MEC-E1005 Abaqus workshop

Rand Finland|Simulation|Software
Plaza Business Park, Loiste
Äyritie 20
01510 Vantaa
Finland

Dr.(Tech.) Kilwa Ärölä
Simulation Manager
Tel. +358 (0) 40 759 5129
E-mail kilwa.arola@rand.fi

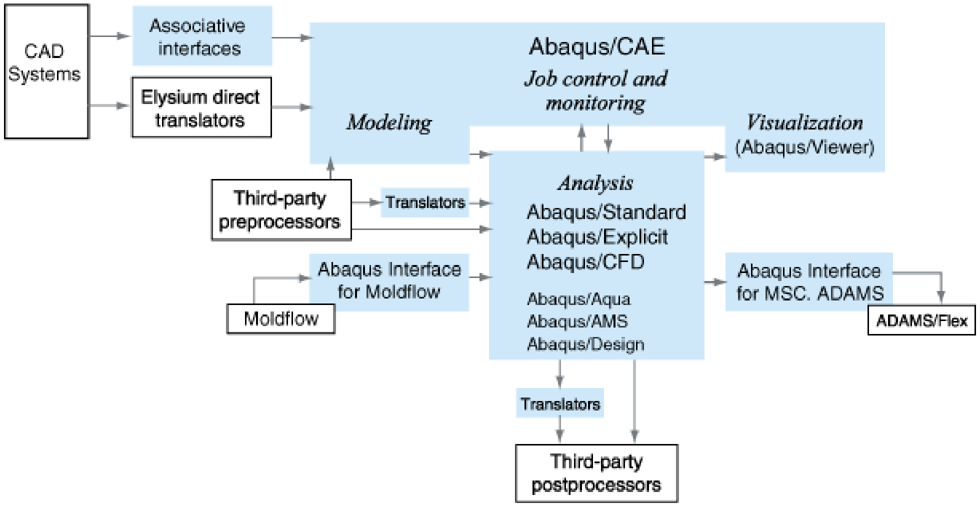
Simulation
Let **Knowledge** be your guide

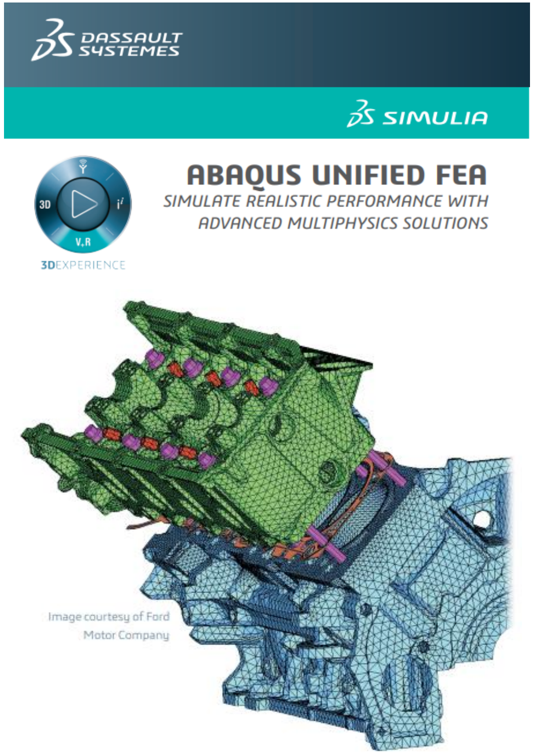
Agenda, Day 1

- Overview of Abaqus
- Basic use and concepts
- Geometry modelling
- Assigning materials and properties
- Meshing and elements
- Loads and Boundary conditions
- Submitting analyses
- Viewing results
- Static analysis Workshops
- Vibration mode analysis Workshop

What is Abaqus?

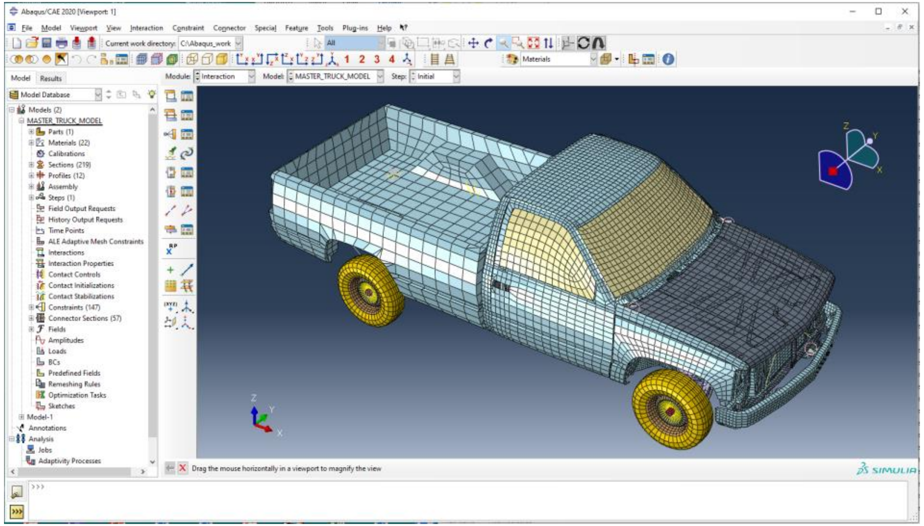
- Suite of finite element analysis modules
- Graphical user interface **Abaqus/CAE**
- Solvers **Abaqus/Standard** & **Abaqus/Explicit**





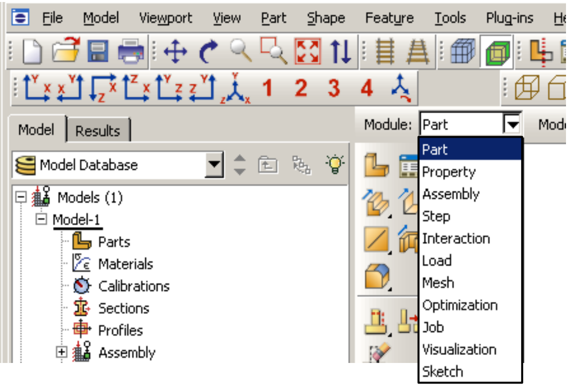
Overview of Abaqus/CAE

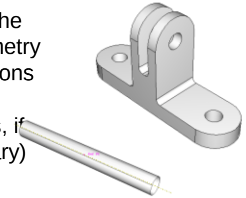
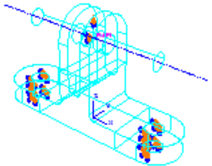
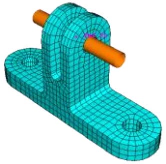
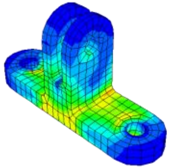
- Graphical user interface to create, edit, monitor, diagnose, and visualize Abaqus analyses
- Supports familiar interactive computer-aided engineering concepts as
  - feature-based, parametric modelling
  - interactive and scripted operation
  - GUI customization
- Provides the most complete interface with the Abaqus solver programs available.
- Uses neutral database files that are machine independent (Windows ⇔ Linux)
- Powerful extensibility & customization
  - Python scripting language
  - Automate repetitive tasks with macros
  - Build customized GUIs



Overview of Abaqus/CAE

- Functionality is presented in modules.
- Each module contains a logical subset of the overall functionality.
- Once you understand the presentation of one module, you can easily understand the presentation of the other modules.



