Can't you see the stress I'm under?

Objectives:

After completing this lab, you will be able to:

- Perform simple beam and shell stress analyses with Abaqus finite element software.
- Identify some of the limitations of finite element analysis.

Overview:

During this lab, you will use the ABAQUS finite element software to analyze the example frame structure from Lab 1 (below) for stress and deflection. First, you will create a "beam" finite element model in ABAQUS, and compare results to your hand calculations. Then, you will use a 2-D planar quadrilateral element model of the same frame to see how element choice may affect the predicted results. Finally, you will report your observations in a 1-page summary.

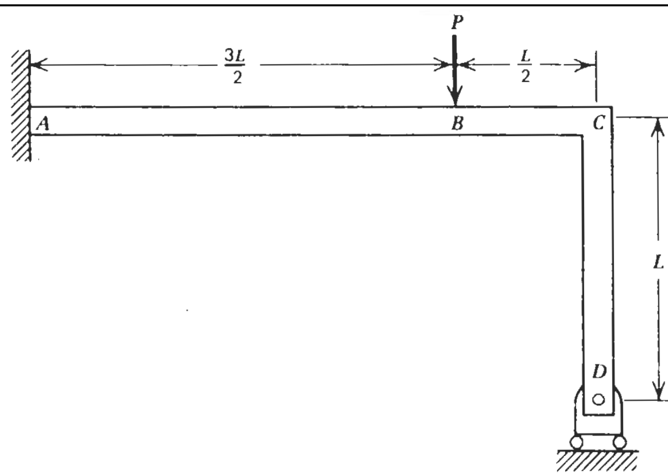
Assignment:

(1) FEA results need to be checked for validity before you can accept them – too many things can go wrong in the process for you to take results at face value! Using your answers from lab 1, calculate the vertical reaction (V) and horizontal deflection (q_H) at point *D*, for dimensions L = 1 m, P = 1 kN, E = 207 GPa (steel), and a solid square cross-section, 50 x 50 mm. Also, determine the maximum bending stress and location.

Member ABCD lies in the plane of the paper. If length L is large compared with the depth of the member, determine the pin reaction V at D and the horizontal displacement q_H of the pin at D.

Ans: $V = 81\,P / 128$

$q_H = 9\,P\,L^3 / 64\,E\,I$



(2) Now, perform a <u>beam element</u> Finite Element simulation of this problem by following the procedure which starts on page 3 of this lab. Compare your FE reactions, deflection, and stress results to your hand calculations.

Note that the software does not keep track of units – it's up to the user to define and stick to a consistent set of units. Here, I chose SI (m,s,kg). This is a good consistent set, although not always the most convenient for every problem.

(3) After you've completed the beam model analysis, develop a 2-D planar <u>quadrilateral element</u> model of the same problem, following the instructions which start on page 7. Compare your results for deflection, stress, and reaction forces. For consistency, I suggest using the stress components (S11) rather than the stress invariant (S, Mises).

(4) Run the solid element analysis again with a mesh size of 0.025, and then 0.0125. Record the results for each case. This is an example of decreasing the element size to increase accuracy (but for larger models the analysis takes longer to run – why?).

(5) One more iteration: Return the element size to 0.050, but change the element type to 'quadratic'. Run this analysis and record the results. This is an example of increasing the degree of the displacement field to increase accuracy (but, again, the analysis takes longer to run – why?). Another option to increase accuracy is to uncheck the "reduced integration" box when defining the element type.

Deliverables:

At the beginning of lab 4 (October 11), turn in:

- A 1-page executive summary, including:
  - Table comparing hand calculation and all FE simulation results.
  - Discussion of which modeling setup worked best.
  - Discussion of incorrect results, and how you might avoid them.
  - Discussion of any general trends you observe.
- Appendix: Attach the *deformed stress contour plot* of your best solid model result.

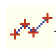
Start Abaqus by double-clicking the "ABAQUS CAE" icon on the desktop.
Select Create new Model Database option.

Define Part Geometry

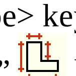
Make sure Module pull-down list reads Part

"Create Part"  button *[Text in quotes represents icon buttons. Hover over the icons to the left of the viewport to see an icon's title]*

Choose a name (e.g. Beam 1)
Modeling space: 2D planar *[Underlined text should be entered or selected]*
Type: Deformable
Base feature: Wire *[Beam elements are defined by lines describing their centerlines]*
Approximate size: 10 *[meters – this defines the overall dimension for the sketch space]*
Continue

"Create Lines: Connected"  button

Draw a single horizontal and vertical line to the dimensions given (2.0 wide, 1.0 tall).
This describes the 1-D beam elements, at the centerline of the actual beams *[m]*
Press <Escape> key when done.

"Autodimension"  button

Holding <Shift> key, use pointer to select both lines
Done *[at bottom of screen]*

Check dimensions and use "Edit Dimension" button if need to change. Note that if you use "Edit Dimension," you will have to 'Cancel'  out of that command when finished.

