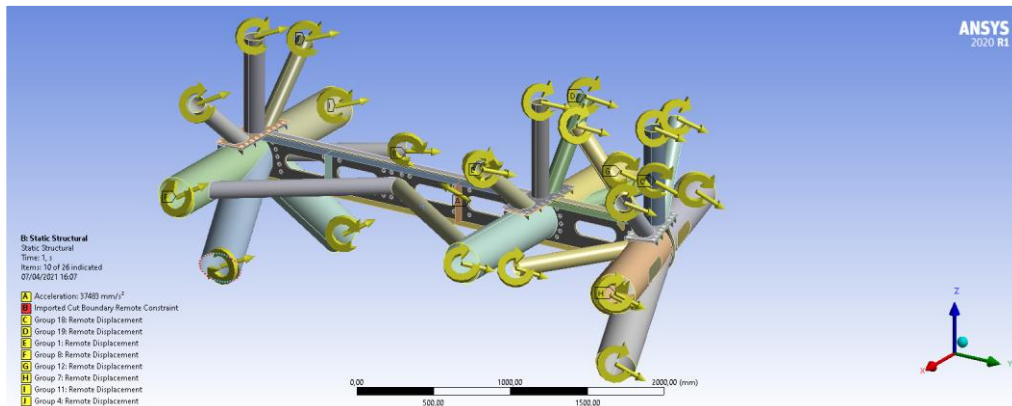


Figure 10. Remote displacements of the sub-model

In order to simulate the earthquake effects, the worst case outcoming from the global model, is considered. In this case the imposed acceleration field is illustrated in the following table and represents the Load Case 13 of simplified “quasi-static” earthquake analysis for a validation of a main support structure installed on the Nasmyth Platform.

The choice was made since it is the hardest load case for the central joint.

Table 1. Earthquake Accelerations Load case 13 for the central joint.

X-direction [mm/s ²]	Y-direction [mm/s ²]	Z-direction [mm/s ²]
9708.5	-32361.8	16230

In order to simulate the behaviour of the bolts, non-linear contact between the bolted surfaces has been used. The surfaces are free of sliding and separating, but no penetration is allowed. The beams which represent bolts are fixed-joined to the surfaces. The cross sections of the bolts are circular sections with a diameter of 16 mm and 12 mm, because M16 and M12 bolts have been used in the CAD model.