Part	Property	Assembly
• Create the part geometry (and regions for sections, if necessary) 	• Define materials • Define additional part regions • Define and assign sections to parts or regions	• Position parts for initial configuration. 
Step	**Interaction**	**Load**
• Define analysis steps and output requests	• Define contact and other interactions on regions or named sets, and assign them to steps in the analysis history	• Apply loads, BCs, and ICs to regions or named sets; and assign them to steps in the analysis history 
Mesh	**Job**	**Visualization**
• Split assembly into meshable regions and mesh 	• Submit, manage, and monitor analysis jobs	• Examine results 

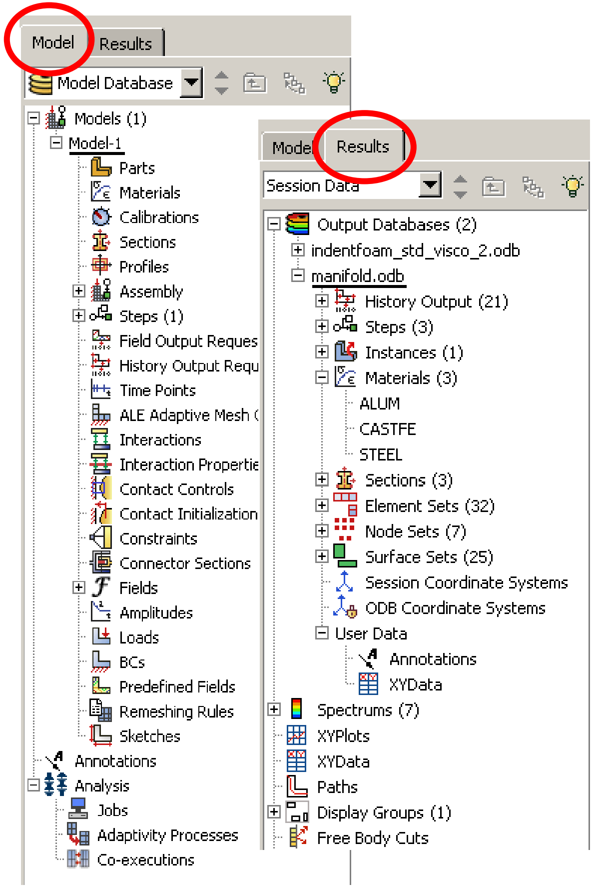
www.rand.fi | Rand Simulation Oy

Simulation

Let **Knowledge** be your guide

Overview of Abaqus/CAE

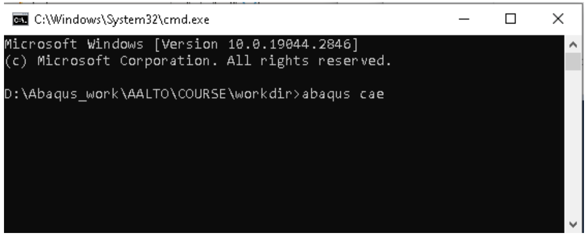
- Model Tree and the Results Tree
  - The Model Tree provides you with a graphical overview of your model and the objects that it contains.
  - The Results Tree is used to display analysis results from output databases as well as session-specific data such as *X*–*Y* plots.
  - Both trees provide shortcuts to much of the functionality of the main menu bar, the module toolboxes, and the various managers.



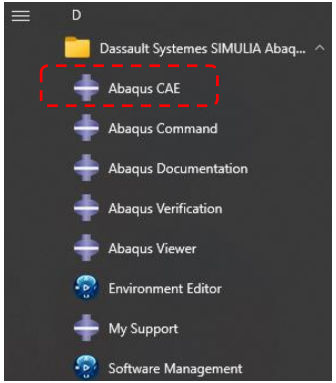
Starting Abaqus/CAE

1. Command line
   - **`abaqus cae`**
   - Opens Abaqus/CAE in current directory
2. Windows Start menu
   - Opens Abaqus/CAE in startup directory set during installation
3. Double-click **`.cae`** or **`.odb`** file in Windows folder
   - Opens Abaqus/CAE in current directory

Option 1

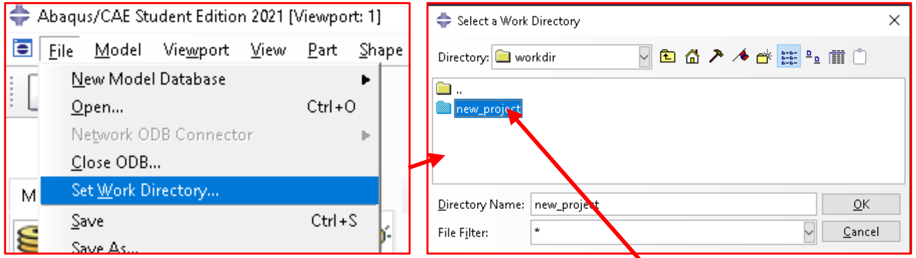


Option 2



Suggested workflow when starting new project

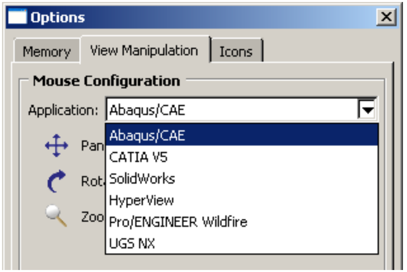
- Start Abaqus/CAE from Start menu
- Create and/or select a new directory for the project
- Now all analysis and results files are created in this directory, and things stay organized
  - Otherwise, the default work directory given during installation is used for all files



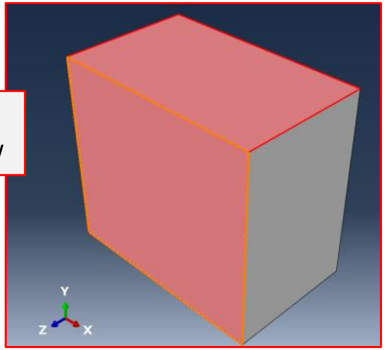
Select directory (Do not open/go into directory)

Basic use of Abaqus/CAE

- View manipulation
  - Toolbar to control view (pan, zoom, rotate, etc.).
  - Alternatively, can use a combination of keyboard and mouse actions:
    - Rotate: [Ctrl]+[Alt]+MB1.
    - Pan: [Ctrl]+[Alt]+MB2.
    - Zoom: [Ctrl]+[Alt]+MB3.
  - You can reconfigure these combinations to mimic the view manipulation interfaces used by other common CAD applications
- Selecting entities from main window
  - Add to selection: [Shift] + MB1
  - Remove from selection: [Ctrl] + MB1



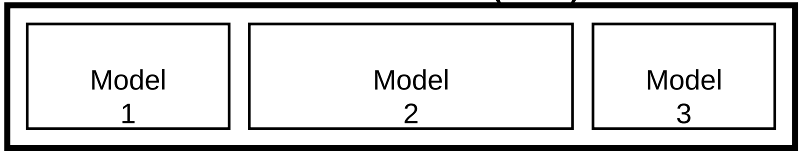
Example: Selecting faces from model in Main window

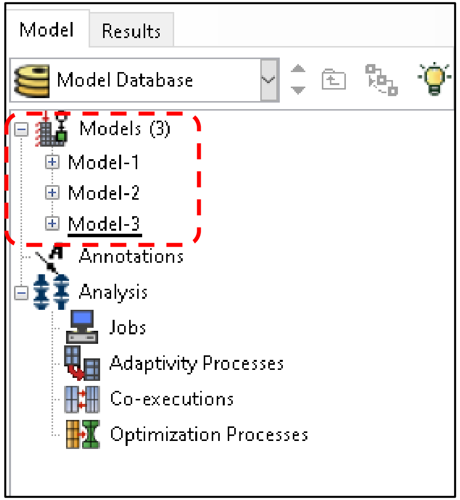


Overview of Abaqus/CAE

- What is a model database file (extension `.cae`)?
  - Contains all the information for any number of models.
  - Typically contains one model or several related models.
  - Only one model database can be opened in Abaqus/CAE at a time.

