COMSOL CONFERENCE 2023 MUNICH

# Designing a satellite component assembly resistant to high vibrational stress using COMSOL Multiphysics®

V. Mesle[1], J.-D. Wheeler[2], M. Jouan[1], V. Bruyere[2], J. Leost[1], L. Couteleau[1]
1. RAKON France SAS, 2 rue Robert Keller, 10150 Pont-Sainte-Marie, France
2. SIMTEC, 5 rue Félix Poulat, 38000 Grenoble, France

## Abstract

RAKON is a worldwide expert in high performance crystal oscillator. The company decided to expand its product range to larger space equipment, the Master Reference Oscillator (MRO). This product is composed of stacked mechanical modules, joined together by structural screwed assemblies.
During the launch to enter orbit, this equipment is subject to high acceleration and stress, well defined by space standards and products specifications.
Numerical analyses are required to ensure the design of such screwed assembly. For that, RAKON extensively assesses their devices under representative conditions. To guarantee the most realistic model possible, a precise determination of the structure's physical parameters and an adaptation of the screwed assembly analysis methodology is required.
In this paper, an experimental analysis is performed by RAKON with the help of SIMTEC to assess the damping behaviour of the mechanical structure. The quality factors are first determined, and then numerically implemented in an acceleration spectral density (ASD) analysis model, representing the random vibrations imposed on the system.
Those numerical simulations are traditionally ran using legacy software, which are not always adapted for complementary analysis. The versatility of COMSOL Multiphysics® gives a better control of condition's analysis. By upgrading the software's physics readily available spring elements with rotation degrees of freedom, SIMTEC proposes a method finely adapted to RAKON's needs.
This changes in COMSOL Multiphysics® are the opportunity to validate the mechanical behaviour of structural screwed assembly under vibrational stress.

**Keywords: Random Vibration, FEM analysis**

## Introduction

The design of complex mechanical assemblies necessarily requires the sizing of screwed connections. These fasteners are the place of complex phenomena involving structural mechanics, materials physics, and tribology. In the context of a space environment, they are subjected to extreme mechanical stresses during the orbiting phase, and micro vibration during operation phase in space. A correct modelling of those stresses is essential to obtain the right dimensioning. This helps avoid breakage, but also oversizing the assembly, related to a problem of mass economy inherent to the space industry.
RAKON faces these challenges with the introduction of a new product range to its catalog. The company, already a leader in the design of high-performance quartz oscillators, is now launching the manufacture of MRO (Master Reference Oscillator) modules, a system that is more massive (up to 6kg) and more cumbersome (110x230x110mm) than the company's reference products.
To model this system, RAKON uses a multi-disciplinary finite element method (FEM) software named COMSOL Multiphysics®.
The system under consideration is made up of structural screw assemblies, which resist external mechanical stresses, and fastening screws, which hold the electronic equipment in place.
The aim of this article is to explain the method used to numerically represent the mechanical environment of these screwed assemblies. In the first part, a representative mechanical structure is submitted to random vibration stresses to determine the dynamic response of the dummy. Then, in a second part, numerical fastener model is developed on COMSOL Multiphysics® with the help of SIMTEC experts.

## Structural damping assessment

### Structure to analyse

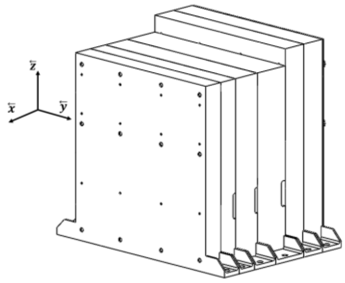


*Figure 1: Experimental Structure*

Under vibration, the assembly is submitted to propagation of mechanical wave spreading through the structure. Depending on its geometry and its material properties, those solicitations will be either transmitted, accentuated by physical resonances, or dissipated within the structure.

This energy loss is quantified by the quality factor (Q factor). It is defined as the ratio of the initial energy stored in the resonator to the energy lost in one radian of the cycle of oscillation:

$$Q = \frac{f_r}{\Delta f}$$

where $f_r$ is the resonance frequency, and $\Delta$f the is the half-power bandwidth frequency interval. High Q factor leads to high amplifications (see *Figure 2*), meaning that at the frequency studied, the system will see important stress that can lead to mechanicals damages.