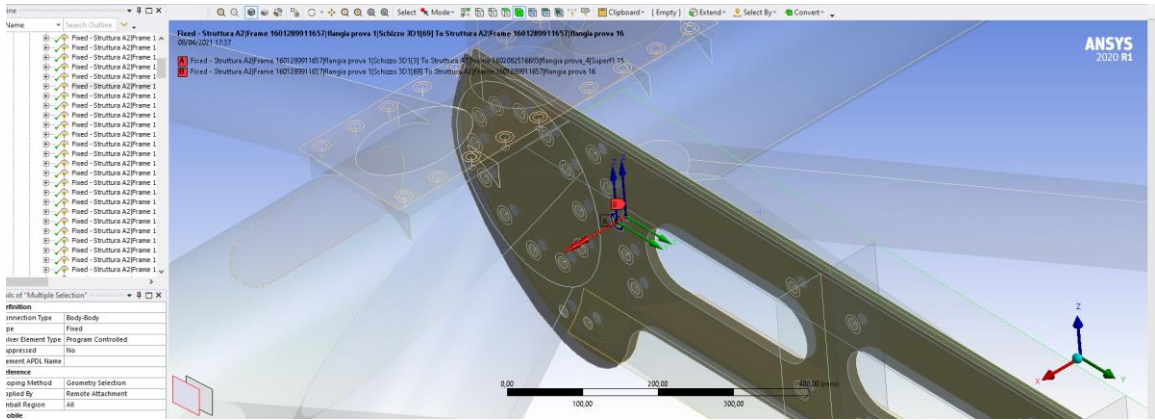


Figure 11. Bolts detail of the sub-model

4. FEA RESULTS

This paragraph is the core of the study. The results presented underline the importance to improve a global analysis with submodeling. In first instance, the results of the sub-model analysis in terms of displacements and stresses were compared to the results of the entire model in order to verify the correct importing of the boundary displacement conditions and the general behaviour of the submodeling too (Figure 12). As it can be seen, there is a very good match between the displacements field of the sub-model and the one of the global FE model of the MSS, being the shape of the contours similar for both analyses. Despite the different scales (due to the fact that the global analyses took in account also parts and elements of the MSS that are not present in the local FEA of the Central Joint), this results highlight that the problem is well placed and the feedback of the analysis is reliable. Similar considerations can be made for the equivalent maximum von Mises stress.

The analysis of the global model is performed with the number of nodes and elements reported in the following table.

Table 2. Global model details of nodes and elements.

Item	N° nodes	N° elements
MORFEO MSS	74764	42442

The information is directly extracted from Ansys software. The size of the single meshed element is almost 17.2 mm and it has been obtained setting a default mesh size. The same numbers for the local model, which represents only a part of the global one, are almost 10 times higher. All this affects the calculation times of the model.

The results of the quasi-static analysis are compared below, under survival conditions (earthquake analysis). The maximum displacement, in both models referring to the same area, is about **4.6 mm** and the maximum value of the Von Mises Equivalent Stress is **240 MPa**.

The maximum deformation is right in the proximity of the boundary conditions import zone. In this way we also verified that the two models were loaded the same way. It is important to note that, the color maps on the global and local models overlap perfectly and that the scales of the two models are nearly aligned. These results, in terms of maximum deformations, are widely acceptable from an engineering point of view.