Model database (`.cae`)

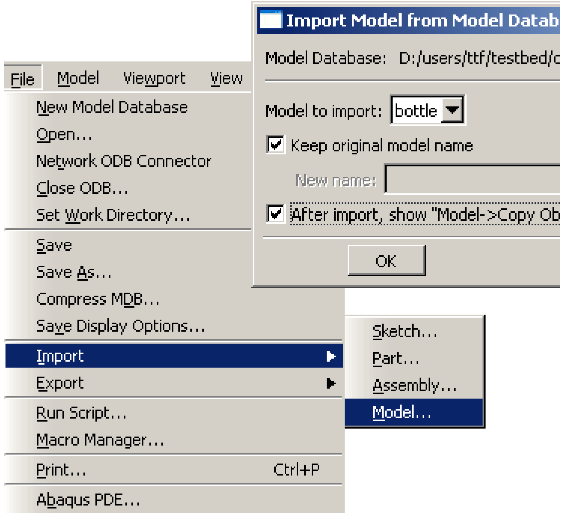


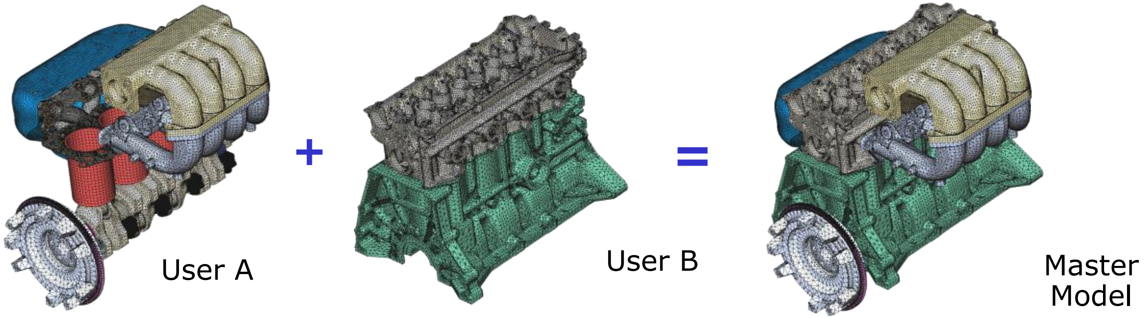


- What is a model?
  - Contains all the necessary information for an analysis.
  - Contains any number of parts and their associated properties.
  - Is independent of other models in the model database.
    - Objects such as parts and materials can be copied between different models in the same database.
  - Contains a single assembly of part instances, including the associated contact interactions, loads and boundary conditions, mesh, and analysis history.

Basic use of Abaqus/CAE

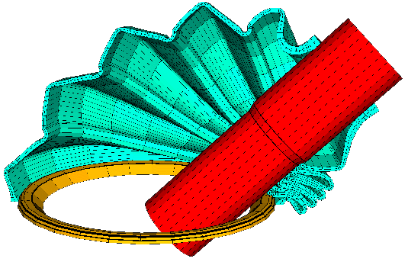
- Models can be imported into one database from another
- Model data from the imported database is copied into the current database.
  - E.g., parts, sections, assemblies, materials, loads, BCs, etc.
- Analysis job definitions and custom data are not copied



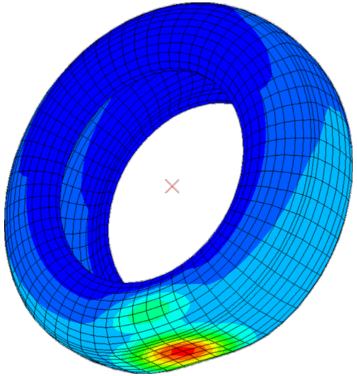


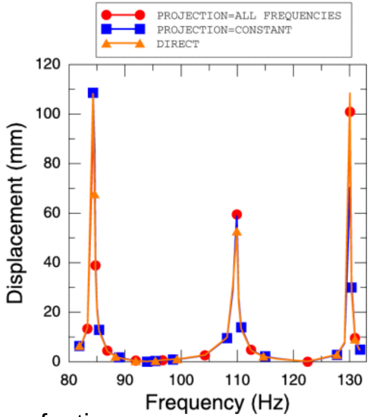
Overview of Abaqus/Standard and Abaqus/Explicit

- Abaqus/Standard analysis types
  - Static stress/displacement analysis:
    - Linear and nonlinear analysis
    - Rate-dependent or rate-independent response
    - Eigenvalue buckling load prediction
  - Linear dynamics:
    - Natural frequency extraction
    - Modal superposition
    - Harmonic loading
    - Response spectrum analysis
    - Random loading
  - Linear/Nonlinear dynamics:
    - Transient dynamics, Implicit time integration



Articulation of an automotive boot seal

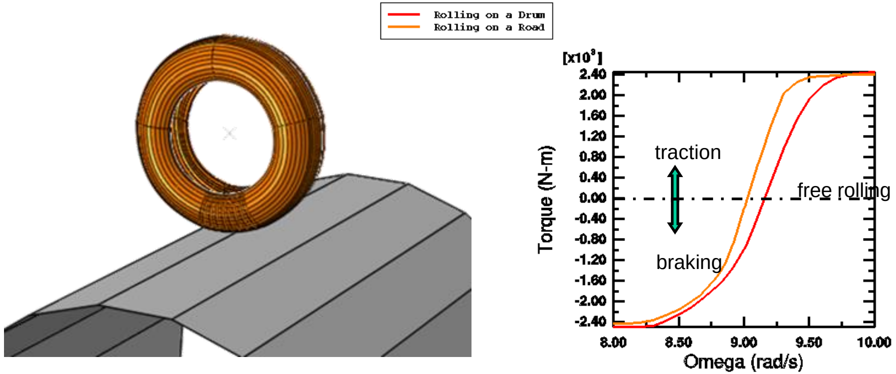




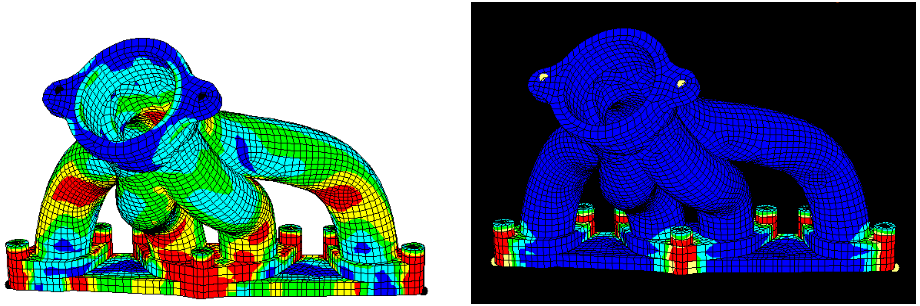
Harmonic excitation of a tire

Overview of Abaqus/Standard and Abaqus/Explicit

- Other analysis types available in Abaqus/Standard:
  - Heat transfer
  - Acoustics
  - Mass diffusion
  - Steady-state transport
- Multiphysics with Abaqus/Standard:
  - Thermal-mechanical analysis
  - Structural-acoustic analysis
  - Thermal-electrical (Joule heating) analysis
  - Linear piezoelectric analysis