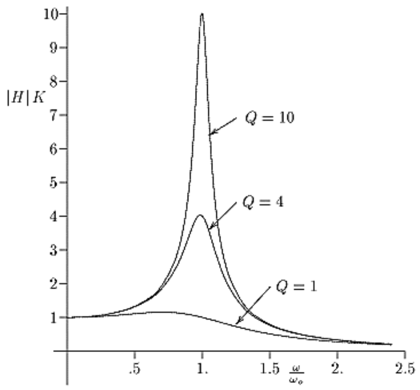


*Figure 2: Amplification variation at resonance*

Since digital simulation is highly dependent on material properties, a precise knowledge of the structural damping is crucial to model the stress transmission. Experimental measurements help to finely tune numerical model material properties to better predict a representative behaviour. The measure is performed on a structural dummy (shown in Figure 1 and Figure 3). It aims to be representative of the actual system under stress. It is mainly composed of aluminium and steel for the structural parts, epoxy boards for the electrical circuits, and additional mass to represent specific heaviest components.

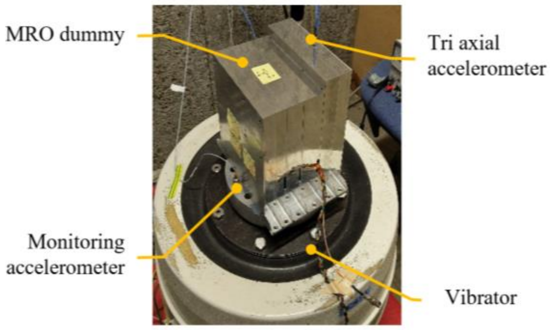


*Figure 3: Test set-up*

It is mounted on a vibrator from LDS manufacturer (cf Table 1). A tri-axial sensor accelerometer allows for collecting data on the structure during the test. The testing set-up is shown in F*igure 3*.

The control sensor is used as a validation of the input on the structure. To excite all the frequencies in the specification range (*20-2000Hz*), random vibrations levels are injected to the structure, corresponding to a uniform acceleration spectral density (*ASD*) with an overall energy of 4grms.

The tests are performed with the nominal torque load on the screws. No observation of preload's shift is observed after the tests.

**Experimental set-up**

| **Equipments** | **Fabricant** | **SN** | **Calibration Date** | **Sensibility (mV/g)** | |
|---|---|---|---|---|---|
| *Monitoring Accelerometer tri-axial* | *HBK* | *LW25 7613* | *01.02. 19* | *x* | *4.98* |
| | | | | *y* | *4.58* |
| | | | | *z* | *4.61* |
| *Control Accelerometer* | | *4514B* | *28.04.22* | / | |
| *Console* | *LDS* | *5455* | *04.01.22* | / | |
| *Amplifier* | | *S1445 /004* | *23.01.19* | / | |
| *Vibrator* | | *MCH-115* | *04.01.22* | / | |

*Table 1: Experimental set-up*

**Results**

The results expected are an amplification at resonance frequencies of the acceleration seen by the product.

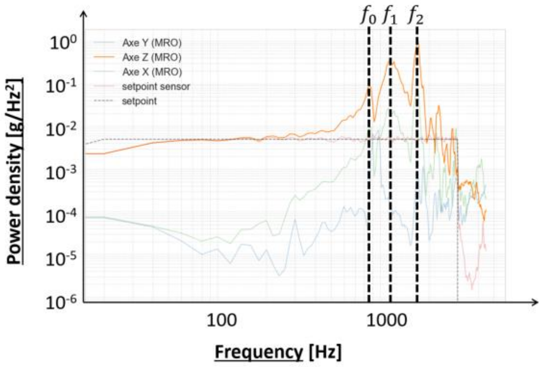


*Figure 4: Structure mechanical response under vibration*

*Figure 4* shows 3 principal eigenfrequencies in the specification range (20-2000Hz):

- $f_0 = 860\ Hz$
- $f_1 = 1140\ Hz$
- $f_2 = 1720\ Hz$

The quality factor is evaluated for each one of them and displayed in Table 2.

We observe the following results:

| *Eigenfrequencies [Hz]* | *Bandwidth at $1/\sqrt{2}$ [Hz]* | *Quality Factor* |
|---|---|---|
| *860* | *86* | *10.0* |
| *1140* | *228* | *5.0* |
| *1720* | *105* | *16.4* |

*Table 2: Eigenfrequencies and quality factors*

The first two eigenfrequencies present the lowest quality factors. This implies soft resonances modes, probably due to circuit board. The last frequency represents modes of the whole structure. The quality factors are in agreement with similar structures under similar conditions described in the COMSOL Application library (1), and hence the numerical damping is the same as the literature suggests. This results on the structure behaviour will be numerically implemented through a Rayleigh damping, detailed in the next section.

