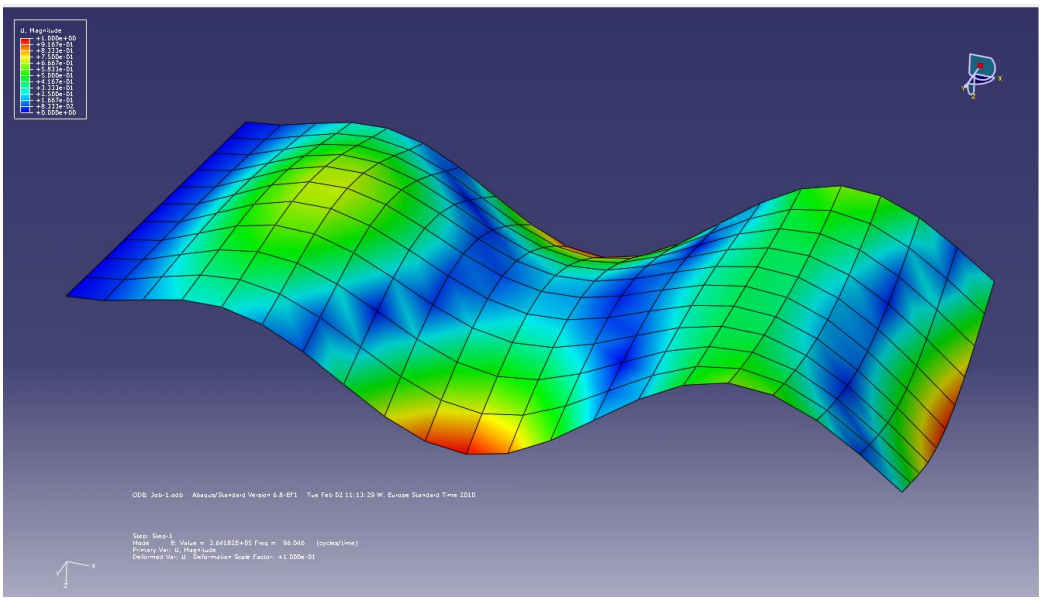
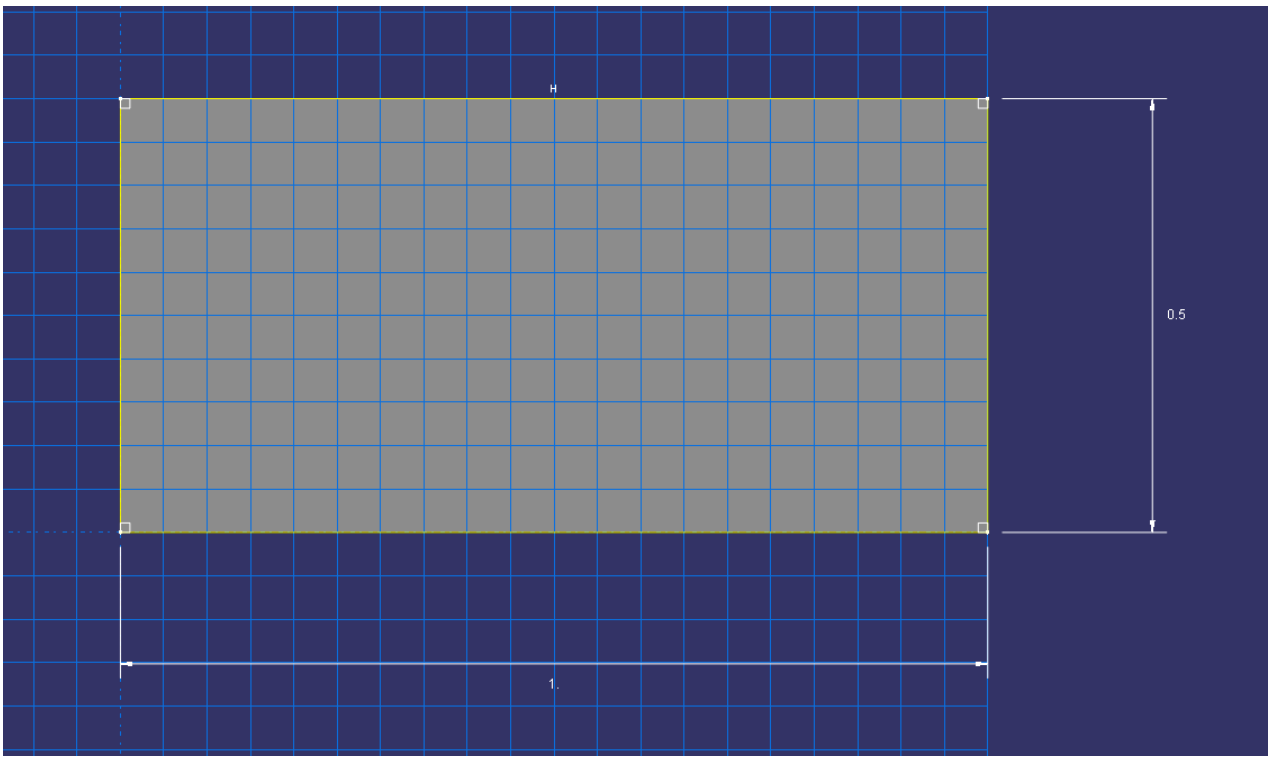
ABAQUS – Tutorial 4