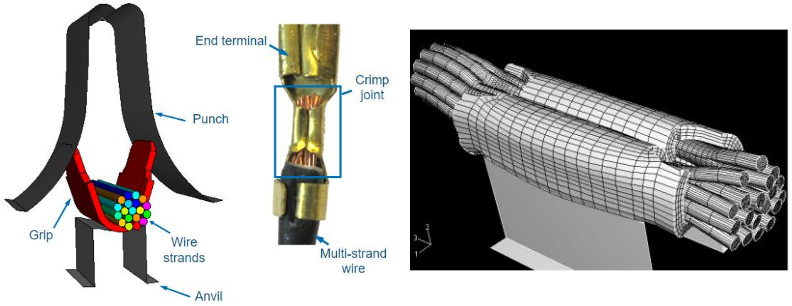


Steady-state transport: rolling of a tire on a drum

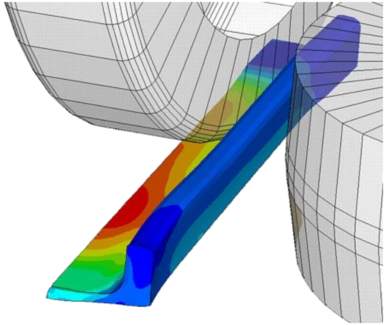

Thermal stresses in an exhaust manifold

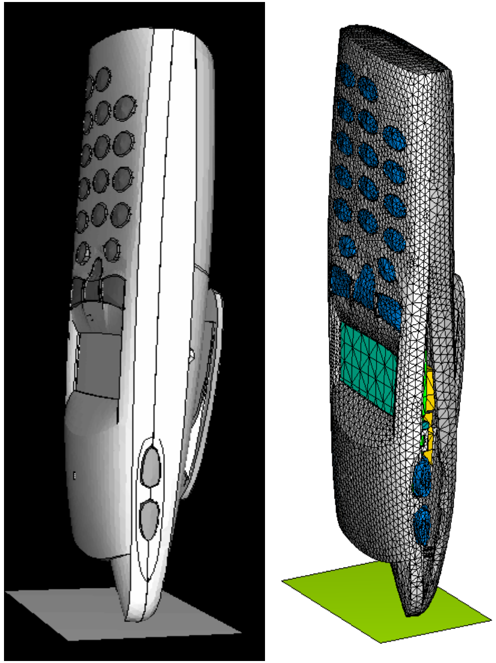
Overview of Abaqus/Standard and Abaqus/Explicit

- Abaqus/Explicit
  - Simulates high speed dynamic events such as drop tests.
    - Explicit algorithm for updating the mechanical response.
  - Also, a powerful tool for highly nonlinear quasi-static analyses
    - Material plasticity & damage, multiple compilated contacts, large deformations,...
    - Annealing is available for multistep forming simulations



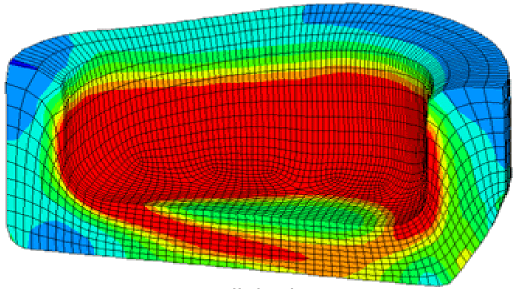
Crimping of terminal


Rolling of a symmetric I-section

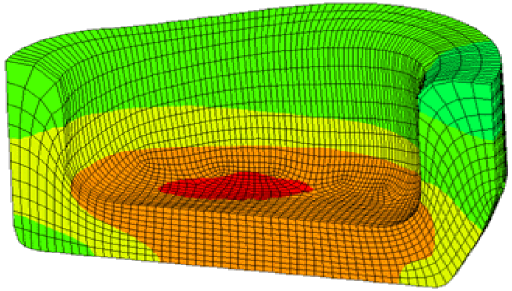

Drop test of a cell phone

Overview of Abaqus/Standard and Abaqus/Explicit

- Multiphysics with Abaqus/Explicit
  - Thermal-mechanical analysis
    - Fully coupled: Explicit algorithms for both the mechanical and thermal responses
    - Can include adiabatic heating effects
  - Structural-acoustic analysis
  - Fluid-structure interaction



adiabatic

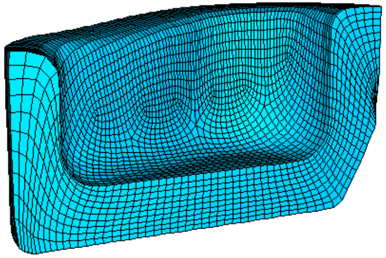


fully coupled temperature-displacement

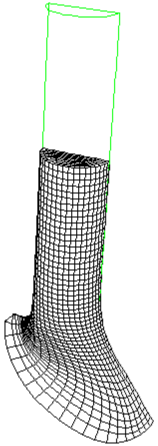
Two-stage forging, using ALE—contours of temperature

Overview of Abaqus/Standard and Abaqus/Explicit

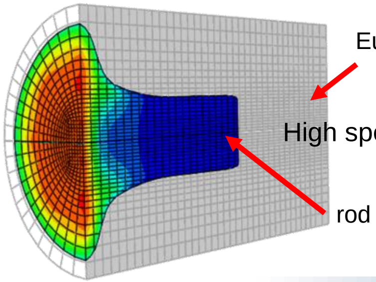
- Special features of Abaqus/Explicit: ALE
  - Adaptive meshing using ALE techniques allows the robust solution of highly nonlinear problems.
  - Mesh adaptivity is based on solution variables as well as minimum element distortion.
  - Elements concentrate in areas where they are needed.
  - Adaptation is based on boundary curvature.
- Special features of Abaqus/Explicit: Coupled Eulerian-Lagrangian (CEL)
  - Define a domain in which material can flow for an Eulerian analysis
    - Flow problems
    - Structural problems with extreme deformation



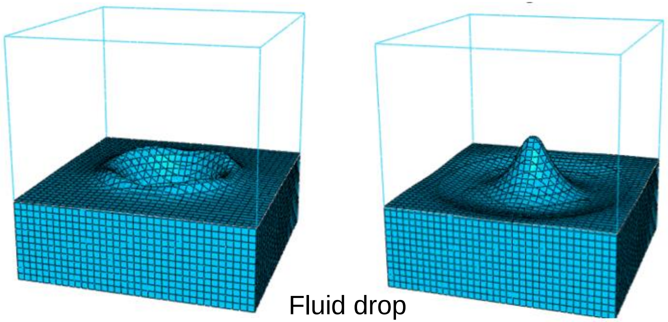
Bulk metal forming



High speed impact



High speed impact of copper rod



Fluid drop

Abaqus Conventions

- Abaqus uses no inherent set of units.
  - User inputs numbers and Abaqus works with these
- It is the user's responsibility to use consistent units. Examples:
  - SI: kg, m, s ⇒ Force: N, Stress: Pa
  - SI (mm): ton (10^3 kg), mm, s ⇒ Force: N, Stress: MPa
- Other unit systems and units shown in table below

