Optimising Aluminium Structures Using the FEA Method

H Cioban
Technical University of Cluj-Napoca, Faculty of Engineering, Department of Engineering and Technology Management, Baia Mare, Romania

E-mail: horia.cioban@cunbm.utcluj.ro

Abstract. As a request of a partner and subject of a research contract, this paper intend to present different stages of studies regarding the behaviour of some Aluminium structures under different types of loads. Starting from existing examples, a company from aerospace industry intended to produce similar products but on a larger scale. On a virtual product created in a CAD environment and using a FEA method is possible to optimize aluminium structures and to apply the conclusions and results given by FEA reports to the real products. Tests made on manufactured structures validate the findings of these studies.

1. Introduction
In Romania, the government statistics indicate a 70% increase in the last 3 years revenue from the aerospace industry. The main products manufactured are aircrafts, helicopters, gliders, engines, landing gears, parts and subassemblies [1]. Studies made in 2019 identify the main trends in R&D focused on materials (aluminium and composites), electric and hybrid engines, mechanical solutions, electronics and software [2].

Using CAD-CAM-CAE software, aircrafts manufactures are beginning to redesign outdated models or create new models. The aluminium structure of the airplane is subject to some of these changes in design activity.

2. Requirements of the studies
Until few years ago, the maximum length of the beams in the aluminium structure of the aircrafts was up to 6m. The progress of concepts and the new technologies allow the length to increase up to 9m.

The aim of the project is to design and to optimize two products: trolley for parts (figure 1) and container for three trolleys (figure 2).

![Figure 1. Trolley for long beams transportation](image1)

![Figure 2. Container for three trolleys](image2)
These extruded aluminium beams are parts of the airplane structure, and transport from the manufacturer's sites to the assembly workshop is a problem that needs to be solved.

In design studies, it started from similar existing products used for 6m beams. The main purpose is to get minimal deformations of these welded aluminium structures.

2.1. Additional constraints
Additional requirements in these studies were related to operating and transport conditions. Thus, it was required to know how the winch is behaving in forklift lifting, and the container at lifting with forklift and bridge crane. Other simulated situations where: deformations of the container under the conditions of 2 overlapped containers, which are the deformations if the container remains suspended in only 3 points, or if the conditions required by the road transport rules are met, where three-ways accelerations can occur, as in figure 3.

![Figure 3. Simulation conditions imposed by road transport regulations](image)

Each of these simulations showed the critical areas in the aluminium structures of the trolley and the container and required changes of the product geometry. After each modification in the project, the stages required by FEA simulation were resumed.

2.2. Type of analysis
The analysis used to study the behaviour of these two welded assemblies is known as static structural, that means the loads and displacements are in the elastic area of the Hooke’s law diagram. FEA analysis concepts accept some simplifications of the model’s geometry:
- a welded assembly having bonded contacts between parts can be simplified to a single part-one body;
- the welding seam can be ignored;
- avoid rounded/chamfered edges; all profiles must have sharp edges;
- avoid small gaps between parts; the model has to be continuous.

3. Pre-processing stage
The pre-processing phase is also known as the model preparation. It is the most labor-intensive step because it implies that only the part of the project that is relevant to the study is chosen and the parts that do not influence the results are excluded from the analysis. Thus, the wheels were removed from the analysis made on the trolley and from the container only the mainframe were retained, and it was ignored the exterior walls and doors, as in figures 4 and 5.

![Figure 4. The complete container design](image)  ![Figure 5. Geometry subjected to FEA analysis](image)
3.1. Material properties
The choice of the materials used was imposed by the initial conditions of the design theme. Aluminium alloys have been required especially because of the lower weight than steel alloys, although aluminium, due to high elasticity, has deformations greater than steel under the same stress conditions [3].

Most FEA software has an internal library of materials or provides functions to create or import data to define the properties of the materials used in analyses. In figures 6 and 7 the properties of a steel alloy and of an aluminium alloy are compared.

![Figure 6. Properties of a steel alloy](image1)

![Figure 7. Properties of an aluminium alloy](image2)

Significant differences can be seen in density (3 times higher in steel than aluminium alloy and giving a larger mass at the same volume), in Young's modulus that gives the elasticity property (higher at aluminum) and in Bulk modulus.

3.2. Meshing stage
This stage involves dividing the 3D model studied into geometric elements of tetrahedron or parallelogram shape, called finite elements. The results of the analysis will be more accurate if the finite elements will have more regular shape. Also, a deeper division of the 3D model leads to results closer to reality.

The meshing stage involves a strong hardware and time-consuming resources. There are often situations when errors occur, and the analysis cannot continue until they are eliminated. For this reason, it is not recommended to directly import the CAD model into the FEA application, but to reconstruct the geometry from scratch, under the simplifying conditions presented in paragraph 2.2. For example, chamfers and fillets are removed from a beam section. The 3D model must be continuous and gaps, even small, are not accepted. Figures 8 and 9 display the result of the meshing stage of the trolley and the container, the reports showing that the trolley 3D model was divided into 37069 finite elements and the container in 36862 finite elements.