"Auto-Fit View"  button to fill the view with the part.
Done

SAVE YOUR FILE

Define Material and Section Properties

Change Module pull-down list to Property
"Create Section" button

Choose a name (e.g. Beam Prop)
Category: Beam
Type: Beam
Continue *[Edit Beam Section window opens]*

Section integration: Before Analysis
Profile Name: Create button

Choose a name for the profile/shape (e.g. square):
Shape: select Rectangular
Continue
a: 0.05, b: 0.05 *[meters]*
OK

Profile Name: Choose the one you just created.
Enter properties: Young's Modulus: 207e9 *[Pa]*, Shear Modulus: 79e9 *[Pa]*
Section Poisson's ratio: 0.3
"Output Points" tab *[to output stress at points off the neutral axis]*

x1: 0, x2: 0.025 <Enter> *[top center of the cross-section]*
x1: 0, x2: -0.025 <Enter> *[bottom center of the cross-section]*
OK

"Assign Section" button
Hold <Shift> key and select both lines
Done
Section: select the section property you just created (e.g. Beam Prop).
OK
Done

"Assign Beam Orientation" button
Select both lines
Done
Enter an approximate n1 direction: 0,0,-1 <Enter> *[this selects the x1 section direction (from your rectangular section definition) in line with the global –Z direction]*
OK *[important – this confirms your input]*
SAVE YOUR FILE

Bring Part into Assembly

Change Module to Assembly

"Instance Part" button
Make sure the part you created (e.g. Beam 1) is highlighted.
Instance type: independent
OK

Define Analysis Method

Change Module to Step

"Create Step" button
Type in a step name (e.g. Step-1)
Procedure type: General
Highlight Static/General in list
Continue…
OK *[accept all defaults]*
SAVE YOUR FILE

Define Loads & BC's

Change Module to Load *[only one part, so we can skip the "Interaction" module]*

Click and hold "Partition edge: Specify parameter by location" button, then select the "Partition edge: Enter parameter" button from the choices
[An icon button with a black triangle has other alternatives hiding beneath]
Select the top beam
Done
Normalized edge parameter (0<t<1): 0.75 *[create a point ¾ of the way down the beam. NOTE: if arrow points to the left, you should enter 0.25 to create the correct point]*
Create Partition

Done

"Create Load" button *[define the vertical applied force]*
Enter a name (e.g. Load-1)
Category: Mechanical
Highlight Concentrated Force in list
Continue…
Select the point ¾ of the way from the left end of the top beam.
Done
CF2: -1000 *[Specify 1000 Newton vertical downward force]*
OK *[you should see an arrow in the right direction in the viewport]*

"Create Boundary Condition" button *[define a constraint]*
Enter a name (e.g. BC-1)
Category: Mechanical
Type: Symmetry/Antisymmetry/Encastre
Continue
Select the point at left end of the top line
Done
Select "Encastre" from the list *[no translation or rotation]*
OK *[a symbol appears indicating the BC]*

"Create Boundary Condition" button, again *[define the other constraint]*
Enter a name (e.g. BC-2)
Category: Mechanical
Type: Displacement/Rotation
Continue
Select point at bottom end of the vertical line
Done
Select check-box next to U2, and enter 0 in text field *[no vertical translation]*
OK *[a symbol appears indicating the BC]*

SAVE YOUR FILE

'Mesh' (Discritize) Geometry

Change Module to Mesh

"Seed Part Instance" button
Approximate global size: 0.5 *[in meters – the average length of a beam element]*
OK *[markers show up indicating possible node locations]*
Done

"Assign Element Type" button
Select all three line segments (the top one has two parts, since you partitioned it)
Done
Element Library: Standard
Geometric Order: Linear
Element Family: Beam
Beam Type: Cubic formulation *[this neglects transverse shear, but is exact to beam bending theory. Later, try the Shear-flexible option to see how your results change]*

OK
"Mesh Part Instance" button
Yes *[The lines change color. With planar elements, you will see the element edges]*
SAVE YOUR FILE

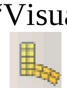
Start the Calculations

Change Module to Job
"Create Job"  button
Enter a job name (e.g. Job-1)
Continue…
OK *[accept defaults]*
"Job Manager" button
Submit *[now the software compiles the matrix equation, solves it, and prepares the results]*
(if asked whether to overwrite job files, select OK)
When status reads "Completed", select Results
[This reads the output files and takes you into the Visualization module]

Interpret the Results

This is the post-processing phase of performing FEA. These instructions will provide an introduction to how you view and record stress, reaction, and deflection data from a model. You will compare these to your hand calculations – an essential task in any FEA. In addition to these specifics, you should explore other aspects of the Visualization module – there is a lot of information to be gained here, and the best way to learn about it is to experiment.

Make sure Module is "Visualization"

"Plot deformed shape"  button *[see the deformation scale factor – the software scales the deformation, sometimes exaggerating the results. Why aren't the beams curved?]*
Select "Result" … "Section Points…" from the menu bar. *[This will allow you to look at bending stresses. Beam elements only report bending stress if you specify section points]*
Select the "Bottom" radio button
Field Output button
Output Variable: S Stress components at integration points
Component: S11 *[axial stress (σ_x) for each beam]*
OK
If "Select Plot State" window appears, select "Contour" option, then OK
OK to exit Field Output window.