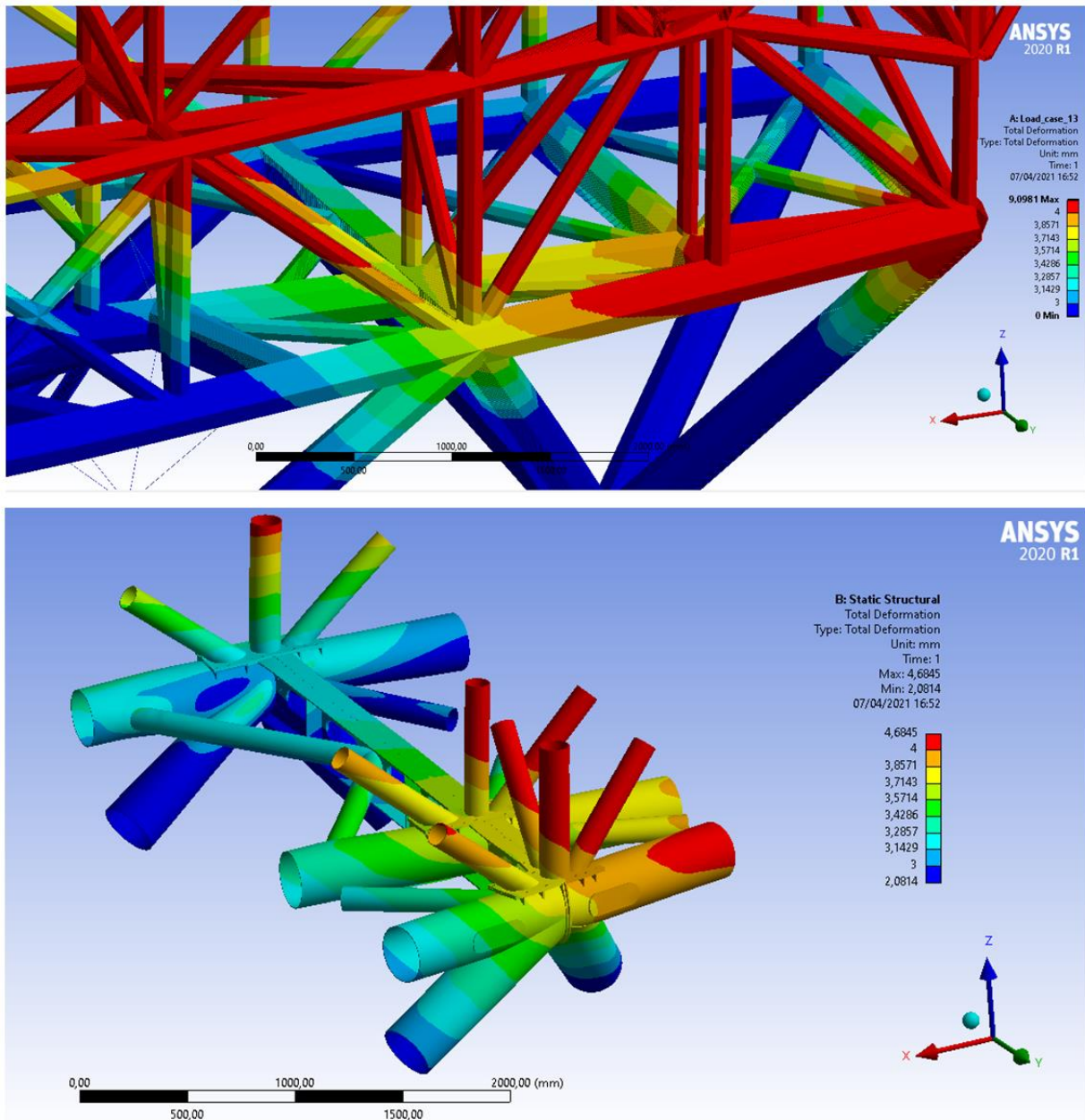


Figure 12. Comparison of maximum displacement between global FE model (upper) and local sub-model (lower).

