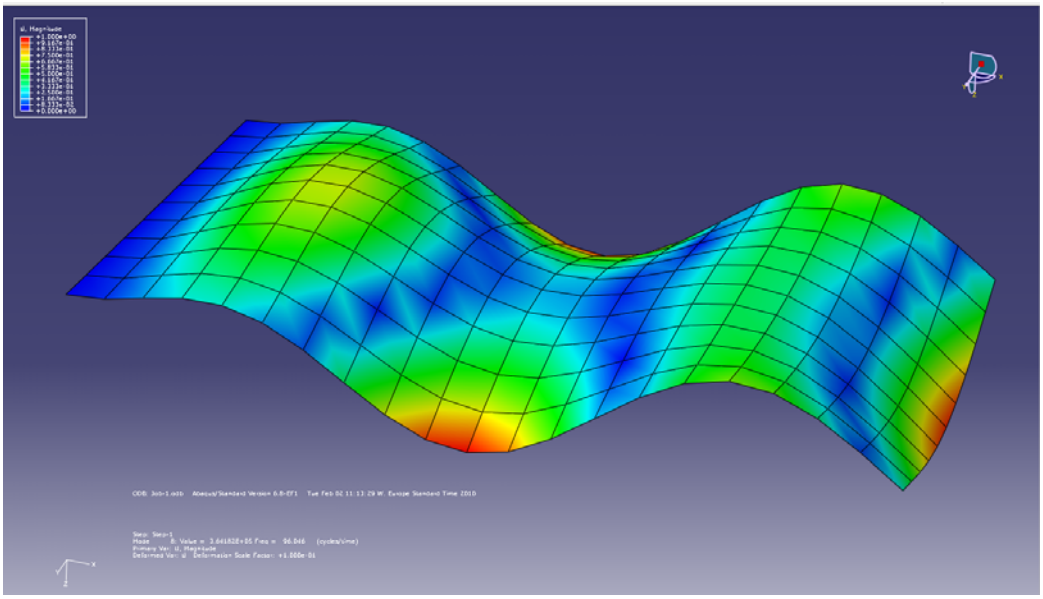
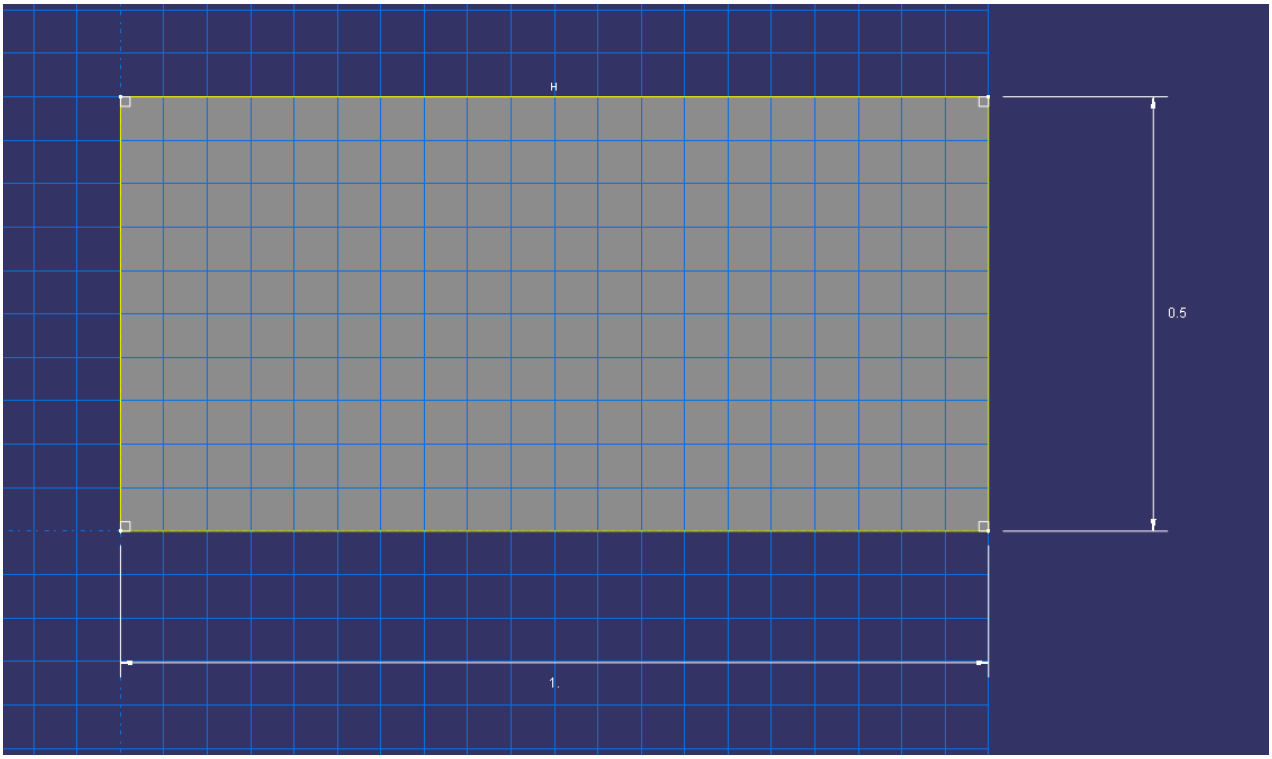
ABAQUS - Tutorial

Part module



1 Creating the plate

To create the plate (the base feature), you create a three-dimensional, deformable, shell Planar part and name it. You then sketch its profile (0.5 m × 1.0 m).



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

Steady state solution from harmonic loading

1. In the step module, Step Manager, delete the frequency step.
    - Create Procedure type: Linear perturbation, Steady-state dynamics, Direct -> Continue.
    - Choose Scale: Linear
    - In the data field: choose Lower Frequency 1, Upper Frequency 20, Number of points 20.
2. In the load module,
    - Create Boundary Condition, Symmetry/Antisymmetry/Encastre,
    - Select left boundary line.
    - Choose encastre (U1=U2=U3=UR1=UR2=UR3=0) => OK
3. Create a Concentrated Force at the two corner points that are opposite from the encastered boundary, Force=1000N in the z-direction.
4. Re-mesh the part if necessary, run the Job and show the results.

Dynamic solution from transient loading

1. In the step module, Step Manager, delete the steady-state 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.
    - Half-Step residual tolerance: 500.
2. In the load module,
    - Create Boundary Condition, Choose Displacement/Rotation
    - Select left boundary line.
    - Type in: U3=0.01m, U1=U2=UR1=UR2=UR3=0.
    - Amplitude: Create Smooth step: 1: Time=0 Amplitude=0; 2: Time=0.05, Amplitude=1
    - Choose the created amplitude.
3. Run the Job and show the results.