## Toward PSD simulations – Bolted joint characterization

A single bolted joint assembly model is proposed in this section. Two metal disks are fastened together using a single M3 8mm long screw passing through their centre. The frequency range of interest is 20-2000Hz. The goal is to assess the behaviour of a single bolted joint in a reference case with a full representation of the screw, the contacts, and the pre-load. Then this reference case is compared to different simplified cases with no screw geometry but more modelling assumptions: with/without pre-load, with-without continuity between the disks directly below the screw head, with/without spring accounting for the screw stiffness.

### Geometry Assembly

The geometry of the reference case is represented in *Figure* 5. The geometry steps are finalized using an assembly to create identity pairs between the components. In the reference model, continuity is assumed between the screw head and disk 1, between disk 1 and disk 2 directly below the screw head (the rest of the disk 1 and disk 2 interface is set free), and between the screw threads and disk 2.

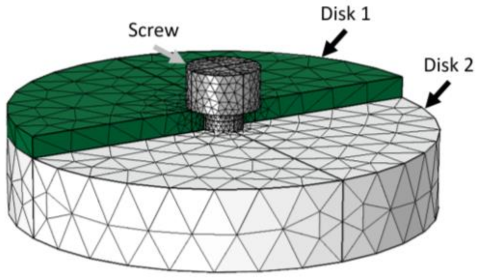


*Figure 5. Geometry and mesh of the reference case*

The screw is removed from simplified cases, and therefore the identity pairs involving it are excluded.

### Domain equations

In this model, linear elastic mechanic equation is solved in the domain. For pre-tension steps (either with the screw geometry and bolt-pretension or a spring with initial extension), the equation reads:

$$0 = \nabla \cdot S$$

with $S$ the stress tensor. In the multi-frequential analyses, it reads:

$$-\rho\omega^2\boldsymbol{u} = \nabla \cdot (FS)^T$$

with $\rho$ the density, $\omega$ the pulsation, $\boldsymbol{u}$ the displacement vector, $F = I + \nabla u$ is the deformation gradient, and $I$ the identity matrix. A transient analysis is also performed for comparison purposes:

$$\rho\frac{\partial^2\boldsymbol{u}}{\partial t^2} = \nabla \cdot (FS)^T$$

Since experimental results showed that the quality factors are typical of this kind of structure, the damping is selected accordingly to the analysis of the kind. A Rayleigh damping is used in all the non-stationary studies, with $f_1 = 40Hz$, $f_2 = 1000Hz$, and the damping ratios $\zeta_1 = \zeta_2 = 0.04$.

### Boundary conditions

A fixed constraint boundary condition is applied to the cylindrical shape around disk 2.

### Screw model

To replace the screw geometrically and the simplify the modelling of its behaviour, different coupling method are applied between the disks:

- a spring (as in Figure 6) can be added between the screw head mark on Disk 1 and the thread location on Disk 2: $F = F_s\frac{x_d - x_s}{l}$, with $F_s = k\Delta l$ the spring force, $\Delta l = l - l_f$

the spring length under load, $l_f = l_0 - \Delta l_0$ the initial spring length with extension, $l = \|\boldsymbol{x_d} - \boldsymbol{x_s}\|$ the computed spring length, $l_0 = \|\boldsymbol{X_d} - \boldsymbol{X_s}\|$ the geometric initial spring length, $\boldsymbol{x_s} = \boldsymbol{X_s} + \boldsymbol{u_s}$ the spring source point (filled green point thread in Figure 6) position under load and $\boldsymbol{x_d} = \boldsymbol{X_d} + \boldsymbol{u_d}$ the spring destination point position (filled grey point thread in Figure 6) under load. The spring points used to apply the spring source (hollow green points on the screw head mark in Figure 6) and destination (hollow grey points on the thread in Figure 6) locations are united using two rigid connectors.

- Pretension $\Delta l_0 \neq 0$ can be applied on the spring to account for the screw pre-load.
- A continuity can be applied at the interface between the assembled part, directly below the screw head.

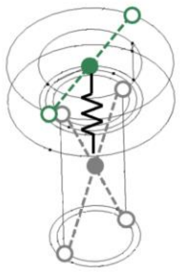

*Figure 6: Spring modelling the bolted joint*

**Mesh**

A tetrahedral mesh (see *Figure* 5) and quadratic serendipity elements are used.

