



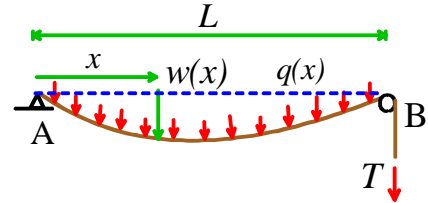
School of Engineering
Brown University

EN234: Computational methods in Structural and Solid Mechanics

Homework 8: ABAQUS user elements

Due Wednesday Nov 27, 2013

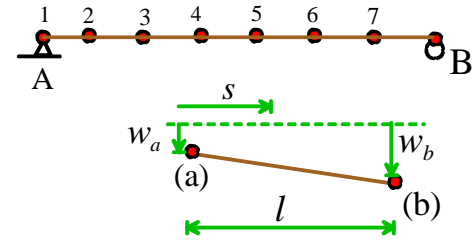
1. Write a UEL for ABAQUS that implements a nonlinear element for a tensioned cable, shown in the figure. The cable is pinned at A, and passes over a frictionless pulley at B. A tension T is applied to the end of the cable as shown. A uniform distributed load q per unit length of cable causes the cable to deflect by a distance $w(x)$ as shown. The FEA code must calculate a discrete approximation to w .



You will need to set up a system of finite element equations for the deflection. It is simplest to use energy methods to do this: the potential energy of the system is

$$V(w) = \int_0^L T \sqrt{1 + \left(\frac{dw}{dx}\right)^2} dx - \int_0^L q w \sqrt{1 + \left(\frac{dw}{dx}\right)^2} dx$$

To develop a finite element scheme to calculate w , you can divide the cable into a series of 1-D finite elements as shown. The variation of displacements within an element can be interpolated linearly. The total potential energy can then be expressed as



$$V(w^a) = \int_0^L T \sqrt{1 + \left(\frac{dN^a}{dx} w^a\right)^2} dx - \int_0^L q N^b w^b \sqrt{1 + \left(\frac{dN^a}{dx} w^a\right)^2} dx$$

The (nonlinear) equilibrium equations can then be computed as

$$R_a(w^b) = \frac{\partial V}{\partial w^a} = \int_0^L \frac{T}{\phi} \frac{dN^b}{dx} w^b \frac{dN^a}{dx} dx - \int_0^L q N^a \phi + \frac{q}{\phi} N^b w^b \frac{dN^c}{dx} w^c \frac{dN^a}{dx} dx = 0$$

$$\phi = \sqrt{1 + \left(\frac{dN^a}{dx} w^a\right)^2}$$

The equation must be solved by Newton-Raphson iteration.

Your mission will be (a) To calculate a formula for the stiffness matrix (by differentiating the equilibrium equations with respect to displacements); (b) Write an ABAQUS UEL that computes the residual vector and stiffness matrix; and (c) set up a .inp file that will instruct ABAQUS to solve the problem. It is best to create the .inp file by hand (you can edit the example provided with the cohesive zone element)

Some suggestions:

- (1) The distributed load term in the equilibrium equations should be coded as a contribution to the RHS vector in the UEL, not applied as a DLOAD. It is a good idea to make the distributed load time dependent – i.e. set $q = \beta t$ where t is time, and β is the loading rate (which should be provided to

the UEL as a property). This will make sure that if ABAQUS takes small time-steps the load increments are small.

- (2) You may find it helpful to set up the problem in FEACHEAP or MATLAB before trying it in ABAQUS. Debugging is easier in those codes and in FEACHEAP you can use the CHECK STIFFNESS key to make sure your stiffness is consistent with the residual vector. But this is not essential if you feel confident about setting up the ABAQUS code correctly.
- (3) The cohesive zone element UEL provided as an example on the programming page of the course website already has the linear shape functions and the integration points you need coded (but you will have to modify the subroutines a bit because the cable elements only have 2 nodes)
- (4) Note that FEACHEAP and ABAQUS treat the displacement variables slightly differently. In FEACHEAP the variable utloc gives the total displacement at the start of an increment and duloc gives the current approximation to the change in displacement during the increment. In ABAQUS the variable U contains the current approximation to the displacement at the *end* of the increment, and DU(I,1) contains the increment in displacement. Thus U includes the change in displacement and is updated after each Newton-Raphson iteration.
- (5) When you define your element in the input file you should use the keyword
*USER ELEMENT, TYPE=U1, NODES=2, COORDINATES=1, UNSYMM

1

(if you prefer you can use DOF numbers 2 or 3 instead of 1 on the second line – but be sure you constrain the correct DOF number in the boundary conditions). This makes sure that ABAQUS knows your stiffness is unsymmetric.

- (6) It is not possible to use ABAQUS/CAE to visualize user-elements. There are various ways you can work around this, but it is simplest just to print the results to the .dat file. You can use the keyword below to accomplish this

*Node Print, freq=1

COOR1,U1

This will print the x coordinate and the displacement to the file.

To run ABAQUS with user-subroutines on a windows PC download the .bat file that sets environment variables, edit it appropriately, open an ABAQUS command window and navigate to the directory with your files, and then type `abaqus_usersub user=name of your Fortran file, without the .FOR extension`

Check the .msg and the .dat file for error messages. If the code produces something saying the LINK command wasn't found (or your code appears to do nothing) close the ABAQUS command window and open a new one and repeat the process.