Most common, and generally recommended

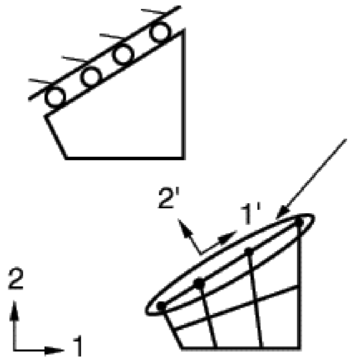
Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2 /in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	tonne/mm^3	slug/ft^3	lbf s^2 /in^4

Common systems of consistent units

Example: Properties of structural steel in SI (mm):
Density: 7.85e-9 [ton/mm^3]
Young's modulus: 206 000 [MPa]

Abaqus Conventions

- For **boundary conditions** and **point loads**, the default coordinate system is the rectangular Cartesian system.
  - Alternative local rectangular, cylindrical, and spherical systems can be defined.
  - These local directions **do not** rotate with the material in large-displacement analyses.

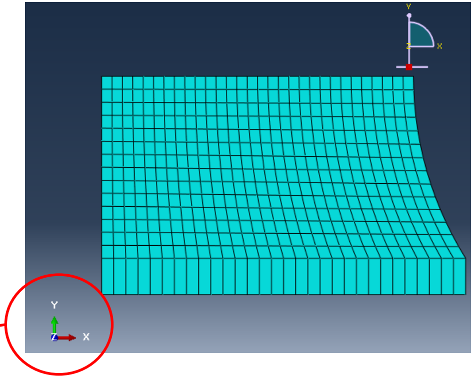


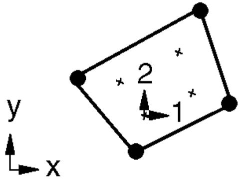
local rectangular coordinate system with YSYMM boundary conditions

Boundary conditions on a skew edge

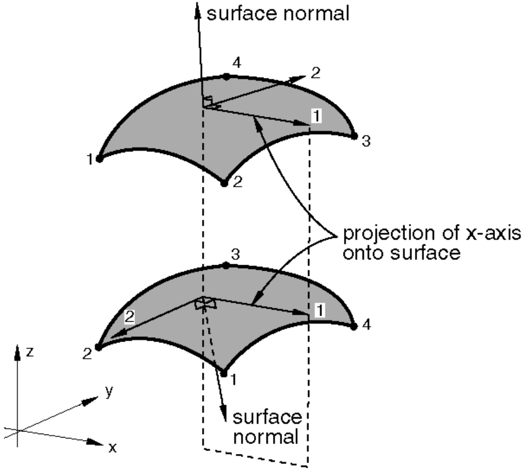
Abaqus Conventions

- For **material directions** (i.e., directions associated with an element's material or integration points) the default coordinate system depends on the element type:
  - Solid elements use global rectangular Cartesian system.
  - Shell and membrane elements use a projection of the global Cartesian system onto the surface.





Default material directions for solid elements

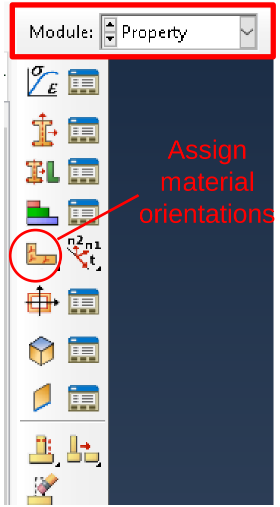


Default material directions for shell and membrane elements

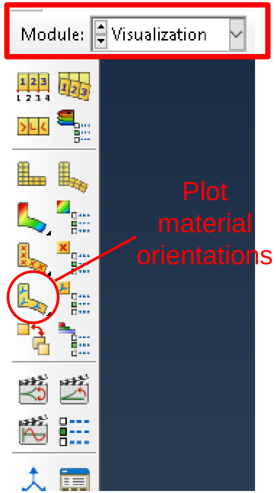
Global coordinate system

Abaqus Conventions

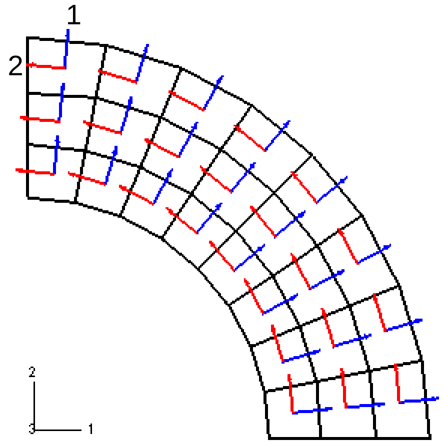
- Alternative rectangular, cylindrical, and spherical coordinate systems may be defined.
  - Affects input: anisotropic material directions.
  - Affects output: stress/strain output directions.
  - Local material directions **rotate** with the material in large-displacement analyses.



Pre processing



Post processing



Abaqus Conventions

- Degrees of freedom
  - Primary solution variables at the nodes.
  - Available nodal degrees of freedom depend on the element type.
  - Each degree of freedom is labeled with a number: 1=*x*-displacement, 2=*y*-displacement, 11=temperature, etc.

1 Translation in the 1-direction.

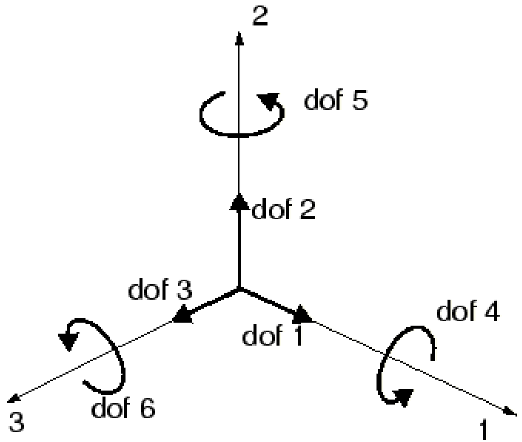
2 Translation in the 2-direction.

3 Translation in the 3-direction.

4 Rotation about the 1-direction.

5 Rotation about the 2-direction.

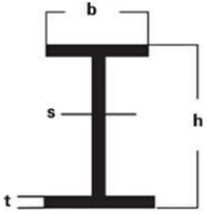
6 Rotation about the 3-direction.



