Faculty of Applied Sciences
Aerospace and Mechanical Department

Structural & Multidisciplinary Optimization
Topology optimization for structural design

Designing the supports of an airplane oil tank

1. INTRODUCTION

Additive Manufacturing (AM) offers unprecedented form freedom and is rapidly being adopted throughout industry. However, to fully exploit its capabilities, designers require specialized design tools, among which Topology Optimization (TO) stands out. Although TO is the most suitable choice for AM users, the connection between TO and AM is still not straightforward since most of optimized designs require the designer's intervention to be adapted to the AM process. Within this context arises the Aero+ project, funded by the Plan Marshall 4.0 and the Walloon Region of Belgium.

As part of the objectives, the Aero+ Project seeks to improve the performance of jet engines using Additive Manufacturing and Topology Optimization. The Safran company, industrial partner of the project, has proposed to optimize the supports of an oil tank, such as that shown in Figure 1.

In this work you are asked to submit a report proposing a design for this industrial problem using Topology Optimization, including a description of the finite element model, the optimization problem and the results.

You will find the details on the problem and the report requirements in the following sections.

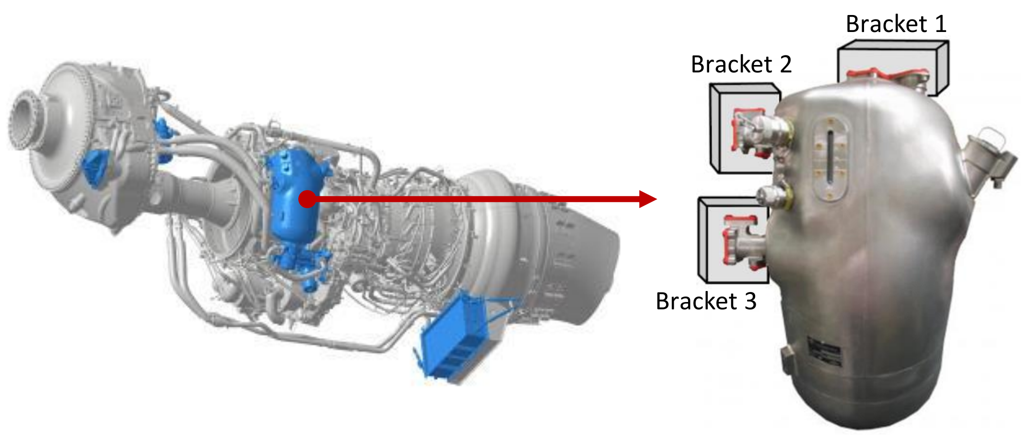

Figure 1: On the left, a representation of the oil circuit. On the right, the oil tank and the supports that must be designed. The pictures are just for illustration purposes and do not represent the real geometry of the problem. Pictures obtained from https://www.safran-aero-boosters.com/.

2. LOAD CASES

The oil tank mass is 21.8 kg. The reservoir weight is not the main loading. The critical loads are dynamic loads due to the engine vibrations. Figure 2 shows the accelerations coming from the engine. Table 1 also summaries the numerical values of the acceleration for some selected frequencies. These accelerations affect the oil tank in all directions, i.e. the maximum accelerations on the X, Y and Z axes are those shown in Figure 2. Under these dynamic loads, the support must not exceed the high cycle fatigue limit. As a critical load case, the Fun Blade Out (FBO) is considered, where accelerations reach 200 [G] and the maximum stresses should not exceed the ultimate strength.

Table 1: Accelerations of engine vibrations.

Freq. [Hz]	5	14	22	49.4	170	205	381	1000	1600	2300
Acceler. [G]	0.1	1	1	5	5	8	17	25	40	40

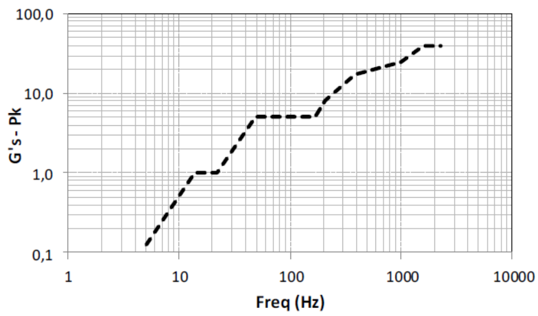

Figure 2: Accelerations of engine vibrations.

