



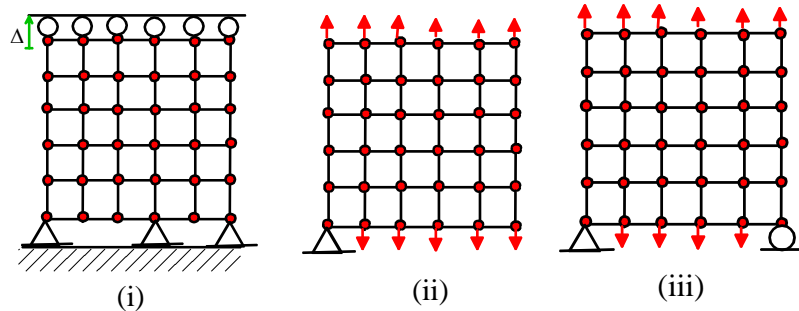
EN1750: Advanced Mechanics of Solids

Homework 1: Introduction to finite element analysis

Due Friday Sept 21, 2018

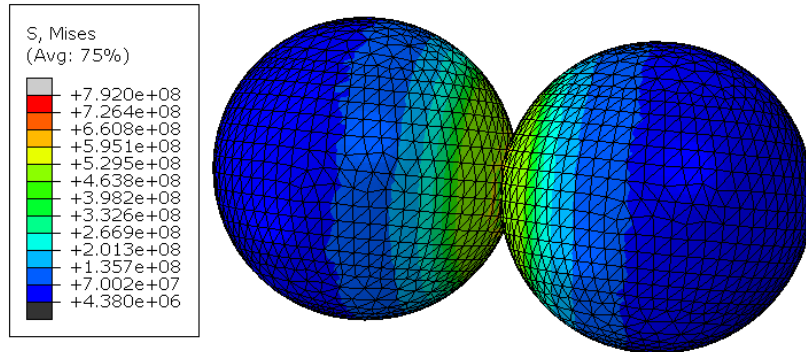
1. Please answer the following general questions about finite element analysis

- What is the difference between a static and a dynamic FEA computation (please limit your answer to a sentence!)
- What is the difference between the displacement fields in 8 noded and 20 noded hexahedral elements?
- What is the key difference between the nodes on a beam element and the nodes on a 3D solid element?
- Which of the boundary conditions shown below properly constrain the solid for a plane strain static analysis?



- List three ways that loads can be applied to a finite element mesh
- In a quasi-static analysis of a ceramic cutting tool machining steel, which surface would you choose as the master surface, and which would you choose as the slave surface?
- You conduct an FEA computation to calculate the natural frequency of vibration of a beam that is pinned at both ends. You enter as parameters the Young's modulus of the beam E , its area moment of inertia I , its mass per unit length m and its length L . Work through the dimensional analysis to identify a dimensionless functional relationship between the natural frequency and other parameters. You can assume that only the product EI appears in the governing equations, (i.e. E and I are not separate parameters).


2. Set up ABAQUS to conduct an explicit dynamic calculation of the impact of two identical spheres, as shown below.




Here's a summary of the model parameters:

1. Sphere radius 0.1m
2. Mass density $\rho = 1000 \text{ kg/m}^3$
3. Young's modulus $E = 210\text{GNm}^{-2}$, Poisson ratio $\nu = 0.3$
4. Analysis time period 0.0007sec
5. First, run an analysis with perfectly elastic spheres. Then repeat the calculation for elastic-plastic spheres, with yield stress $\sigma_Y = 10000\text{MNm}^{-2}$ $\sigma_Y = 5000\text{MNm}^{-2}$ $\sigma_Y = 1000\text{MNm}^{-2}$, $\sigma_Y = 500\text{MNm}^{-2}$, and again with $\sigma_Y = 50\text{MNm}^{-2}$ (you could automate this with a python script if you like, but its probably not worth the effort for just 5 simulations)
6. Contact formulation – hard contact, with no friction
7. Give one sphere an initial velocity of $v_0 = 100\text{m/s}$ towards the other sphere.

