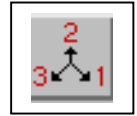


## 2.31 Assignment 11

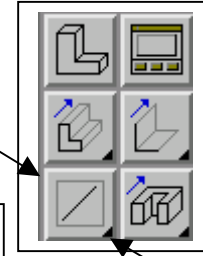
Due Wed, Nov 21 at 9:30 am

Please read Handout 12 (Using Beam Elements) carefully. Follow the step-by-step instructions in section 6.4 to create a model of the cargo crane. I have noted here some hints and suggestions that might help you in the process:

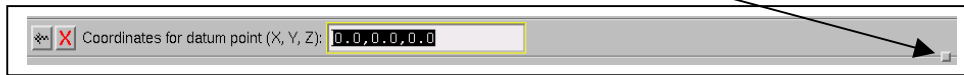
1) Bottom of page 6-14: when you create the datum points change the view to isometric



2) Point 3. on page 6-15: the Create Wire:Planar tool is this

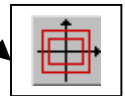


3) **Tip** at the bottom of page 6-16: to resize the message area, click and drag the little square on the top right



4) Point 7. at the top of page 6-17: to get the Create Wire:Two Point tool click on the triangle

5) Point 1. at the bottom of page 6-17. The symbol for the tool that creates the profile is



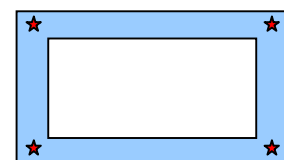
6) When you define the load on page 6-23 the magnitude should be -10000 ( you are loading in the negative y-direction)

Complete the tutorial.

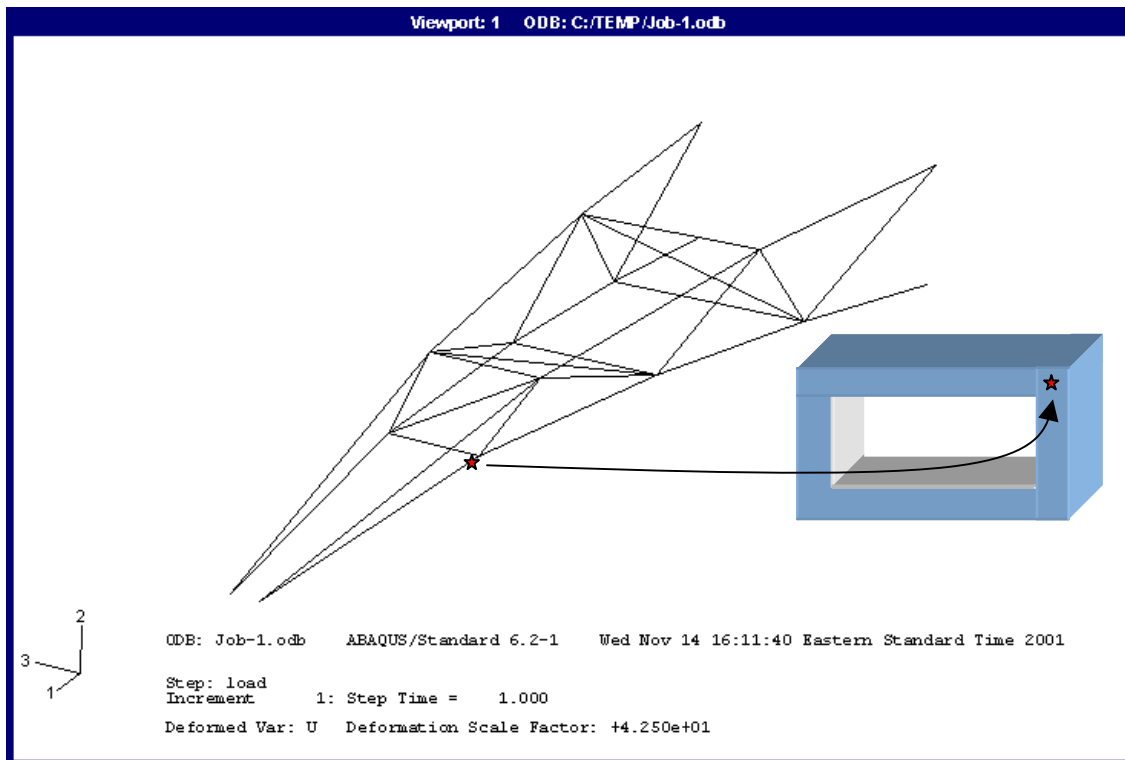
Print and attach to your homework your plots corresponding to Figures 6-23, 6-24, 6-25.

Now for some additional work:

- 1) Which portions of the main members are in tension and which are in compression?  
Contour plot (+print) the Section Force variable that allows you to answer this question.
- 2) Create a Display Group that only includes the cross bracing. Plot (+print) SM1 for the cross bracing. You remember you have attached the bracing to the main trusses with pin joints that cannot transmit moment. But now in your plot you see that there are bending moments in the beams of the cross bracing. Explain why. If in the real structure the cross bracing consists of individual straight beams separately bolted onto the main trusses, is our model consistent with the real structure?
- 3) Now we would like to estimate the max stresses in the structure. If you go look for stress in the Field Output you will not find it because it is not part of the outputs for beam elements when the section properties are evaluated before the analysis. The idea is that if you know forces and moments you can calculate the stresses. But we are lazy. So we go back to the Property module and in the section manager we edit both sections so that now the integration of the sections is done ♦ During analysis.  
Now the program will also output the stresses. By default, for a box section, it will output the stress at the 4 outer corners. (you can require additional locations in the Step module, using the Field Output Request Manager if you want)



You can rerun the analysis, and this time you can visualize the stress fields as well. What are the maximum and minimal principal stresses in the structure? Where are they located? → I want you to identify both the location along the beam axis, as well as the position of the section point, using a sketch like the one below. Explain how you figured this out. Note that the Abaqus identification (e.g., Bottom right corner) refers to the geometry of the cross section in the n1-n2 plane in which you have defined the geometry of the section and not to the actual position with respect to the global coordinate system.



- 4) Obtain reaction forces and moments at the four encastered points. These end points are welded to a massive support. Which welds, and wich regions of the weld would you inspect more closely for possible problems/damage/cracks? Please make a sketch to illustrate your answer.