"Query information"  button on top toolbar *[circle around an "i" on the top tool bar]*
Visualization Module Queries: Probe values *[opens Probe Values window]*
Make sure "Probe:" is set to "Nodes" and box next to S, S11 is checked.
Move this window to the side & position the cursor over different nodes to record the max. axial stress along the bottom edge of each beam (shows in "Probe Values" window).
Field Output… button
Highlight S then Mises *[reports von Mises stress invariant]*
OK

Make sure the box next to S, Mises is checked.
Position the cursor over different nodes to compare to S11 results.
Field Output… button
Highlight RF then RF2 *[Y reaction]*
OK
Make sure "Probe:" is set to "Nodes" and box next to RF, RF2 is checked.
Position the cursor over the end nodes and record reaction forces.
Field Output… button
Highlight U then U1 *[horizontal deflection]*
OK
Make sure "Probe:" is set to "Nodes" and box next to U, U1 is checked.
Record the X-deflection at the lower reaction.

Note that this is the most accurate prediction of deflection you will get from FEA. Although a cubic beam element model (like this one) provides limited stress information, its prediction of deflection is usually an exact match with beam theory (providing the model is built correctly).

RETURN TO THE LAB DESCRIPTION (Page 1)

Now, let's make a different model of the same problem, using planar quadrilateral elements. These instructions will be less specific – rely on your experience with the beam elements to help you develop this FEA model. Remember to save often!

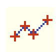
Create a new model by selecting "File … New" from the pull-down menu. Save your old model.

Define Part Geometry

Change Module to Part

"Create Part"  button

Modeling space: 2D Planar
Type: Deformable
Base feature: Shell *[2-D planar elements will describe an area, rather than a line]*
Approximate size: 10 *[meters]*
Continue…

"Create Lines: Connected"  button

Draw an outline of the frame. Use the dimension buttons to define the sizes. Make sure you consider the thickness also when you are defining the dimensions – the listed dimensions in the figure are to the *centerline* of each member.

Done

Define Material and Section Properties

Change Module pull-down list to Property

"Create Material"  button

Pull-down menu below "Material Behaviors" - "Mechanical … Elasticity … Elastic"
Young's Modulus: 207e9 *[Pa]*
Poisson's ratio: 0.3
OK

"Create Section" button

Category: Solid
Type: Homogeneous
Continue…
Select the material you created.
Plane stress/strain thickness: 0.05 *[depth of part into screen]*
OK

"Assign Section"  button

Select the beam system
Done
Select the section you created.
OK

Assemble Parts into Workspace

Change Module to Assembly

Bring an instance of your part into the assembly just like you did before.

Define Analysis Method

Change Module to <u>Step</u>

Create a “static, general” step just like you did before.

Define External Loads

Change Module to <u>Load</u>

“Partition Edge: Enter Parameter”  button (may be hidden, like before)

Select the top edge of the top beam (<u>Done</u>)

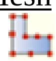
Normalized edge parameter (0<t<1): <u>0.7407</u> (<u>Create Partition</u>) *[Why not 0.75?]*

Use the same procedure as before to place a vertical load to this point.

Assign boundary conditions as before. Now you should constrain the entire left edge from translation (2-D planar elements nodes can’t support rotation constraints). You can use the “Partition Edge” function to put a point at the middle of the lower edge for the lower support constraint, or just constrain the left or right point.

‘Mesh’ (Discritize) Geometry

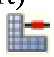
Change Module to <u>Mesh</u>

“Seed Part Instance”  button

Approximate global size: <u>0.05</u> *[meters – this will be the average element size]*

<u>OK</u>

<u>Done</u> (below viewport)

“Assign Mesh Controls”  button

<u>OK</u> *[accept defaults]*

“Assign Element Type”  button

Element Library: <u>Standard</u>

Geometric Order: <u>Linear</u>

Family: <u>Plane Stress</u> *[why not plane strain?]*

Quad: check “Reduced integration”

<u>OK</u> *[accept other defaults]*

“Mesh Part Instance” button

<u>Yes</u>

Look at the mesh – is it uniform? Uniformity is often a problem with automatic meshers, especially for angled geometry and 3-D parts. You can change the mesh by varying the number of ‘seed’ nodes on an edge using the buttons hidden under the “Seed Part Instance” button.

Start the Calculations

Change Module to <u>Job</u>

Create and submit a job like you did before. When complete, click <u>Results</u>

Interpret Results

Make sure Module is “Visualization”

“Plot deformed shape” button *[look at the deformation scale factor]*

“Plot contours on deformed shape” button *[look at stress scale]*

Menu bar: "Result … Field Output". Change the type of stress you are viewing. S11 is the normal stress in the global X (1) direction, S22 is the normal stress in the global Y (2) direction. S, Mises is the von Mises stress invariant.

Use "Query information" to get the stress, deflection, and reactions at the critical points, as before. Record this information.

2-D Planar elements provide a much more satisfying 'picture' of the stress distribution, but they require A LOT more elements to be accurate. We will now vary the number and type of these elements to try to achieve an accurate output.

RETURN TO THE LAB DESCRIPTION (Page 2)