**Cases and numerical method**

The different cases presented in *Table 3* are solved. For each case involving pre-tension, a stationary analysis precedes the dynamic analysis (whether it is frequential or transient). The dynamic analysis then uses the previous step to compute the equation in the pre-deformed state.

Ref 1 pre-tension analysis is also used to compute the screw stiffness and the initial extension of the spring under the 1400 N preload recommended by the state of the art. The stiffness is used for the A1, A2 and C1 analyses, and the initial extension for A1, C1 and C2.

| **Case** | **Screw** | **Spring stiffness (N/µm)** | **Disk contact** | **Pre-tension** | **Dyn. meth.** |
|---|---|---|---|---|---|
| Ref 1 | Geom. | - | Yes | Yes | Frequential |
| Ref 2 | Geom. | - | Yes | Yes | Transient |
| A1 | Spring | 141.3 | Yes | Yes | Frequential |
| A2 | Spring | 141.3 | Yes | No | Frequential |
| B | None | - | Yes | No | Frequential |
| C1 | Spring | 141.3 | No | Yes | Frequential |
| C2 | Spring | 1000 | No | Yes | Frequential |

*Table 3. Cases solved and compared*

The 141.3 N/µm stiffness is obtained from Ref1 simulation whereas 1000 N/µm is typically used in the assembly analysis of satellite structures: the latter stiffness is typically used to compute the forces in the screw, and it is supposed to be sufficiently stiff to leave the dynamical results unaffected in the frequency range of interest.

**Results**

The different cases are solved, and the acceleration seen by a point located on the outside of Disk1 is investigated. This point is likely to move in a large amplitude. The effective acceleration magnitude of such point is plot against frequency in *Figure 7*. The black line results account for the reference cases with the screw, and the Ref2 case is only computed at 2000 Hz. The Ref1 and Ref2 cases show very similar result at 2000 Hz, confirming the consistency of the analysis.

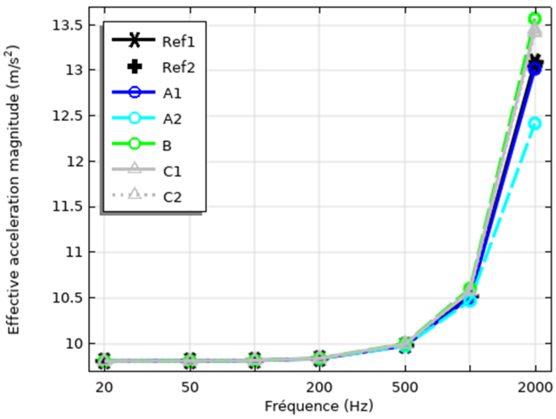


*Figure 7. Bolted joint assembly modelling method comparison*

The different methods show different results around 2000 Hz, with effective acceleration magnitude ranging from 12.5 to 13.5 m/s². The cases without pretension (A2 and B) show the most different acceleration magnitude with the reference cases. The cases C1 and C2 are superposed. It shows that using a representative spring stiffness is not necessary for this specific small structure in the frequency range considered. At last, A1 case uses the most accurate set of assumptions (screw modelled by a spring, contact between the disks directly below the screw head, and pretension) to model the behaviour of the bolted joint assembly: its acceleration response curve is very close to the reference cases.

## PSD simulations – Satellite assembly

This section presents the Power Spectral Density (PSD) analysis performed in this study. The Rayleigh damping coefficient are proposed after the "Structural damping assessment" experimental section, and the bolted joint assemblies after the "Toward PSD simulations – Bolted joint characterization" section.

### Geometry and geometry preparation

The CAD geometry of the MRO assembly is imported after selecting the component contributing to the structural behaviour of the whole system: the casings, the casing tops, and the PCBs.