![Figure 8. Meshing the trolley](image3)

![Figure 9. Meshing the container](image4)

At the first running of the meshing phase, the default settings of the software parameters are used. After all errors in geometry are eliminated and the meshing is successful finished is recommended to refine the mesh and to reduce the size of elements, mostly at the critical areas. Finally, the number of the finite elements can reach hundreds of thousands and even millions.

3.3. Boundary conditions
This is the last step of the model preparation and contains applying of constrains like fixed supports, defining loads (forces, pressure or moments) or applying external forces (earth gravity, accelerations...
etc.) [4]. Boundary conditions are defined by the real environment in which the trolley and the container will be used.

In the trolley case, the position of the fixed supports depends on the handling conditions: it is supported on wheels or is raised on forklift. In the container case, the position of the fixed supports is different when it rests on the feet or is lifted by fork-lift or bridge crane. Each new boundary condition generates a new FEA study.

Earth gravity and 2700N forces (the weight of carried parts) were applied on the trolley, as in figure 10. Accelerations generated by the vehicle’s movement were added in the case of road transport.

At the container, 27000 N forces were applied given by the weight of the 3 loaded trolleys applied to the runway surfaces, or 24000 N, the weight of an empty container placed on the top of another container. In each case, gravity was taken into account.

![Figure 10. Applying of boundary condition on trolley](image)

![Figure 11. Applying of boundary condition on container](image)

When the all model preparations are finished, the finite element analysis can go further to the next step: processing using a solver.

4. Processing stage

The boundary conditions are included in equations and distributed on each finite. Depending of number of finite elements of the model, a system of millions of equations (the total stiffness matrix) has to be solved and this is the role of the solver. Sometimes the processing stages takes too long or even the solver cracks. For these reasons a strong hardware is required.

The results consist of displacements of the single nodes, distortions, stress and nodal forces.

5. Post processing stage

After the processing is finished FEA applications display the results in a very intuitive graphical mode. On the mesh of the 3D model, each finite element is coloured using a graduated scale, from blue to red. Interpretation of results is thus very easy, and critical areas can easily be highlighted.

5.1. Equivalent (von-Mises) Stress

![Figure 12. Equivalent (von-Mises) Stress at the 1st version of the trolley](image)

![Figure 13. Equivalent (von-Mises) Stress at the optimized version of the trolley](image)

One of the solver results is the equivalent tensile stress. The maximum value calculated for each finite element has to be in the elastic area of the Hooke’s law diagram. The resulting values for both
the trolley and the container show that under the conditions imposed by the initial data of the study topic the loads are within acceptable limits.

In the case of the winch, figures 12 and 13 show the results of the analysis for the initial model, respectively after the optimization of the geometry by the addition of hardening beams.

For the container, the results are shown in figures 14 and 15 for lifting with side-loaders and bridge cranes at the 1st version and optimized version.

![Figure 14. Equivalent (von-Mises) Stress (side loaders lifting)](image)

![Figure 15. Equivalent (von-Mises) Stress (bridge cranes lifting)](image)

The values calculated in each case are compared to the admissible value of the aluminum tensile yield strength (280 MPa). In all cases, the maximum values are lower than the admissible values, so the loads to which the winch and the container will be subjected will not affect the mainframe. Changes to geometry show a better behaviour on constrains and loads.

### 5.2. Total deformation

In the optimization of the geometry of the two aluminum structures, the calculation of the displacements of the finite elements gives a total deformation. Figures 16 and 17 show the deformations of the trolley and container structures, comparing the original with the optimized version.

![Figure 16. Total deformation of trolley (side loaders lifting)](image)
Changes to the geometry of the welded structures have led to decreasing of deformations from 14.8mm to 2.5mm to the trolley and from 3.9mm to 1.5mm to the container.

6. Conclusions
Studies using the finite element method have proven to be extremely useful in the cases presented. One of the possible solutions is that of using another material with better physical properties, but was excluded from the initial data of the theme: aluminium alloy was imposed. Geometry optimization was the only possible option.

Also starting from the initial conditions 3 situations has resulted for the trolley and 4 for the container: lifting with side loader, lifting with bridge crane (with 3 and 4 supporting decks) and transport behaviour. From the first version of the geometry to the final version, there were at least 6 intermediate stages, resulting in over 50 FEAs performed. The results of the analyses showed a behaviour of the welded aluminium structures within the permissible limits, with enough safety margin. Measurements made on prototypes showed deformations close to those calculated on virtual products. Currently, both the trolley and the container are in operation.

References
[1] Costescu M 2016 Aerospace in Romania, http://investromania.gov.ro/web/wp-content/uploads/2016/07/Aerospace_EN.pdf
[2] PwC Advisory 2018 Turning the Aerospace and Defence industry into a source of added value for the economy, Atlantic Black Sea Security Forum, Bucharest June 5, http://aspeninstitute.ro/wp-content/uploads/2018/05/PwC-Presentation_Atlantic-Black-Sea-Security-Forum_June5_2018.pdf
[3] Moldovan Ş A, Banica M, Năsui V 2017 Defining the machining process for complex parts made from aluminium alloy extruded profiles by correcting raw material deviations, MATEC Web of Conferences Vol. 112 pp.01011
[4] Butnar L A, Basarman A P, Tranisan D I, Groza I 2017 FEA Deformations Analysing of the Polyurethane Armchair Seat Materiale Plastice Vol.54 Issue 3 pp.593-595