The system including the oil tank and the three supports is hyperstatic and the stiffness of the optimized supports may modify the reaction forces going through the supports. Determining their exact values requires to solve a full finite element simulation including the supports and the reservoir, which is computationally too expensive in the framework of this student project. To circumvent the difficulty, we have run a finite element simulation of the full system (see Figure 3) to determine the reactions forces taken by each of the support and we will consider that these reaction forces will be frozen during our optimization run. The loads (reaction forces) which must be considered are summarized in Table 2. They contributions must be multiplied by the acceleration $m \cdot a$.

Table 2: Reaction forces on the 3 supports for the 3 load cases. Forces are normalized by the product $m \cdot a$.

	Load case 1 (F_x)			Load case 2 (F_y)			Load case 3 (F_z)		
	f_x	f_y	f_z	f_x	f_y	f_z	f_x	f_y	f_z
Bracket 1	0.3046	0.0055	0.0139	0.1622	0.3479	−0.0298	−0.0181	−0.0434	0.2720
Bracket 2	0.2897	0.0105	−0.0951	−0.1212	0.2145	0.0439	0.2970	−0.0517	−0.0030
Bracket 3	0.4057	−0.0159	0.0812	−0.0410	0.4376	−0.0141	−0.2785	0.0951	0.7310

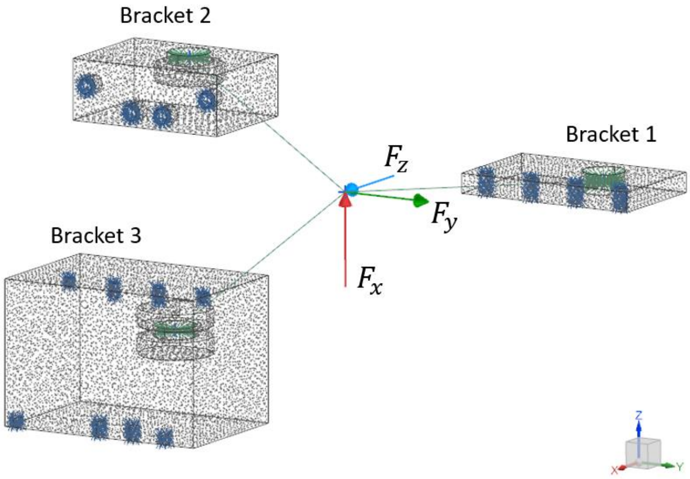

Figure 3: The finite element model used to obtain the reaction forces on the 3 supports. Three load cases are considered, F_x, F_y and F_z. The magnitude of these forces is the product of the mass of the tank and its acceleration $m \cdot a$. The forces are applied on the Center of Gravity of the oil tank.

3. DESIGN DOMAINS & MATERIAL

The design domain of Bracket 3 is shown in Fig. 4 and the material properties are listed in Table 3. You will receive a copy of the part file with the proper dimensions.

Table 3: Mechanical properties of the material used to manufacture the 3 supports.

Mechanical Properties	[MPa]
Yield Strength	700
Ultimate Strength	750
High Cycle Fatigue	320
Young Modulus	200000
Poisson’s ratio	0.33 [-]