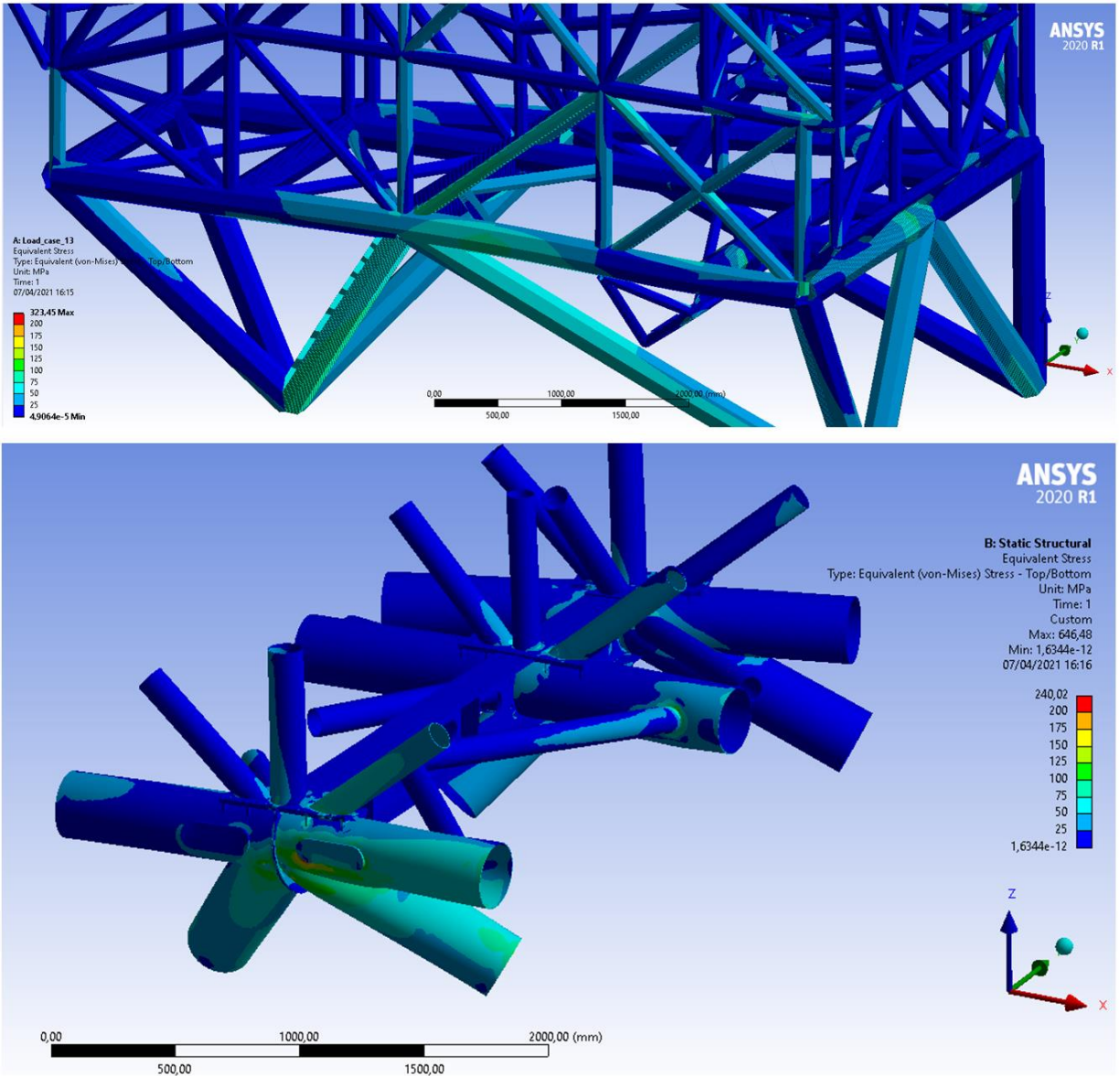


Figure 13. Comparison between Equivalent von Mises Stress between global FE model (upper) and local sub-model (lower).

Also for this case, under a stress point of view, is clear that the values between the global and local model are in line.

According to the stress analysis criteria the compute of the **safety factors** has to be performed basing on the ESO documents. They are needed to impose conservative conditions both for material and design. A verification approach is that the "**Reserve Factor**" **RF** shall be greater than or equal to 1.0:

$$RF = \sigma_m / (\sigma \times SF \times SFM) \geq 1.0$$

where:

1. σ_m yield strength for ductile materials, ultimate strength for brittle materials
2. σ maximum working stress, von Mises stress for ductile stress, maximum principal stress for brittle materials
3. SF Safety Factor, as reported in the following table
4. SFM Material Safety Factor, as reported in the following table

Loading and environmental conditions	Safety Factor (SF)
On Site Assembly, Integration	1.35
Operation, Stand-by and Maintenance	1.35
Survival, Accidental	1.1

Material	Material's Safety Factor (SFM)
Metal	1.1
Optical glass, glass-ceramic	1.4
Glue	1.4
CFRP	1.4
Concrete	1.5
Reinforcement steel	1.15

Figure 14. SF and SFM extracted directly from ESO documents.

In order to give the proper mechanical and physical characteristics to the structure, different materials have been used. The materials chosen are:

- **ANSYS Structural Steel**, for global analysis of the MSS, common materials used in the FE analysis and described in the articles [7][6].
- **S355JR steel**, for the central joint, is made of S355JR steel, that is strongest than the ANSYS Structural steel. 355 in its name refers to the yield stress of the material in MPa.

The **Reserve Factor** (RF) is computed in according to ESO documents:

$$\sigma_{\text{yieldS355JR}} > \sigma_{\text{max}} \rightarrow 355 \text{ MPa} > 240 \text{ MPa} \rightarrow \mathbf{RF} = 355 / (240 * 1.1 * 1.15) = \mathbf{1.17} > \mathbf{1.0}$$

The structural steel used is a **S355JR** (with a minimum yield stress of **355 MPa**).