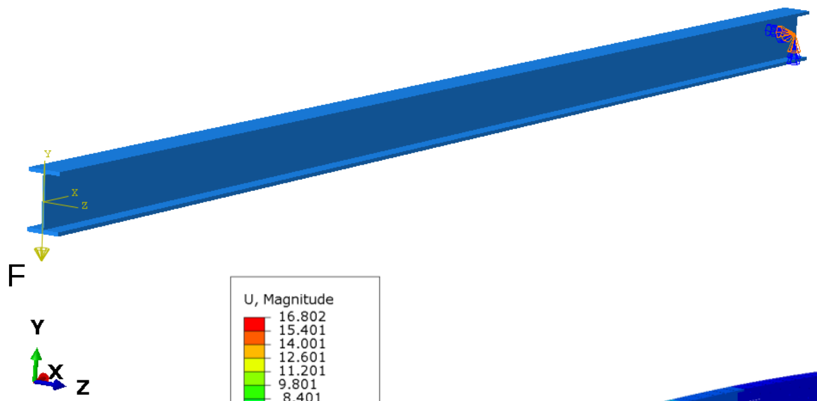
Demonstration of Abaqus/CAE basic functionality

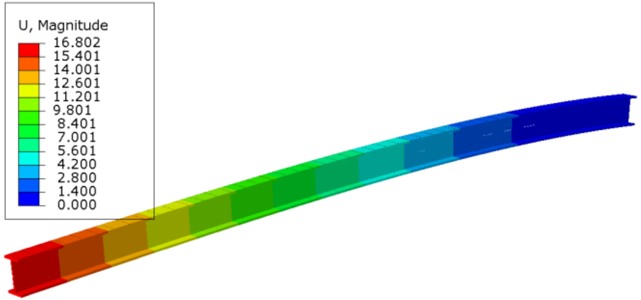
Static analysis of I-profile cantilever beam

- IPE80
  - h = 80
  - b = 46
  - t = 5.2
  - s = 3.8
- E = 206 GPa, ν = 0.3
- F = 1000 N
- L = 2000 mm
- Use beam elements



This end is fixed





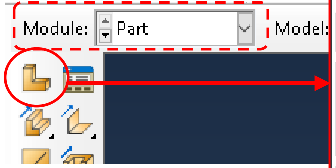
Step 1: Create a part

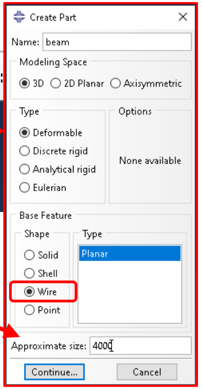
For **beam** elements

- Modelled geometry: **Line/Wire**
- Data given by section properties:
  - Material
  - Profile shape
  - Profile Orientation

Part module

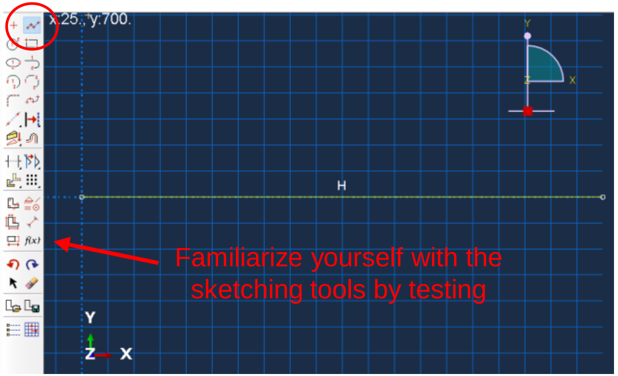
Create part





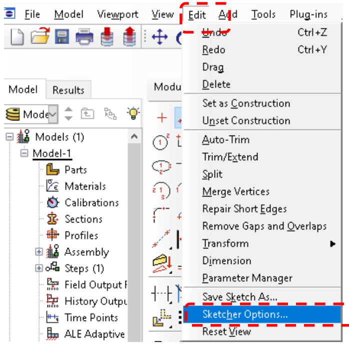
Give approximate size of drawing canvas

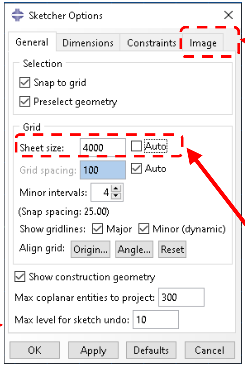
Sketch line



Familiarize yourself with the sketching tools by testing

Sketcher options



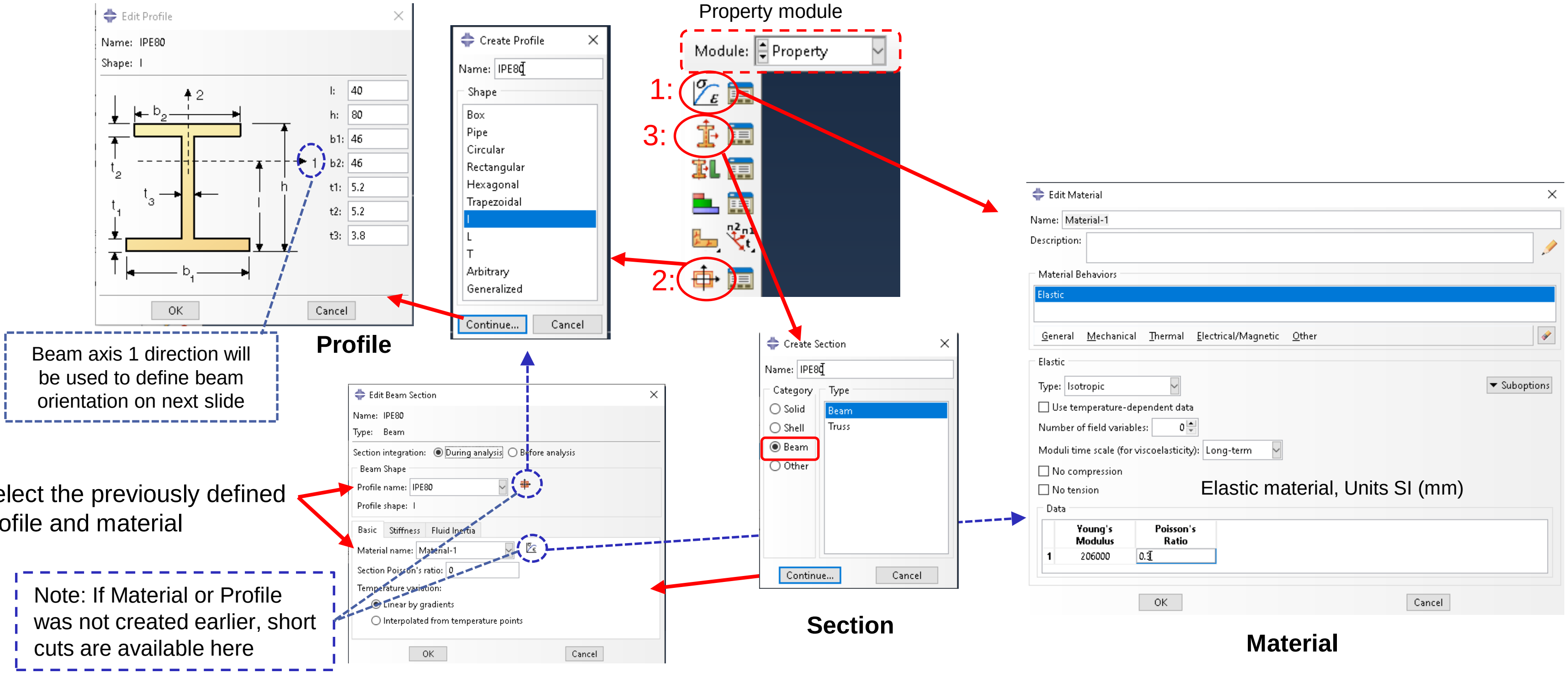


An image can be imported, scaled, and set on the background of the canvas. This can help in modelling complex shapes: take a photo, set is as background image, and draw lines according to the photo