Part module



1 Creating the plate

To create the plate (the base feature), you create a three-dimensional, deformable, shell Planar part according to the instructions below. The measurements of the plate are: L=1m, W=0.5m and t=3mm.



1. Start Abaqus/CAE, and create a new model database.
2. In the Model Tree, double-click the **Parts** container to create a new part.
3. Name the part Plate. Change to the following settings:
    - A three-dimensional, deformable body
    - A shell planar base feature
4. In the **Approximate size** text field, type 2. You will be modeling the plate using meters for the unit of length, and its overall length is 1 meter; therefore, 2 meters is a sufficiently large approximate size for the part. Click **Continue** to create the part.
5. From the Sketcher toolbox, select the rectangle tool  .
6. Sketch an arbitrary rectangle, and click mouse button 2 in the viewport to exit the rectangle tool.
7. Dimension the lower and left edges so that it is 1 m long and 0.5 m high.
8. Click mouse button 2 to exit the Sketcher.

Property Module

1. Create Material and name it, for example: Aluminum.
    - Mechanical, Elastic, Young's modulus = 70GPa, Poisson's ratio = 0.25.
    - General, Density = 2500 kg/m^3 .
2. Create Section and name it, for example: Aluminum_section ,
    - Choose: Homogeneous, Shell Section, Thickness=3mm.
3. Assign Section, Assign section to plate.

Assembly Module

1. Instance Part, Choose Independent (Mesh on Instance) => OK

Step Module

1. First create a step to determine the eigenfrequencies and the eigenmodes.
    - Procedure type: Linear perturbation, Frequency -> Continue.
    - Choose Number of eigenvalues requested: 10

Load Module

1. Create Boundary Condition, Symmetry/Antisymmetry/Encastre,
    - Select left boundary line.
    - Choose encastre (U1=U2=U3=UR1=UR2=UR3=0) => OK

Mesh Module

1. Assign Mesh Control, Choose:
    - Element Shape => Quad
    - Technique => Structured
2. Seed Part Instance, Approximate Global Size = 0.05 m.
3. Mesh Part Instance => Yes

Job Module

1. Create Job => Continue => OK
2. Job Manager => Submit => OK
3. When the analysis has completed, In the Job Manager => Results

Visualization Module

1. Plot Contours on deformed Shape.
    - Push  to choose which eigenmode to visualize.
    - The mode number and the eigenfrequency value is printed on the screen.
2. Animate the modes using Animate: Harmonic
    - Under Animation Options, Scale Factor: Full cycle.

Change analysis type

Dynamic solution from transient loading

1. In the step module, Step Manager, delete the frequency step.
    - Create Procedure type: General, Dynamic, Implicit -> Continue.
    - Time period: 1
    - In the Incrementation tab: choose Type: Automatic, Maximum number of increments: 1000, Increment size: Initial=0.01, Minimum=0.0001, Maximum=0.01.
2. Create Boundary Condition, Symmetry/Antisymmetry/Encastre,
    - Select left boundary line.
    - Choose encastre (U1=U2=U3=UR1=UR2=UR3=0) => OK
3. Create Load/ Concentrated force/Continue:
    - Select corner point on right boundary line.
    - CF3: 100
    - Amplitude: Create Smooth step:
        1: Time=0.0, Amplitude=0
        2: Time=0.1, Amplitude=1
        3: Time=0.2, Amplitude=0
    - Choose the created amplitude.
4. Re-mesh if necessary
5. Run the Job and show the results.

Modal reduction solution from transient loading

1. In the step module, Step Manager, delete the Dynamic, Implicit step.
2. First create a step to determine the eigenfrequencies and the eigenmodes to be used in the reduced model and then create a transient step based on the reduced model.
    - Create procedure type: Linear perturbation, Frequency -> Continue.
    - Choose Number of eigenvalues requested: 10 (or another choice.)
    - Create procedure type: Linear perturbation, Modal dynamics
    - Choose Time period: 1
    - Choose Time increment: 0.01
3. In the load module re-create the load and the boundary conditions following the procedure above "**Dynamic solution from transient loading**" from step 2 to 5.
   (Note that you can use the amplitude already created)