The FE sub-model also gives results about the bolts that have been modelled as beams, being their behavior assailable to a one dimensional objects. The bolts, in fact, are subjected to strong axial forces while in the other directions the effort is negligible, or, to be more precise, is of another order of magnitude. The maximum value of von Mises equivalent stress acting on the bolts is shown in the next figure. The value is **646.5 MPa**. As previously made, the RF is computed:

$$\sigma_{\text{yieldS355JR}} > \sigma_{\text{max}} \rightarrow 900 \text{ MPa} > 646.5 \text{ MPa} \rightarrow \mathbf{RF} = 900 / (646.5 * 1.1 * 1.15) = \mathbf{1.1} > \mathbf{1.0}$$

This result help in the choice of the bolts. The bolts chosen belonging to the “**10.9**” class, that can be taken as good because of their minimum yield strength is **900 MPa**.

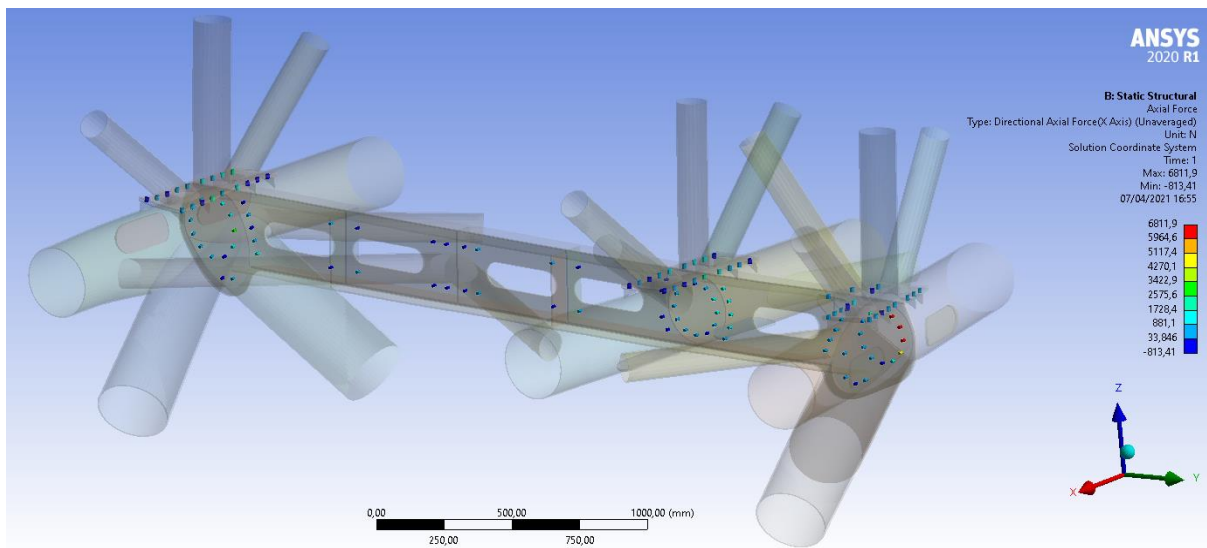


Figure 15. Bolts FEA results of the central joint sub-model, axial force (X Axis).

