ABAQUS

ME 498CA1 Fall 2016

Loading & Analysis

ABAQUS Loading

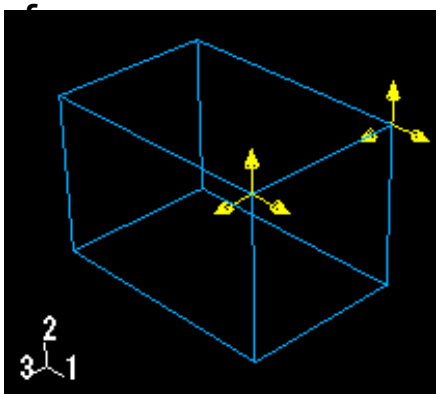
Primary References

- *Abaqus/CAE User's Manual*, §16
- *Getting Started with Abaqus: Keywords Edition*, §11
- *Abaqus Theory Manual*, §2, 6

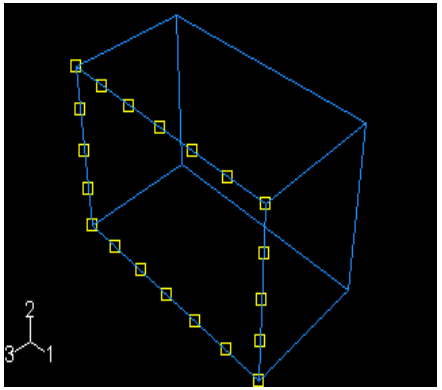
ABAQUS Loading

Prescribed Loads

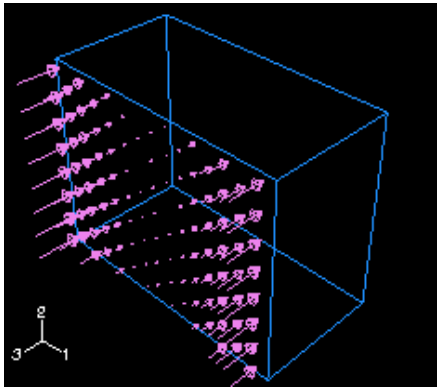
concentrated



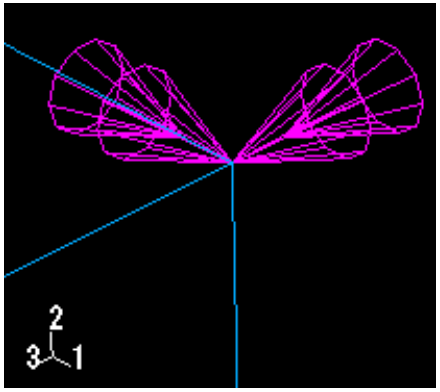
uniform field



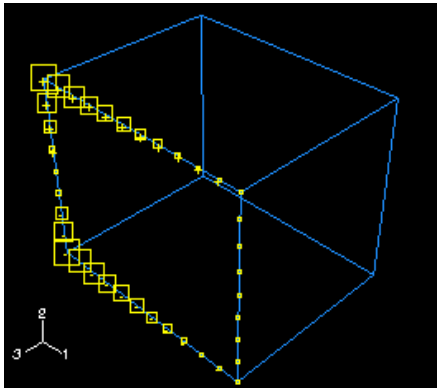
variable field



degree of freedom



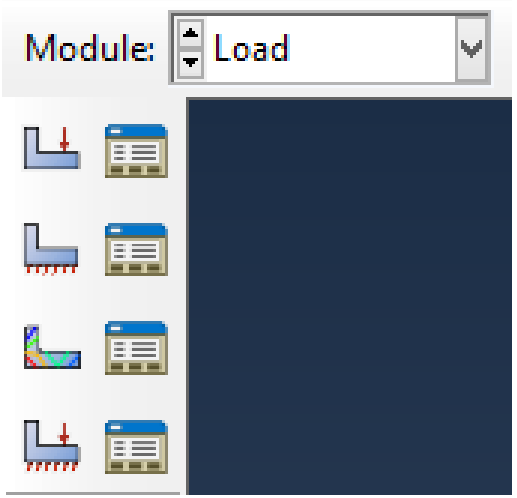
analytical field



ABAQUS Loading

Load Module

- Analysis steps (no BCs or predefined conditions)
  - Physical loads (point DOFs)
  - Boundary conditions (*d*/*v*/*T*)
  - Predefined fields (void ratio, temperature, exp'l data)
  - Load cases (sets of BCs + loads)



- Rigid body motion error:
  - static: "numerical singularity", "zero pivot"
  - dynamic: "excessive plasticity"

ABAQUS Loading

Step Module

- Initial step
- Analysis steps (no BCs or predefined conditions)
  - General steps (nonlinear/linear, standard/explicit)
  - Linear perturbation steps (linear/standard)

  Using the previous load/deformation state as a basis, simulate a linear perturbation.

