

# Problem Set 0

**Paper Problem** This is the first kind of problems that you will have to solve in this class. These problems, as the title suggests, will be solved on paper and their purpose is twofold.

1. You will have the chance to put the theory we introduced in class to the test by using what you've learned to solve relatively simple problems. You will better understand the concepts and techniques by working out the calculations yourself!
2. They provide a great opportunity to prepare you for the paper problems that you will eventually have to solve in your midterms later on in this class

The paper problems in your problem sets may not be the most exciting problems that you are going to solve but they are great for warming you up when you start solving a problem set. Most of the things you are going to apply in paper problems will be necessary for the next two kinds of problems as well. ■

**Matlab Problem** This is the fun part! Throughout this course you will develop a Finite Element code in Matlab from scratch. Starting from nothing, in Problem Set 1, you will continuously develop your FEA code, one step at a time, until you're able to solve challenging and interesting problems for multiple structures and element types!

The best part of this kind of problems is that the code is yours; you will be the one to develop it and you will understand each and every line of it. Either you plan to use this code later on (i.e. in a future project, research) or not, it will definitely be a great experience to develop your own little "software", step by step, continuously enhancing its capabilities. In-house codes are great and trust us: *You never know when you might find them useful!* ■

**Abaqus Problem** The final kind of problems that you will have to solve in this class will be on the ABAQUS © general purpose finite element software. In-house codes are great, but when it comes to solving very complicated problems (like the ones that often companies have to solve, or new research breakthroughs) it is much more convenient to have access to the extended and powerful toolbox and multiple functionalities of a commercial FEA product like ABAQUS.

Developed by engineers in Providence, RI back in the '80s it is now the gold standard when it comes to solving structural and solid mechanics problems. Whether you want to investigate the response of a greatly intricate and complicated structure or you want to model the behavior of a novel material, ABAQUS can help you do it! That is, of course, if you know how to make it work for you! This is why in this course you will be exposed to some of the main ABAQUS features in order to understand how to use it, double check your results from your in-house Matlab code, and extend your FEA solving abilities by challenging yourselves to more difficult problems for your project or future research. ■

All homework needs to be submitted through Gradescope. Please avoid submitting low resolution images of your solutions. When submitting your solutions to the problems sets make sure you include all files relevant to your solutions. Such files may include: README, PDF of solutions, Matlab codes, Abaqus CAE or Input files, figure files, Python script files, Mathematica notebook files etc. Please make sure to fully comment your code prior to submission.

## Tasks

- Ensure that you have access to Abaqus either locally or through FASRC.
- Ensure that you have a recent version of Matlab installed on your computer