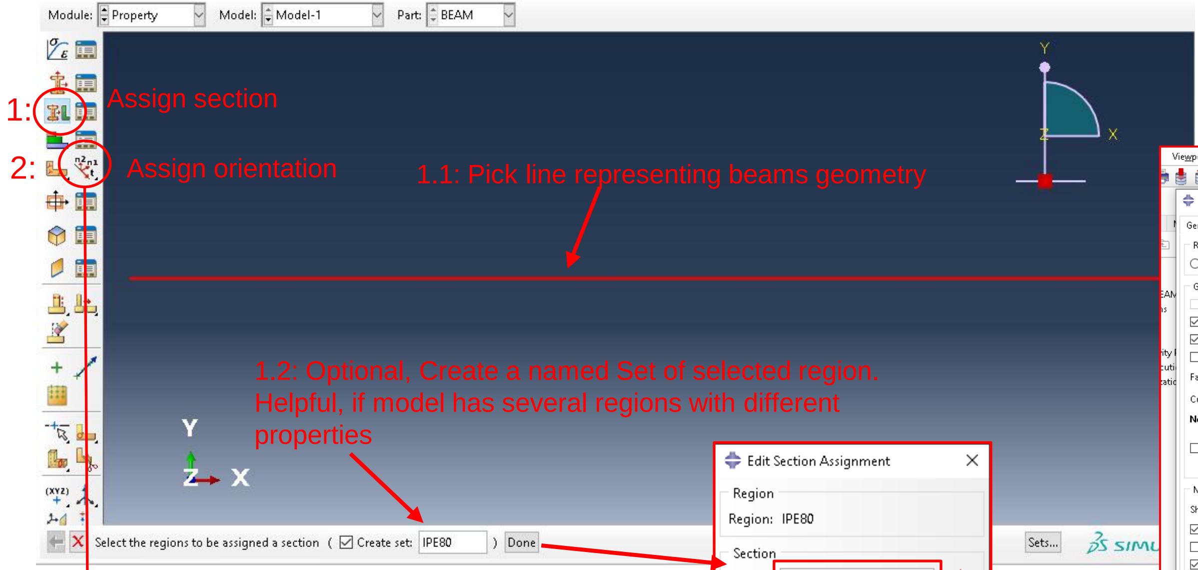
Modify size of drawing canvas

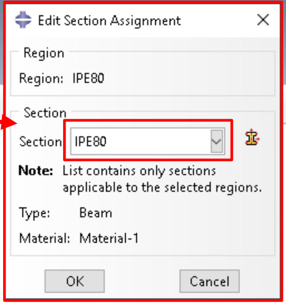
Simulation
Let **Knowledge** be your guide

Step 2: Create material, profile, and beam section



Step 3: Assign beam orientation

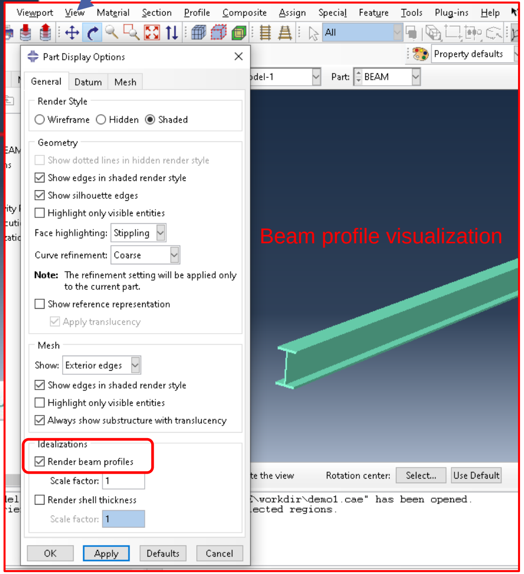




1. Select region (the line representing the beam)
2. Abaqus/CAE shown arrows along beam tangent. Give approximate direction for beam cross section axis 1 (see profile definition on previous slide for axes 1 and 2 directions)

After section and orientation assignments are done, the beam profile can be visualized:

Main menu: View > Part Display Options

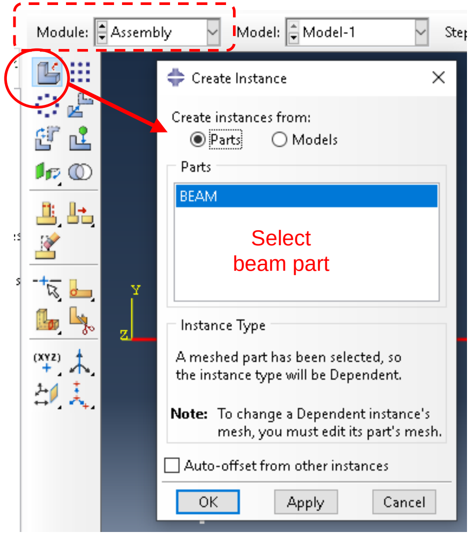


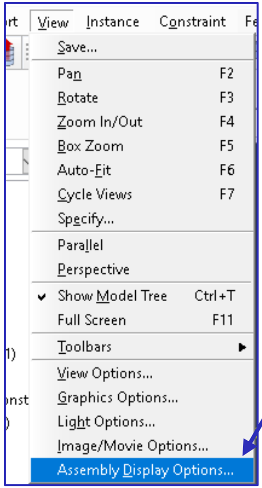
Step 4: Create the Assembly

- An "Instance" of a part is created in the assembly
- Multiple instances of the same part can be created
  - Part instances can be moved and rotated as needed
  - If changes are made to the part; modified dimensions, materials, section properties and assignments,... all instances of the part in the assembly are updated
- **IMPORTANT**: The assembly is what is used in the analysis
  - Parts that are not instanced are not considered in the analysis

Assembly module

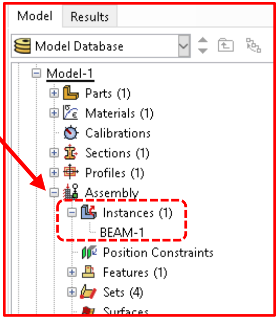
Create Instance





Tip 1: After creating instances, check the number of instances under *Assembly > Instances* in the model tree. Make sure you have not created multiple instances of the same part by accident

Tip 2: Assembly level has it's own display settings. Beam profile rendering in the Assembly level modules can be switched on from Main menu > View > Assembly Display Options

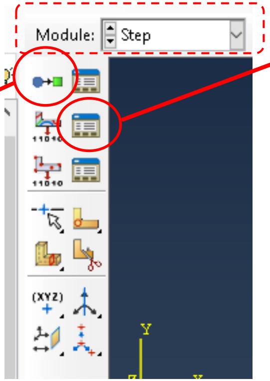


Step 5: Create analysis step

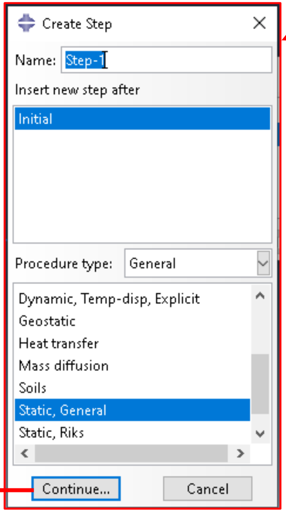
Analysis step defines what is calculated

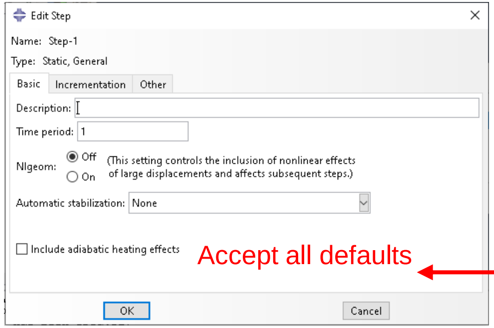
Modify the default output request to add output of beam section forces SF. This is needed to visualize the stress on the beam cross section.