- Loads are specified *absolutely*, not as additional accruals.

E.g., 1000 N→3000N should specify 1000, then 3000

(*not* 1000 then 2000)

ABAQUS Loading

Step Module—Load Manager

- Initial step
- Analysis steps (no BCs or predefined conditions)
  - General steps (nonlinear/linear, standard/explicit)
  - Linear perturbation steps (linear/standard)
  - Using the previous load/deformation state as a basis, simulate a linear perturbation.
- *Nlgeom*—persists across later steps
- Loads are specified *absolutely*, not as additional accruals.

E.g., 1000 N→3000N should specify 1000, then 3000

(*not* 1000 then 2000)

ABAQUS Loading

Step Module

- Total time

increases throughout all general steps

- Step time

begins at zero for each step

ABAQUS Loading

Output Requests

- Run complex simulations in stages
- Restart stores values in:

  `res` (Abaqus/Standard)

  `adq` (Abaqus/Explicit)

  odb of same name req.
- Frequency: every increment or interval by default
- The model used for a restart **must be identical** to the previous one in *Geometry*, *Property*, *Mesh*, prior *Steps*.

ABAQUS Loading

Restarts

- Field History
- History Output

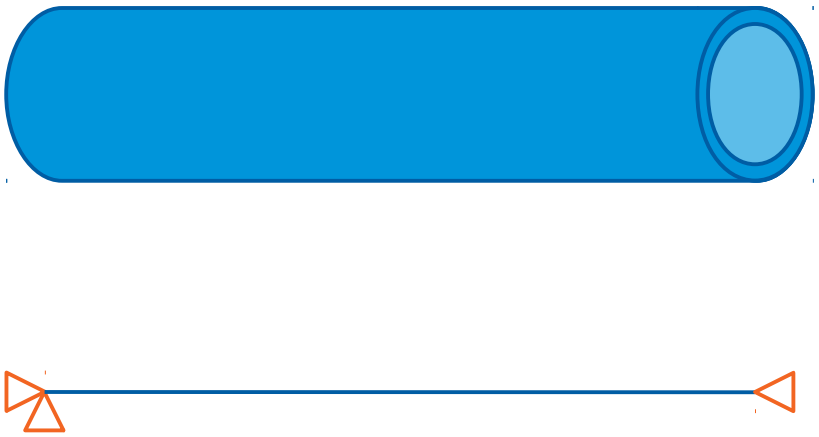
- **variables** of interest
- **region** to output
- **rate** of output

odb file—default output

`rpt` file—requested output (tabular)

