CE 595: Finite Elements in Elasticity

Term Project

Total: 5+10+10 = 25 points + upto 3 Bonus Points for going beyond the requirements.
(The grading scheme is flexible. Points may be adjusted after submission of the Final report.)

In this project, you will use a commercial finite element software (such as ABAQUS) to study a problem of practical interest. You may ***collaborate*** with your fellow students to discuss and exchange modeling strategies, results and insights. However, you must submit an ***individual*** report. Requirements, guidelines and some suggested problems for the term project are given below. If you are unsure about any of the requirements, come speak with me ***asap***.

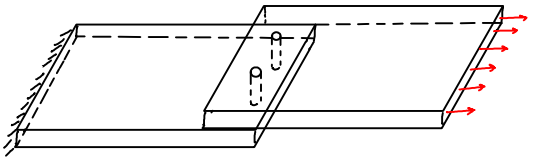
Suggested Problems:

1) Bolted / Riveted Connection
2) Simply Supported Reinforced Concrete Beam
3) Beam-Column Connection
4) Defective Corbel
5) Composite layered panel
6) Soil-Structure Abutment / Piles
7) Topics related to your research
8) Alternate 2D Coding Projects:
   a. Coupled thermal-structural code
   b. Dynamics
   c. Non-linear material – plasticity
   d. Large deformations

   You will need to verify your code with ABAQUS for simple problems.

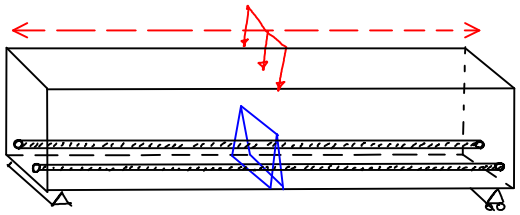
General Guidelines:

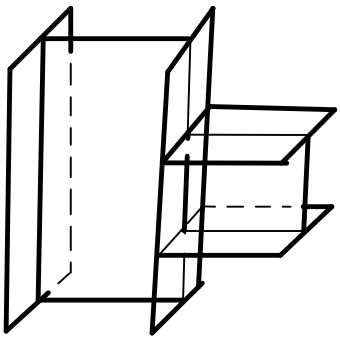
1) Practical problem of interest (2D/3D) to study ***physical*** and/or ***numerical*** problem parameters
2) Problem parameters
   a. Different types of geometry
   b. Different load types and/or stress states
   c. Different materials properties
   d. Different levels of "complexity" of finite element models
3) Verification
   a. Hand calculations
   b. Mesh refinement
   c. Limitations of the model
      - How to improve your model
      - How to get comparable results from a simpler model
4) Emphasis on "Discussion and Insights" from the results



Bolted / Riveted Connection
How to model?
Effect of
- Closeness of holes
- Arrangement of holes
- Relative modulus of bolt and plate
- Overlap length
- Loading mode (Tension, shear, torsion)
- ...

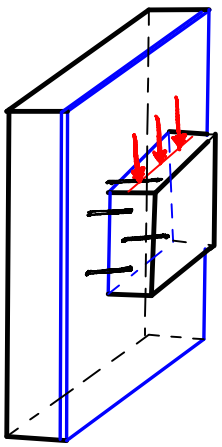
Reinforced Concrete Beam
How to model?
Effect of
- Size & number of bars
- Placement of reinforcement
- Modulus of concrete and/or steel
- Depth / width of the "numerical crack"
- Number of "numerical cracks"
- ...





Beam-Column Connection

Defective Corbel



CE 595: Finite Elements in Elasticity

The project will consist of 3 stages:

1. **Proposal** (5 points) : Due April 7, 2010
   Write a 2-3 page proposal describing the problem you want to study. Your proposal must include the following information:
   - Problem Title (and any collaborators).
   - Problem Statement – Just saying "Finite element analysis of ___" is ***not*** enough. You need to ***study*** a practical problem, such as "Effect of ___ on ____".
   - Define the problem parameters that you want to study. Define the geometry, material properties, different combinations of loading and boundary conditions and any other relevant information. (You may make reasonable changes to the problem later, if needed.)
   - Define 3 different levels of models or finite element ***meshes***.
     (Try to have at least one model with less than 1000 nodes. In addition, try to have a "monolithic" model with minimum number of "interacting" parts. Keep in mind that the bigger meshes will take longer to run and there are limited resources for ABAQUS in the ECN labs. You may run bigger problems as a group and share the results. However, you must run the smaller problems by yourself).
   - Expected behavior & justification – What behavior do you ***expect*** to capture with your model? Is your model capable of capturing this behavior?
   - Bonus: Any preliminary results.

2. **Progress report** (10 points) : Due April 19, 2010
   Build upon the proposal providing additional details and results from your analyses. Your report must include the following: (in addition to the information from the proposal)
   - Any changes to the Problem Statement
   - Introduction and brief literature survey
   - Brief theoretical background on your specific problem. You may include some information about the finite element method but you do not need to describe the derivation in detail.
   - Describe your finite element model. Describe, in detail, any advanced capabilities of ABAQUS that you may be using, such as non-linear material models, contact constraints etc.
   - Results: Summarize the results you have obtained thus far.
   - ***Verification***: Show calculations to support the results obtained from your model.
   - Discussion: Are the results, what you expected? How would you improve the model? Can you capture the same behavior with a simpler 2D model?
   - Bonus: ***Correct*** use of any advanced capabilities. Any preliminary insights into the behavior of your model vs. reality.

3. **Final Report** (10 points) : Due **Finals week**, latest by 5pm, May 7, 2010
   Your final report must be in the form of a technical report. The following section headings are suggested. Feel free to modify as needed.
   - Problem Statement
   - Introduction
   - Theoretical Background
   - Finite Element Model
   - Verification
   - Results – Summarize all the results from the analyses you conducted. Your results must be presented in a logical form (tabular or x-y plots) clearly showing the "Effect of ___ on ____" that you were studying. You should also include representative plots of the finite element deformation / stresses etc. to show the typical behavior. You may attach supplementary plots in appendices.
   - Discussion & Insights – What did you learn from your project? This is the ***main*** section for this stage of the project. Make sure you do a thorough appraisal of the finite element results you obtained.
   - Conclusions – Summarize your findings. Make recommendations for future studies.

There is no minimum or maximum page limit. You do not need to "fill-up" pages. ***Brevity*** is important. Write in a concise manner. Your report should be ***organized*** in a logical manner.