Structural Analysis using Finite Elements

1 – Select a structure or structural component for aerospace applications.

2 – Contextualize the application of the chosen structure/component, highlighting operating conditions, applied loads, critical conditions, structural constraints, geometric characteristics, materials used, etc.

3 – Considering critical operating conditions, build a finite element model (using Abaqus or Ansys). for a quasi-static analysis of the chosen structure/component.

4 – Explain the simplification assumptions in the construction of the numerical model, in terms of geometry, loadings, and boundary conditions. Explain and justify the types of finite elements used.

5 – Perform mesh convergence analysis, in terms of the variable deemed pertinent, to define the mesh adequate for analysis.

6 – Using an adequate mesh, perform the structural analyses, in terms of the information deemed necessary: displacements, reactions, internal forces, deformations, stresses, etc.

7 – Adopting appropriate failure criteria, discuss the structural integrity of the analyzed assembly/component, as well as possibilities for improving the original design.

Notes:

• The assignment should be submitted as a report in PDF format, accompanied by a presentation in PPT format or an equivalent.

• Include a variety of images from the simulations.