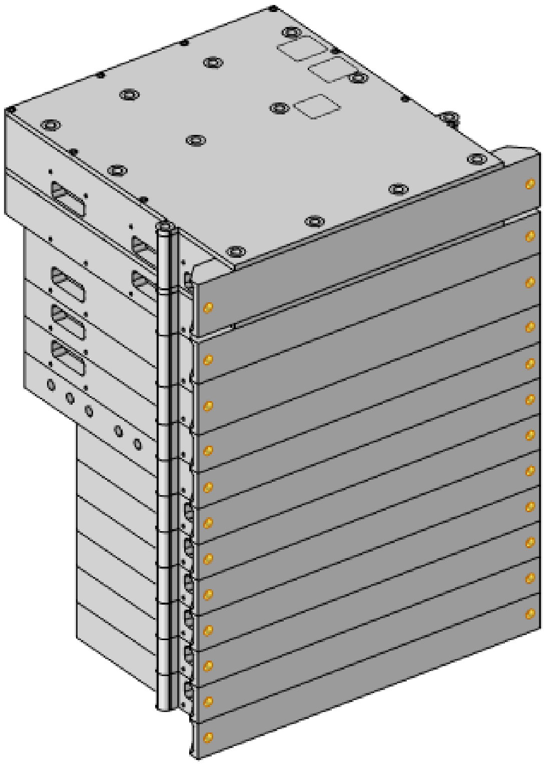

*Figure 8. The MRO assembly investigated.*

The screw head are reported as a mark (see *Figure 9*) on the material directly in contact with it. This allows for prescribing the spring rigid connector which connects the points used for one side of the spring.

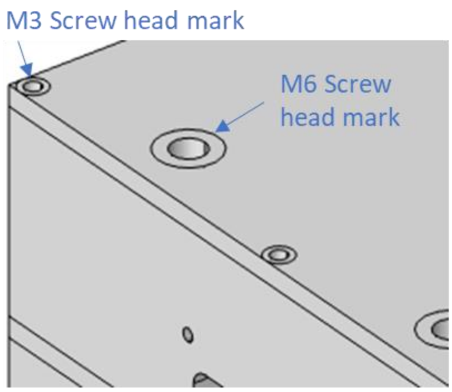


*Figure 9. The area under the screw head is reported on the assembled component.*

Moreover, the screw head shape is reported at the junction between the assembled bodies to apply a continuity condition right below the screw head.

### Materials

MRO modules boxes are made of aluminium and the PCB is made of Arlon 35N. The density of the Arlon PCB is increased to account for the copper circuit and the component masses. The properties are summarized in *Table 4*.

| **Material** | **Young modulus [GPa]** | **Poisson coef. [ - ]** | **Density [kg/m$^3$]** |
|---|---|---|---|
| Aluminium | 69 | 0.33 | 2750 |
| Arlon 35N + components and copper circuit | 26 | 0.15 | 7968 |

*Table 4. Material properties*

### Domain equations and damping

Similarly, to the single screw analysis, the pre-tension is computed using stationary linear elastic equation:

$$0 = \nabla \cdot S$$

Then an eigenfrequency computation is performed to assess the modes shape and energy using;

$$-\rho\omega^2 \boldsymbol{u} = \nabla \cdot (FS)^T$$

with $F = I + \nabla \boldsymbol{u}$. The eigenfrequency analysis is made on the state under pre-tension and geometric non-linearity are included.

### Masses

In some modules, heavy components are likely to influence the behaviour of the system. They are applied as distributed masses on the fastening screw holes.

### Rigid connector

The whole MRO unit is fastened to the satellite using support faces of the modules. This fastening is modelled using rigid connectors applied to the screw holes (represented in yellow in *Figure 8*). The 3

displacement directions are set to zero. The rotations are also constrained, apart from the rotation along the axis of the screw.
Rigid connectors are also used without constraints to couple the points of the screw head area of each bolted joint on the one hand, and the points of the screw thread on the other hand.

**Springs**
The COMSOL springs used to model the bolted joints are upgraded to include 6 degrees of freedom and pre-tension state. The tension-compression stiffness along the screw axis is set to the value computed using Ref1 model and it therefore varies between M3 and M6 screws. The transverse stiffness is set to 1e9 N/m. The rotational stiffness is set by computing the average rotation from source and destination points of the spring. The rotational stiffness in all directions is set to 1e7 N.m/rad. Lastly, in the matrix configuration of the stiffness, the pre-deformation variable is not available in the COMSOL interface. It is added manually in the relative displacement expression in the direction of the screw axis.

**Power spectral density**
The structure described here should be able to withstand high vibrational stress. Such stress profile is ideally described by a power spectral density (PSD). The PSD applied in this study is represented in *Figure 10*.