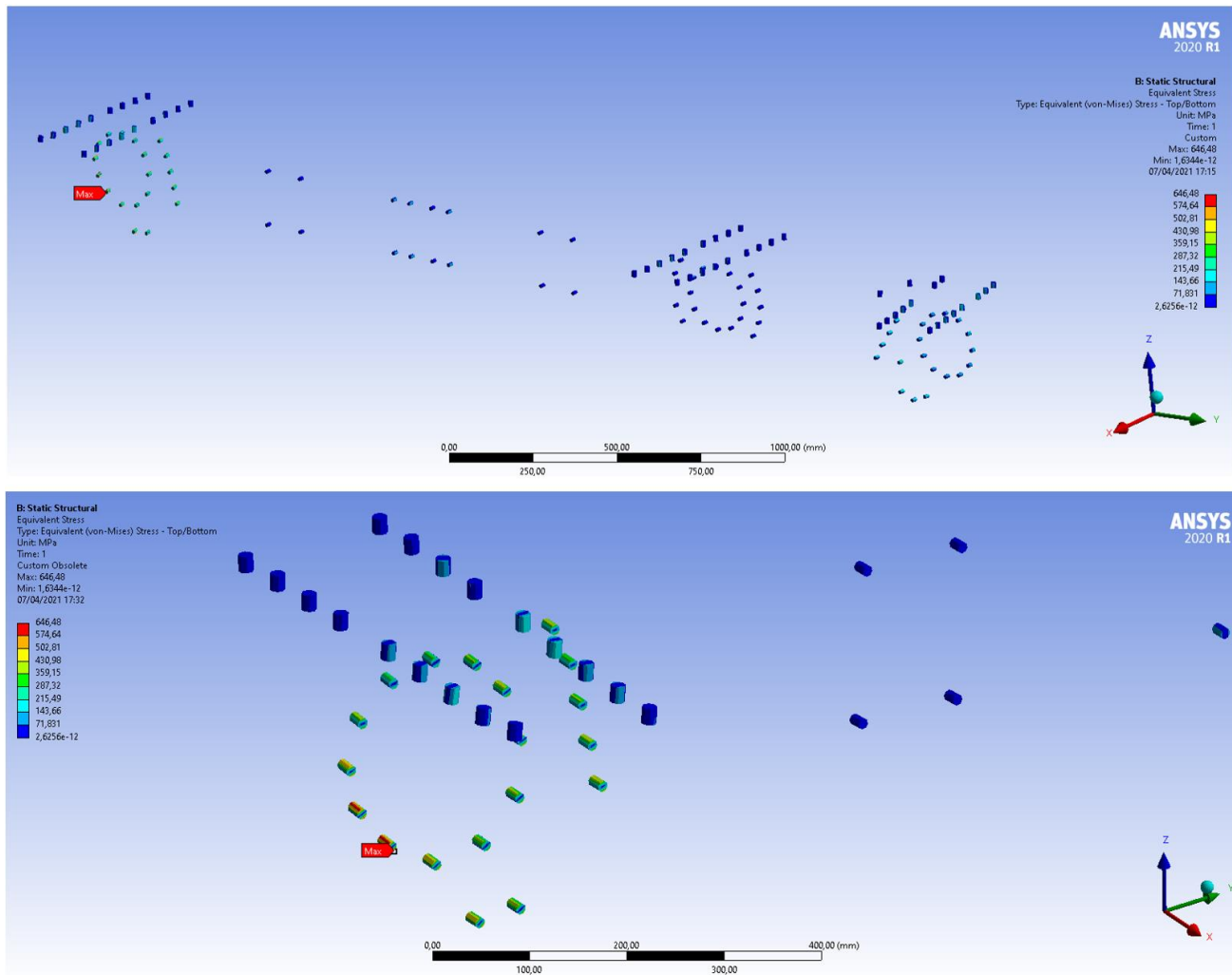


Figure 16. Bolts FEA results: maximum Equivalent stress (upper) and detail maximum Equivalent stress (lower).

5. CONCLUSIONS

The study illustrates the noteworthy results of the analysis performed. It has been decided to report the worst load case as an example, because it is the most representative of the effectiveness of the submodeling. In fact, in this case, the most displacement and the most extreme boundary conditions are expected. This technique has demonstrated very useful when simplified modeling is used to describe and study the global system. The combination between submodeling and “quasi-3D” models, developed in a 3D space, but composed by 0, 1 and 2D components, allows a very fast computational time with excellent results. In fact, what is highlighted in the comparison shown in this paper, is the huge similitude between the heavier and the lighter models. As illustrated, the technique of submodeling can describe a large range of cases: from simple components like bolts, to complex geometry, like a tubular joint with a lot of connections. As proof of the success of this model, the MORFEO Main Structure FE Analysis approach was submitted to the Preliminary Design Review in the first half of 2021. No critical issues [21][22] were identified during the review process.

6. ACKNOWLEDGMENTS

The MORFEO instrument is developed in the framework of the Agreement No. 65221/ESO/15/67001/JSC between the European Organisation for Astronomical Research in the Southern Hemisphere (ESO) and the Istituto Nazionale di Astrofisica (INAF) on behalf of the Consortium consisting of INAF, Institut National des Sciences de l'Univers du Centre National de la Recherche Scientifique (INSU/CNRS) acting on behalf of the Institut de Planétologie et d'Astrophysique de Grenoble (IPAG) and of School of Physics at the National University of Ireland Galway (NUIG).