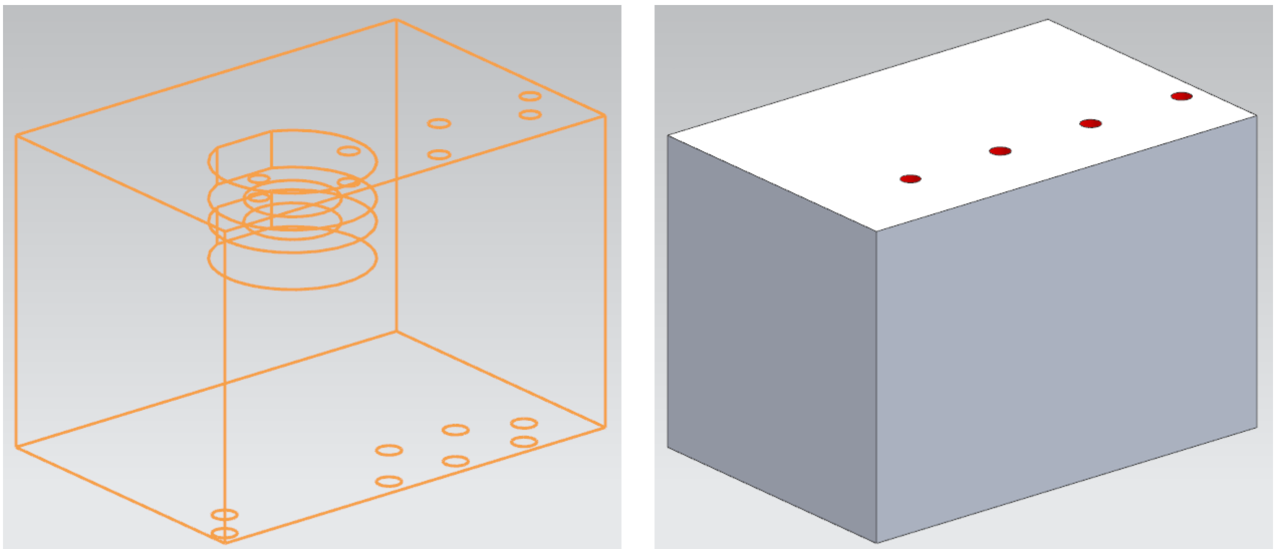

Figure 4: Bracket n° 3.

4. PROBLEM DEFINITION

In the present project you are asked to find an optimized design using Topology Optimization (NX TOPOL). The design should make use of the smallest possible mass (in other words for a given design domain, use the smallest volume fraction in the volume constraint) while respecting the mechanical properties summarized in Table 3. You can exceed the yield strength at few elements, but you are not allowed to exceed the ultimate strength.
You will have to perform the analysis for two operating conditions: nominal condition ($a = 40G$) and FBO condition ($a = 200G$).

1. Using the .prt file that you received, complete the finite element model of the design domain given to you.
   a. Create the mesh.
   b. Impose the loads and boundary conditions.
   c. Add any modification to the part that you consider necessary to perform the optimization.
2. Perform a topology optimization considering the given load cases and boundary conditions. Select the parameters of the optimization run and justify your choice. Don't hesitate to modify these ones and assess their influence on the optimized topology.
3. Include a finite element analysis on the purged model. Discuss about the stress level reached in the model with respect to the yield and ultimate strength. Compare the results for both operating conditions.
4. It is possible that for the FBO condition ($a = 200G$) the stresses exceed the ultimate strength values. You are asked to propose a solution to overcome this problem (change of material, increase of volume fraction, design modification, etc.).
5. Perform the post-processing of the raw solution, i.e. the smoothing and the production of a cad model.

5. REPORT REQUIREMENTS

You are asked to prepare and send a written PDF report of 20 pages maximum including, but not limited to:

1. The details of your finite element model (mesh, boundary conditions, forces, passive elements if present, etc.).
2. The details of your Topology Optimization problem (number of iterations, parameters, etc.).
3. Your optimized results (material distribution, stresses, material choosing, etc.).
4. Analysis of every results and considerations.
5. The quality of the report (optimized design, figures, text, tables, etc.) will also be part of the evaluation.

Send a .rar or .zip file named as «student1_student2» with the report and the NX files (only *.prt, *.fem and *.sim) to efsanchez@uliege.be and palarcon@uliege.be before **05/01/2020 at midnight.**