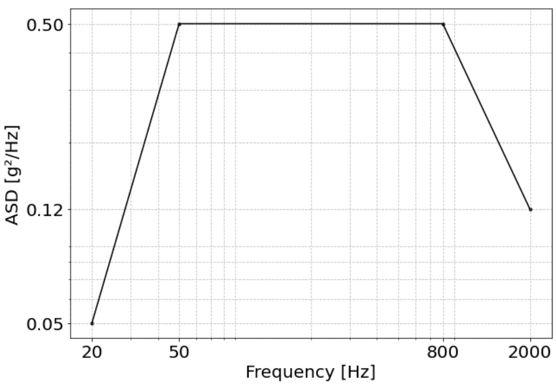


*Figure 10. Power spectral density*

**Mesh**
A mesh is applied to the whole geometry described earlier. When possible, an extruded mesh is applied to the shell-like volume domains to best predict flexural behaviour.

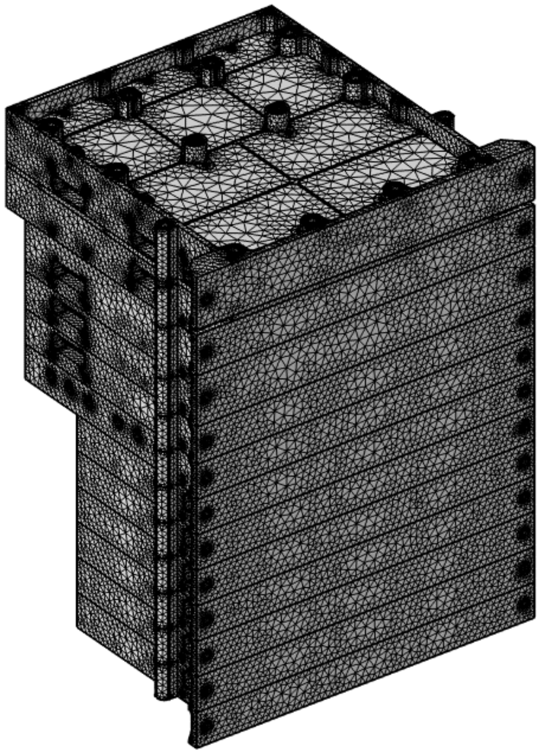

*Figure 11. Mesh applied to the domain*

**Numerical procedure**
At first, the pre-deformed state is computed using stationary solver. Then the Eigenfrequencies are computed in the pre-deformed state, including geometric non-linearity.
Then, a filter is applied to distinguish the most significant modes and delete the lower contributions to the reduced order model (ROM) of the others. At last, a ROM is built based on most significant modes and a frequency analysis at 400Hz. It is a modal reduction. The reduction is made with a specific linearization point: the pre-deformed state.
The goal is to compute the response of the system under a power spectral density, which can be performed as a post-treatment using the ROM.

To implement with more ease the numerous screws in the model, java routines and a specific user interface is made available to the user.

**Results**
The investigation presented here aims at predicting the ability of the bolted joint to prevent:

- slip between the parts,
- and loss of contact between the screw head and the component right below it.

For this purpose, all the forces and torques of the bolted joints assembly are computed thanks to the ROM as a post-treatment under the applicable PSD.

Forces on each screw are computed through their respective numerical springs described in the previous sections.

The following values are obtained on one of the M6 bolted joint:

- Axial force: 1351 N
- Radial force: 483 N

Margin of safety is computed to know if the integrity of the assembly is preserved. The criteria used for this example is the separation of the screwed flanges.

The ECSS handbook (2)gives the following formula:

$$MoS_{sep} = \frac{F_{v,min}}{(1 - \Phi_n)F_A\, sf_{sep}} - 1$$

With $F_{v,min}$ the minimum preload driven by the tightening uncertainty, $\Phi_n$ the screw compliance ratio balanced by the loading plane factor, $F_A$ the axial force, and $sf_{sep}$ the safety factor, taken according to the used case. To be valid, the result should be strictly positive. For the M6 screw described above, this formula gives a Margin of Safety separation factor of 2.2, which validates the mechanical solidity of the screw. Finally, this analysis is performed at each screw of the assembly, to validate the global behaviour of the whole system.

## Conclusions

After assessing experimentally the structural damping properties of a large new satellite structure, the MRO, a numerically study is performed. A bolted joint model is selected between the different hypotheses tested and the model is applied to the full structure of the MRO.

Whereas only one fastener performance is measured in this document, the full study investigates each individual bolted joint to verify its ability to withstand the important stress during the launch to enter orbit.

## References

1. Random Vibration Test of a Motherboard, COMSOL Application library. 2021. p. 1–30.
2. ECSS-E-HB-32-12A ECSS Handbook.