REFERENCES

- [1] Ciliegi, P., et al., "MAORY: the adaptive optics module for the Extremely Large Telescope (ELT)", Proc. SPIE 11448, 114480Y (2020).
- [2] Ciliegi, P., et al., "MAORY: a multi-conjugate adaptive optics relay for E-ELT", ESO Messenger, No. 182 (2020).
- [3] De Caprio, V. et al, MAORY for ELT: preliminary mechanical design of the support structure, Proc. SPIE 10703, 107034H (2018).
- [4] De Caprio, V. et al, MAORY Main Structure design: general overview, Proc. SPIE 11448-156 (2020).
- [5] De Caprio, V., et al., "Mechanical design overview for the Main Structure of MAORY/MORFEO" Proc. SPIE, 12184-106 (2022), in these Proceedings.
- [6] Cianniello, V., et al "Lightweighting strategies for Main Support Structures of ELT instrumentation", Proc. SPIE 11451, 114512R (2020).
- [7] Ciliegi, P., et al., "MAORY@ELT: general overview up to the preliminary design and a look towards the final design", Proc. SPIE, 12185-40 (2022), in these Proceedings.
- [8] Cianniello, V., et al "Finite Element modelling technique: trade-off between two different FE models of a mechanical selector for Astronomical Instrumentation", Proc. SPIE, 12184-97 (2022) in these Proceedings.
- [9] Cianniello, V., et al., "Structural Analyses of the MAORY/MORFEO Main Support Structure at global level for Preliminary Design Review", Proc. SPIE, 12184-138 (2022), in these Proceedings.
- [10] D'Auria, D., et al., "Mechanical and electronic design of the MORFEO (formerly known as MAORY) Calibration Unit selector", Proc. SPIE, 12184-109 (2022), in these Proceedings.
- [11] Redaelli, E., et al., "MAORY@ELT: Optomechanical preliminary design", Proc. SPIE, 12185-177 (2022), in these Proceedings.
- [12] Aliverti, M., et al., "MAORY@ELT: Thermal Control System preliminary design", Proc. SPIE, 12185-176 (2022), in these Proceedings.
- [13] Pariani, G. et al, MAORY optical design and performances: status at preliminary design review, Proc. SPIE 12185-212 (2022), in these Proceedings.
- [14] Farinato, J., et al., "The MAORY MAIT strategy in Europe", Proc. SPIE, 12185-267 (2022), in these Proceedings.
- [15] Riva, M., et al., "MAORY@ELT: System Engineering activity up to preliminary design review", Proc. SPIE, 12187-60 (2022), in these Proceedings.
- [16] Cascone, E., et al., "General overview of MORFEO (formerly known as MAORY) Instrument Control Hardware design", Proc. SPIE, 12184-110 (2022), in these Proceedings.
- [17] Cascone, E., et al., "MAORY Instrument Control Hardware: general overview", Proc. SPIE 11447, 1144735 (2020).
- [18] Di Rico, G., et al., "MAORY@ELT: Calibration Unit overview", Proc. SPIE, 12185-204 (2022), in these Proceedings.
- [19] Eredia, C., et al., "Analysis of the requirements and their impact on the design of electronic cabinets for the current generation of ESO ELT instruments", Proc. SPIE, 12184-112 (2022), in these Proceedings.
- [20] Eredia, C., et al., "Trade-off between different PLC based architectures of Instrument Control Hardware for ESO ELT class of instrumentation", Proc. SPIE, 12184-111 (2022), in these Proceedings.
- [21] Chinellato, S., et al., "MAORY: the RAM analysis approach for preliminary design", Proc. SPIE, 12187-9 (2022), in these Proceedings.
- [22] D'Auria, D., et al., "Hazard and failure modes analysis of the Instrument Control Hardware design for ELT class of instrumentation", Proc. SPIE, 12187-53 (2022), in these Proceedings.