ABAQUS Loading

Example: Modal Analysis



- 3D beam
- L=5m

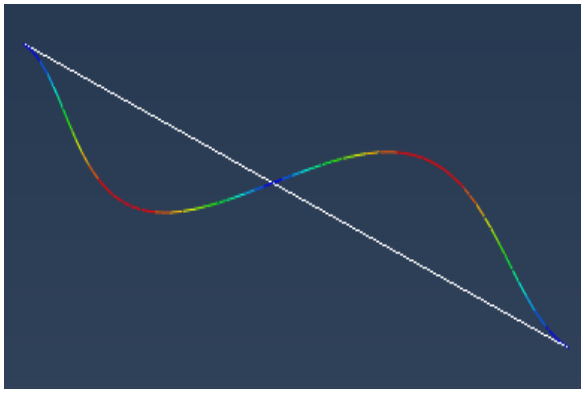
D=0.18m

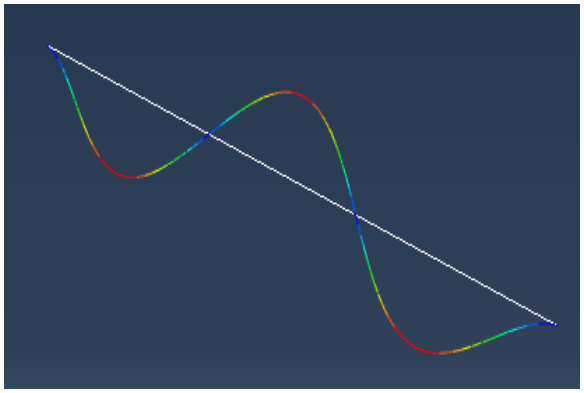
t=0.02m

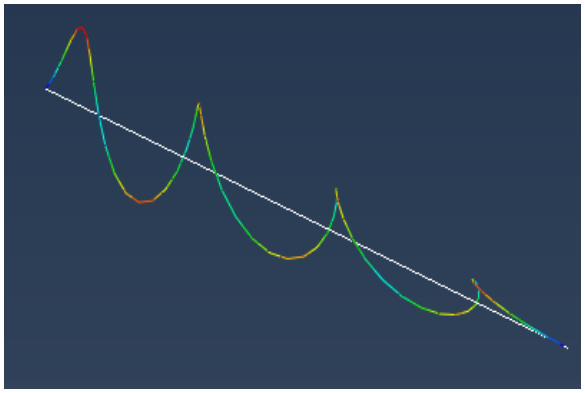
- need load such that lowest vibrational mode > 50 Hz
- start with F=4 MN

ABAQUS Loading

Example: Modal Analysis







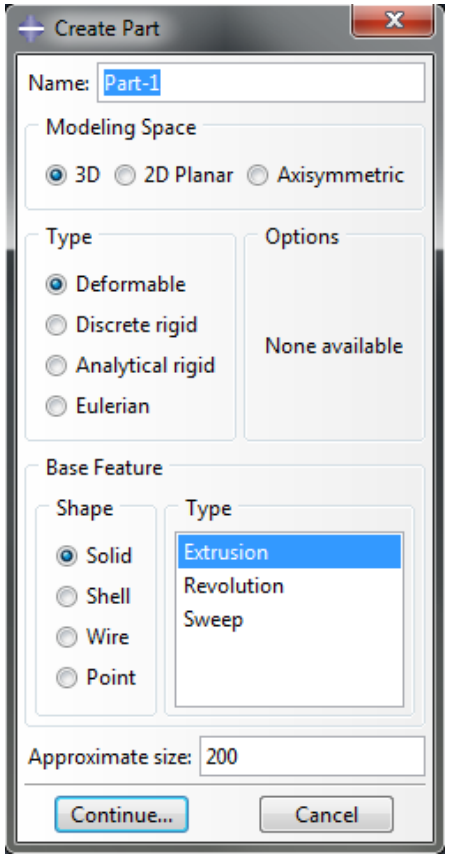
ABAQUS Loading

Geometry Features

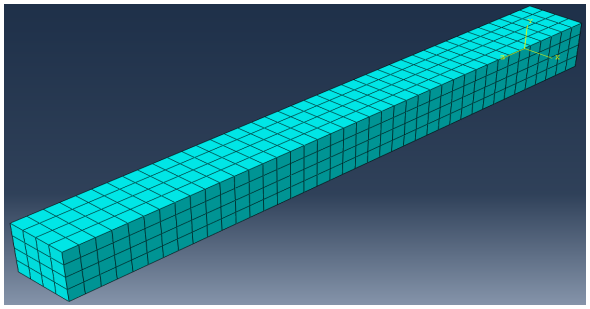
- 3D/ 2D/ Axisymmetric
- Part Type:

  Deformable/Discrete rigid/
  Analytical rigid/Eulerian
- Shape:

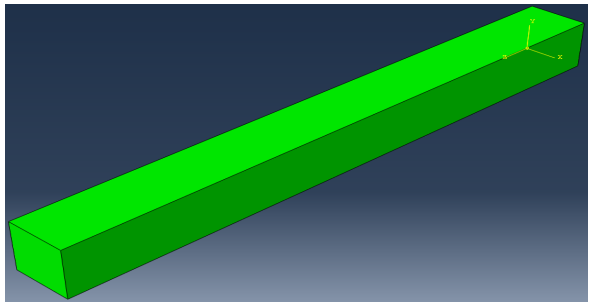
  Solid/ Shell/ Wire (Beam)/ Point



Deformable/ Discrete Rigid



Analytical Rigid



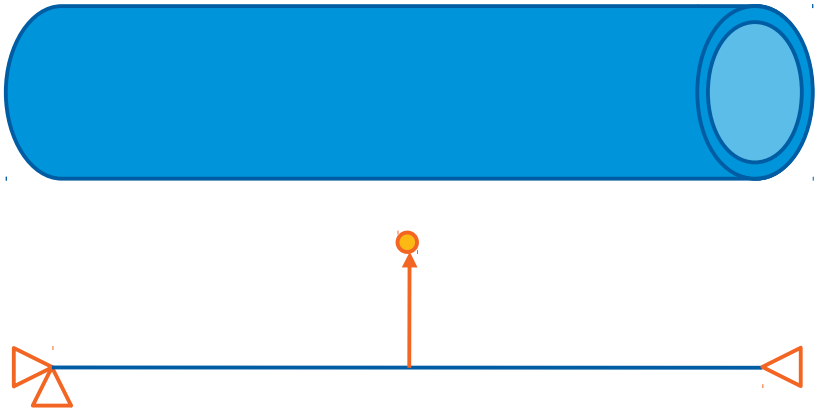
ABAQUS Loading

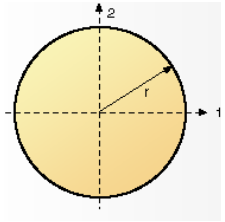
Geometry Features

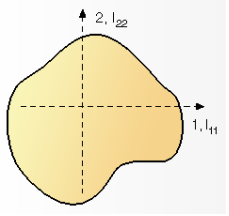
- Wire (Beam)

one dimension sig. greater, stress most imp.

section point (orientation)







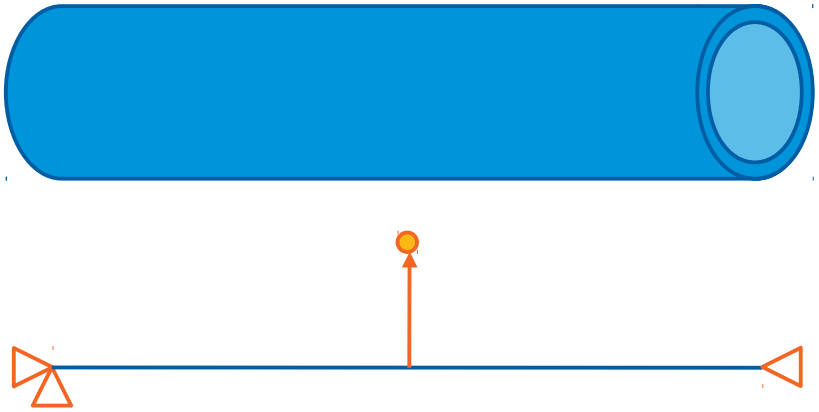
ABAQUS Loading

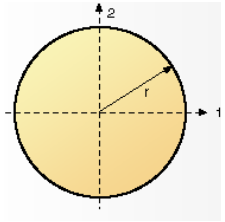
Geometry Features

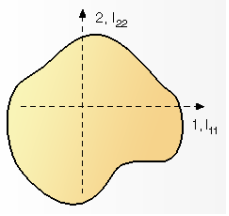
- Wire (Beam)

one dimension sig. greater, stress most imp.

section point (orientation)







ABAQUS Loading

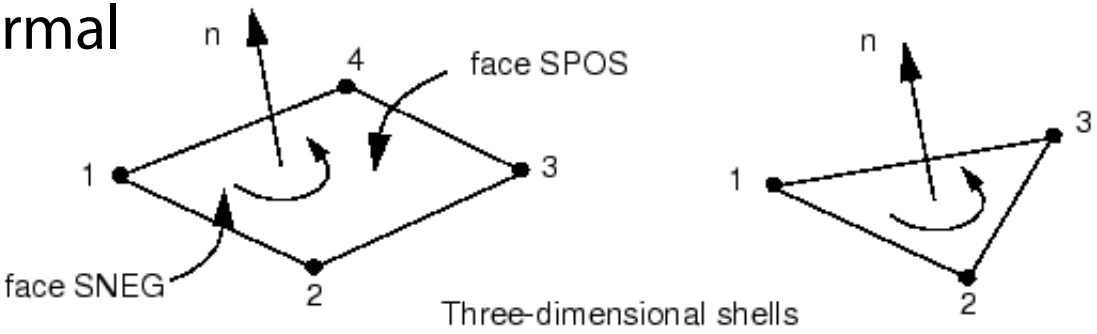
Geometry Features

- Shell

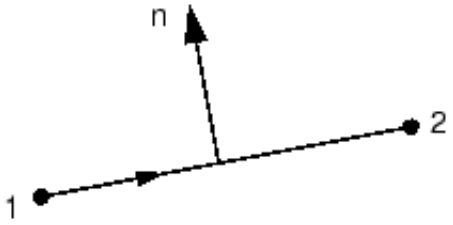
one dimension sig. smaller, stress insig.

section point (orientation)

RH rule for normal



Three-dimensional shells



Axisymmetric shells