3rd Bac. Mech. and isolated
MECA-0036

Finite Element Method Project 2017-2018

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]:

1. **Strength of Materials** (10%)**:**

   - Write, in details, the equations of your mechanical problem[2] : Displacement-strain relations, constitutive equations, equilibrium equations and kinematic/static boundary conditions. (∗)
   - Describe all simplification hypotheses used to transform your mechanical problem into a simplified model, with their range of validity. (∗∗)
   - Determine, on your simplified model, the expressions of the stress and displacement fields and the expression of the Total Potential Energy as a function of the applied load: use beam theory, energy theorems, strength of material, linear elasticity theory, Rayleigh-Ritz method (kinetically and/or statically admissible method). (∗ ∗ ∗)
   - Analyze the stress and displacement fields, in order to determine stress concentration zones and high stress gradient zones from strength of material and linear elasticity: no demonstration, you can directly use the results from bibliographic sources[3] and comment them. (∗∗)
   - **Discuss your results**. (∗ ∗ ∗)

2. **Analysis with the Finite Element Method** (60%):

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

[1] Degree of relevance: (∗)= 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.

- Fix the rigid body modes[4], **if applicable**. $(**)$
- Check if the boundary conditions do not create singularities[5] in the finite element model: what can be said about the local convergence of the stress field if it is the case? $(***)$
- Use the symmetry of the problem (symmetry of load and geometry), **if applicable**. $(**)$
- Describe **precisely** the modeling of the mechanical problem on the NX software: load, boundary conditions, etc. $(*)$
- On the basis of the results from the strength of material analysis, mesh your structure using three different meshes[6] for first order triangular elements $T3$ and second order $T6$, and for first order quadrangular elements $Q6$ and second order $Q8$: $(***)$
  - Coarse mesh, i.e. no sub-domains;
  - Simple mesh, i.e with simple sub-domains;
  - Advanced mesh, i.e. with advanced sub-domains and/or varying distribution of finite elements among them.
- Show the advantages and disadvantages of the different finite element types used to discretise your mechanical problem, for example: $(**)$
  - approximation of the geometry : check the area/volume of your discretised model;
  - obtained results: check the spatial distribution of different components of the stress tensor and check if the boundary conditions are correctly respected.
- Justify the mesh quality using NX tools: Jacobians ratio, Aspect ratio, Skew angle, Taper, Corner Angle, etc. Use only the tools you find relevant. $(**)$
- 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

[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] 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).

[6]For each, specify the number of finite elements, the number of degrees of freedom and the load applied on the mechanical problem.

[7]Use CPU time indicated after "END OF STRESS STORAGE ON U18 FILE" in the .res file.

elements, for each element type and for each type of mesh[8], comment only the graphs you find relevant. **From those results, deduct a compromise on your mesh (global vs. local convergence)**. $(*)$

- Discuss your results (stress and displacement fields) and **compare them with the results from the strength of material analysis**. $(***)$
- Study the sensitivity of your results (stress and displacement fields) to the size, distribution and type of the finite elements. Analyse the local convergence of the stress field! $(***)$

3. **Optimization** (20%):

   - Answer the problem imposed by your statement with three different optimizations. 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 discretisation, especially if the optimization drastically affects the creation of an advanced mesh). $(***)$
   - Optimize the shape and the dimensions of the geometry marked in red on the structure in your personal project statement: use circles, lines, ellipses or parabolas for the optimization, **if applicable**. $(**)$
   - 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). $(**)$
   - Justify the quality of the meshes every time. $(**)$
   - Submit a final drawing of the most optimized structure, from the three trials, with dimensions. $(*)$

4. **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.

[8]Regroup similar information on the same graph.