

Faculty of Maritime Engineering and Marine Sciences

Finite Elements

Final Project

Jan. 9th, 2023

Team work

The objective of this project is to analyze the structure assigned to each group using ANSYS program applying finite element method under static loading. First use Rhinoceros software to prepare the geometry plan. Then after importing the geometry from ANSYS, define the adequate mesh for the structure. Establish support on the boundaries and apply at least two loading conditions; one of these can be the structure with reduced thickness due to corrosion. Before analyzing results, you must check convergence and make sure that your structural model is correct (“results make some sense”).

To present results, prepare a report which without considering cover page, index and appendix, cannot be longer than 6 pages, and you are required to do an oral presentation (some groups will be randomly selected to participate in the XIX NEST Sessions). To prepare the report, consider the recommendations previously explained in class and with suggested content: I. Introduction, II. Development of the model, III. Results and analysis, and IV. Conclusions and Recommendations.

Evaluation: Oral presentation (quality of material, english communication and answer to questions): 50%, and report (Aesthetics, completeness and clarity, and analysis of results) 50%. For the grading, this is a competition, so from the best and down. Research will be considered, so when preparing the report emphasize on that aspect.

Progress controls: whole group has to participate, and on each session one member is responsible for all the explanations.

Thursday Jan. 12nd: geometric model using Rhino

Monday Jan. 16th: structural model in Ansys and first results

Thursday Jan. 19th: final results.

Due dates:

Report: 8 am Wednesday Jan. 25th, loaded in AulaVirtual and printed copy, and **15-min oral presentations** from 8-10 am on that day, together with Mechanical Engineering student FE projects.