

## ABAQUS Tutorial – Plate Bending

Consider a circular aluminum plate ( $E=10e6$  psi,  $\nu=0.3$ ) of radius 10” and thickness 0.2”. The plate is simply supported around its outer perimeter and is subjected to a transverse pressure of 10 psi. Using plate (shell) elements, determine the deflection at the center of the plate. Plate theory gives the plate deflection as

$$w = \frac{PR^4}{64D} \left( \frac{5+\nu}{1+\nu} \right)$$

where

$$D = \frac{Et^3}{12(1-\nu^2)}$$

For our case, the predicted deflection is 0.290”.

### Finite Element solution (ABAQUS)

Start => Programs => ABAQUS 6.7-1 => ABAQUS CAE  
File => Set Work Directory => select folder for Abaqus generated files  
Select 'Create Model Database'  
File => Save As => save .cae file in Work Directory

#### Module: Sketch

Sketch => Create => Approx size - 50  
Add=> Circle => center point (0,0), perimeter point (10,0) => right click => Cancel Procedure  
=> Done

#### Module: Part

Part => Create => select 3D, Deformable, Shell, Planar => Continue  
Add => Sketch => select 'Sketch-1' => Done => Done

#### Module: Property

Material => Create => Name: Material-1, Mechanical, Elasticity, Elastic => set Young's modulus = 10e6, Poisson's ratio = 0.3 => OK  
Section => Create => Name: Section-1, Shell, Homogeneous => Continue => Shell thickness = 0.2 => Material - Material-1 => OK  
Assign Section => select entire part by dragging mouse => Done => Section-1 => OK

#### Module: Assembly

Instance => Create => Part-1 => OK

#### Module: Step

Step => Create => Name: Step-1, Initial, Static, General => Continue => nlgeom off => OK

### Module: Load

Load => Create => Name: Step-1, Step: Step 1, Mechanical, Pressure => Continue => select top face => Done => set Magnitude = 10 => OK

BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement / Rotation => Continue => select perimeter => Done => U1=U2=U3 =0

### Module: Mesh

Model Tree => Parts => Part-2 => double click on Mesh

Seed => Edge by Size => select entire model => Done => Element Size=0.5 => press Enter => Done

Mesh => Controls => Element Shape => Quad

Mesh => Element Type => Shell => Quadratic => OK => Done

Mesh => Instance => OK to mesh the part Instance: Yes => Done

Tools => Query => Region Mesh => Apply (*displays number of nodes and elements at bottom of screen – note: teaching license limit is 10,000*)

### Module: Job

Job => Create => Name: Job-1, Model: Model-1 => Continue => Job Type: Full analysis, Run Mode: Background, Submit Time: Immediately => OK

Job => Manager => Submit => Job-1

Results

### Module: Visualization

Plot=> Contours => Result => On Deformed Shape

Result => Field Output => Name - U => Component = U3 => OK

View => Graphics Options => Background Color => White

Common Options => Other => Translucency => unselect Apply translucency = OK

Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image