Follow these steps to set up the analysis

1. In the part module use Part> Create to create a 3D deformable body; use Solid/Revolution for the base feature. Enter 2 for the Approximate Size, then draw a closed semicircle with radius 0.1m in the sketch window. Add a construction line  down the y axis. Right click the window; select Cancel Procedure; then select Done. Then select the construction line to define the axis of revolution. Enter 360 degrees for the revolve angle
2. In the Property module, select Material > Create; then under the General tab enter the density; and under the Mechanical> Elastic tab enter Young's modulus and Poissons ratio. Later you will use Material>Manager to edit these properties and add a 'plastic' definition. To define a material with a constant yield stress you just enter the yield stress and 0 in the two columns of the table. Next, create a section (Homogeneous) (Section > Create, etc), with the material properties you just defined. Then assign the section to the sphere (Assign > Section, then select the sphere).
3. In the Assembly module you can create the two spheres by making two instances of the same part. When you create the second instance, check the button that says 'auto offset from other instances.' This will automatically place the two spheres with their centers on the x axis, a small distance apart.
4. In the Step module create an Explicit Dynamic step, with time period 0.0007s. You can use default options for everything else. Also use Output>Field Output Requests > Manager and edit F-output-1. In the Frequency menu change the Interval from 20 to 100.

5. In the Interaction module (i) Select Property>Create from the Interaction menu (ii) Select Contact in the menu and hit Continue (iii) In the next menu select the Normal Contact option under the Mechanical menu, and hit OK to accept the default model. (iv) Select the Tangential contact option and accept frictionless; (v) Then go to the Interaction>Create... (vi) Select Explicit (general contact) and hit continue (vii) Hit OK to accept the default settings.
6. You use the 'Load' module to assign an initial velocity to one of the spheres (this is a bit weird). Go to the Load module, select Predefined Field>Create. **Select Initial in the menu next to step near the top of the window**, then check the Mechanical radio button just below it. Hit OK. Select the leftmost sphere by **dragging a box around the entire sphere**. Don't click on it – if you do this it will only assign velocities to a piece of the sphere, not the whole thing. This causes serious problems in the analysis later. Hit `Done`. Finally enter 100 m/s for V1 in the appropriate box and hit OK.
7. In the Mesh module make sure the 'Part' radio button is checked above the viewport. Select Mesh > Controls, select Tet elements and press OK. The part should change color. Then select Mesh> Element type, and assign quadratic tet elements to the part (this should be the default, but you never know). Then seed the part with a mesh size of 0.01m and mesh it.
8. Create a job and submit the analysis. You can use Job>Monitor to track the progress of the computation – the 'Step Time' column will tell you how far along the simulation has got.
9. In the Visualization module open the odb file. You can select  to plot the deformed shape and Mises stress contours, and Animate to play a movie of the collision.

Please hand in a copy of graphs showing the variation of the total kinetic energy of the system as a function of time, for each analysis case. You can plot the graph by going to the Result>History output menu, and selecting the ALLKE variable, then pressing Plot. The easiest way to print them is to use the snipping tool and paste the graphs into a word document.

Finally, please answer the following questions:

1. Suppose that the main objective of the analysis is to compute the restitution coefficient of the spheres, defined as $e = \frac{|\mathbf{v}_{A1} - \mathbf{v}_{B1}|}{|\mathbf{v}_{A0} - \mathbf{v}_{B0}|}$ where $\mathbf{v}_{A0}, \mathbf{v}_{A1}$ denote the initial and final velocities of sphere A, and the same convention is used for sphere B. List all the material and geometric parameters that appear in the problem (include the initial condition).
2. Express the functional relationship governing the restitution coefficient in dimensionless form. Show that for a perfectly elastic material, the restitution coefficient must be a function of a single dimensionless group (and one additional dimensionless variable). For an elastic-plastic material, you should find that the restitution coefficient is a function of two groups and a variable.
3. If the sphere radius is doubled, what happens to the restitution coefficient? (DON'T DO ANY FEA TO ANSWER THIS!)

4. Show that the kinetic energy lost during impact can be expressed in dimensionless form as

$$\frac{\Delta K}{R^3 \rho v_0^2} = f\left(\frac{\rho v_0^2}{\sigma_Y}, \frac{\rho v_0^2}{E}, \nu\right)$$

Note that the second term is very small for any practical application (including our simulation), so in interpreting data we need only to focus on behavior in the limit $\rho v_0^2 / E \rightarrow 0$.

5. Use your plots of KE as a function of time to determine the change in KE for each analysis case. Hence, plot a graph showing $\Delta K / R^3 \rho v_0^2$ as a function of $\rho v_0^2 / \sigma_Y$ (you can use MATLAB or any other plotting tool to do this, you don't need to use ABAQUS).
6. What is the critical value of $\rho v_0^2 / \sigma_Y$ where *no* energy is lost? (you may find it helpful to plot $\Delta K / R^3 \rho v_0^2$ against $\log(\rho v_0^2 / \sigma_Y)$ to see this more clearly). If no energy is lost, the impact is perfectly elastic.
7. The usual assumption in classical mechanics is that restitution coefficient is a material property. Comment briefly on this assumption in light of your simulation results.