MECA-0036

Finite Element Method Project 2020-2021

The goal of the project is to perform, for a mechanical problem of known geometry, boundary conditions and loading, the following tasks using the Siemens NX Pre/Post software with the Samcef Solver[1]:

I **Strength of Materials (10%):**

1) Write, in details, the equations of your mechanical problem (without simplification)[2] : Displacement-strain relations, constitutive equations, equilibrium equations and kinematic/static boundary conditions. (**You do not need to solve them**); ($**$)

2) Describe all simplification hypotheses used to transform your mechanical problem into a simplified model, with their range of validity. (**Do not hesitate to oversimplify your model!**); ($**$)

3) Determine, on your simplified model, the expressions of the stress and/or displacement at critical points **as a function of the applied load**: use beam theory, strength of material, linear elasticity theory, superposition principle, etc. Do not hesitate to refer to the course MECA0001-2 and MECA0012-6. (**Only a rough order of magnitude is sought, not the exact answer**); ($***$)

4) Based on the literature, strength of materials and linear elasticity, determine stress concentration zones and high stress gradient zones in the real structure. (No demonstration needed, you can directly use the results from bibliographic resources [3] and comment them); ($**$)

5) **Discuss your results**. ($***$)

---

[1] Weight of each task: ($*$)= low, ($**$)= medium, ($***$)= high

[2] The mechanical problem is the same as the one analysed with Siemens NX.

[3] You can use the online library of the University of Liege: ULg Library.

II Analysis with the Finite Element Method (70%):

All the following tasks are performed **in order to determine the maximum admissible load** to apply to your mechanical problem:

1) Fix the rigid body modes[4], and/or use the symmetry of the problem (symmetry of load and geometry), **if applicable**. **Make sure that the new boundary conditions are in accordance with your strength of materials analysis** (**)

2) Describe **precisely** the modeling of the mechanical problem on the NX software[5]: modeling, meshing, load, boundary conditions, etc. (**)

3) Check if the boundary conditions or the geometry do not create **singularities**[6] in the finite element model and show it. What can be said about the **local convergence** of the stress field and the **global convergence**? Before going any further in your analyses, fix, if necessary, the issues resulting from the singularities, show that they are no longer a problem for your simulations and explain the changes in your numerical model. (* * *)

4) On the basis of the results from the strength of material analysis and an arbitrarily chosen applied load, you are asked to perform a finite element analysis on a coarse mesh (i.e. no sub-domains and automatically generated by NX) using four different types of finite element: first and second order triangular elements: $T3$ and $T6$; and first and second order quadrangular elements: $Q6$ and $Q8$. (**No combination of different elements!**). To do so, you are asked to perform the following tasks for each type of element:

   a. Define each type of element used; (*)

   b. Analyse the geometry approximation: check the area/volume of your discretised model; (*)

   c. Analyse the obtained results: check the spatial distribution of different components of the stress tensor and check if the boundary conditions are correctly respected (cf. question I.1); (*)

---

[4]If you have to **arbitrarily** chose other additional boundary conditions than the ones specified in the statement, check the influence of this choice on the obtained results (stress and displacement) and justify it! (e.g. with reaction forces, obtained stress/displacement field)

[5]We should not have to open your NX files to fully understand your modeling

[6]In the context of continuum mechanics a singularity is a material point where the solution to the problem is not completely defined, meaning that either the stress/strain or the displacement field locally goes to infinity (cf. Singularity in FEM).

d. Check the mesh quality using NX tools: Jacobian ratio, Aspect ratio, Skew angle, Taper, Corner Angle, etc. Use only the tools you find relevant; (∗∗)

e. Study the sensitivity of your results (stress and displacement fields) to the size of the element and the number of degrees of freedom. **Analyse the local convergence of the stress field**; (∗ ∗ ∗)

f. Plot the graphs of the total potential energy and of the computation time[7] as a function of the number of degrees of freedom and of the number of finite elements[8], comment only the graphs you find relevant.(∗)

5) Discuss your results and conclude on the advantage and disadvantage of the different finite element types used. Which type of element is the most suitable for your simulation? (∗∗)

6) Based on your results using coarse meshes:

a. Develop an advanced mesh for both triangular and quadrangle elements (i.e. with advanced sub-domains and/or varying distribution of finite elements among them); (∗ ∗ ∗)

b. Show that the quality of the meshes greatly improves; (∗∗)

c. For the best element found in question QII.5, show that the convergence of your results greatly improves (global & local convergence, computation time). Deduct a compromise on your mesh. (∗ ∗ ∗)

7) Find the maximum admissible load related to your statement, discuss your results (stress and displacement fields) and **compare them with the results from the strength of material analysis**. (∗ ∗ ∗)

Please define every tool, concept, element, etc. you are using in your report, be concise and use a consistent structure for your report.

III **Optimization** (10%):

1) Answer the problem imposed by your statement with one optimization[9]. The choice of mesh and finite element type used during the optimization process is

---

free and must be justified in the report (you can use a different discretization, especially if the optimization drastically affects the creation of an advanced mesh, as long as the mesh quality remains good). **If applicable**, you can optimize the shape and the dimensions of the geometry marked in red on the structure in your personal project statement by using circles, lines, ellipses, parabolas, etc. $(* * *)$

2) Justify the steps of your optimization with the help of your observations in the strength of material analysis and from the results of the finite element method (analyse the stress and displacement fields). You can show different considered configurations and and select the most efficient one; $(* * *)$

3) Submit a final drawing of the most optimized structure with dimensions. $(*)$

IV **Quality of the report** (10%):

A particular attention will be given to the quality of the report, to the ability to summarize and to the use of adequate scientific vocabulary. For more information, please refer to the "Report instruction" document.

IMPORTANT: All questions should find an answer in your report!