Step module

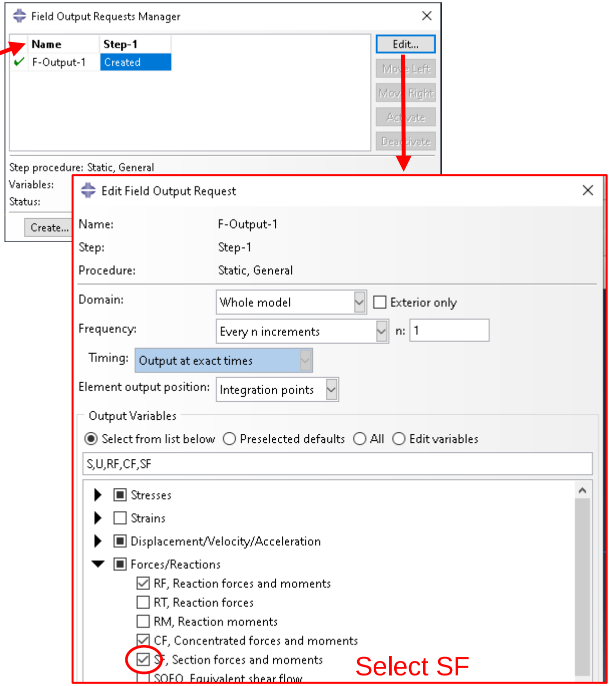


Create Step





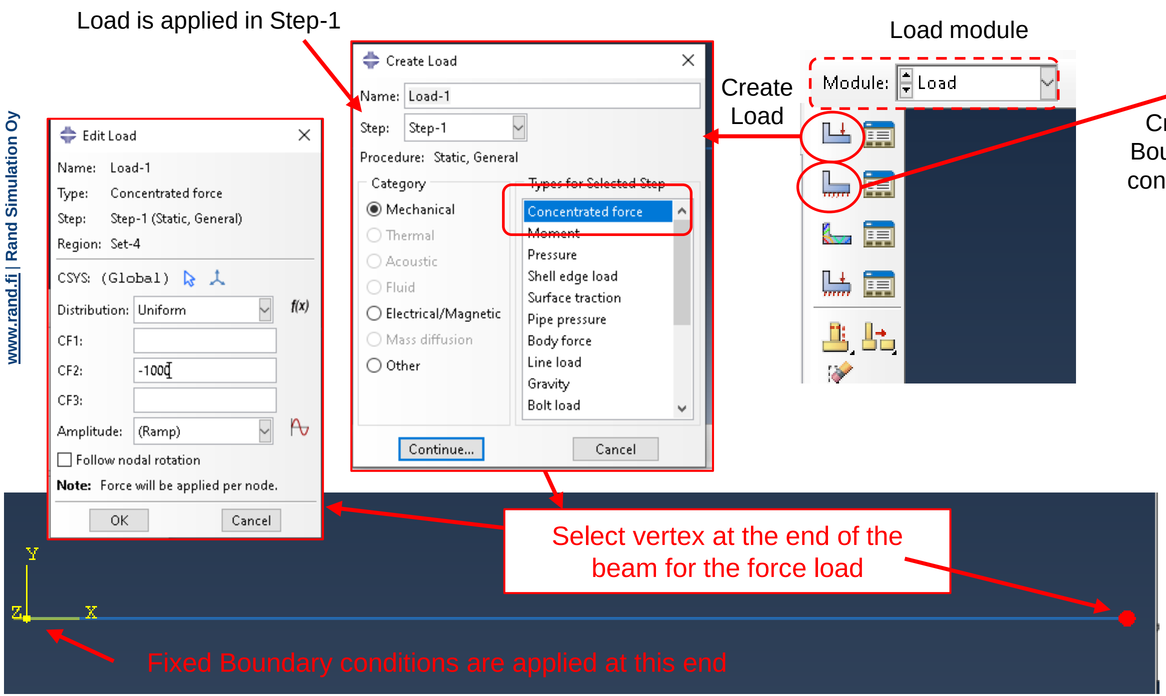
Accept all defaults

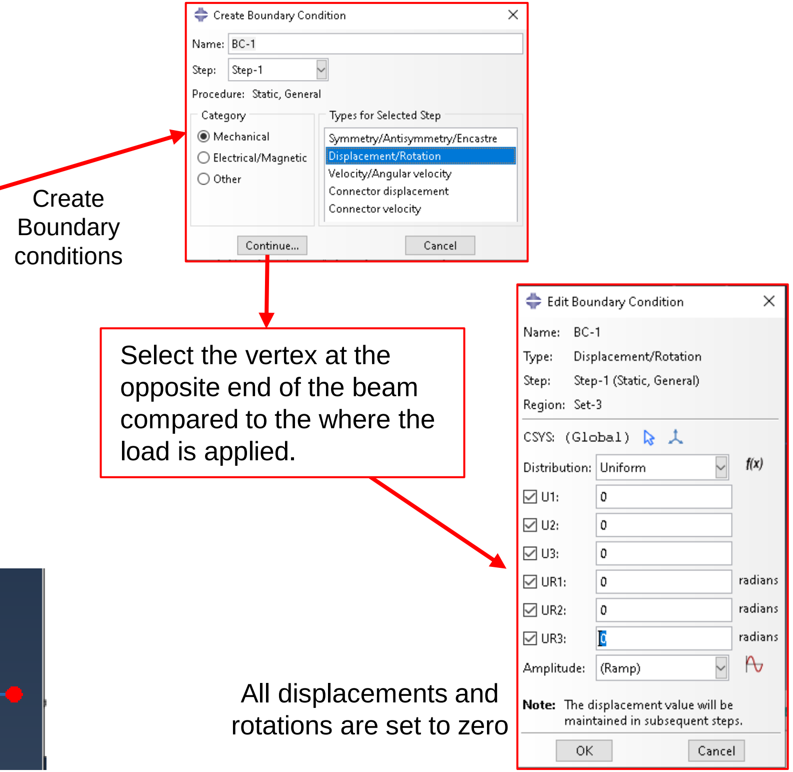


Select SF

Step 6: Define loads and boundary conditions

The analysis step defines what is calculated

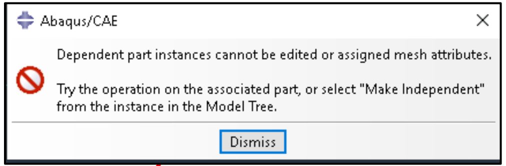




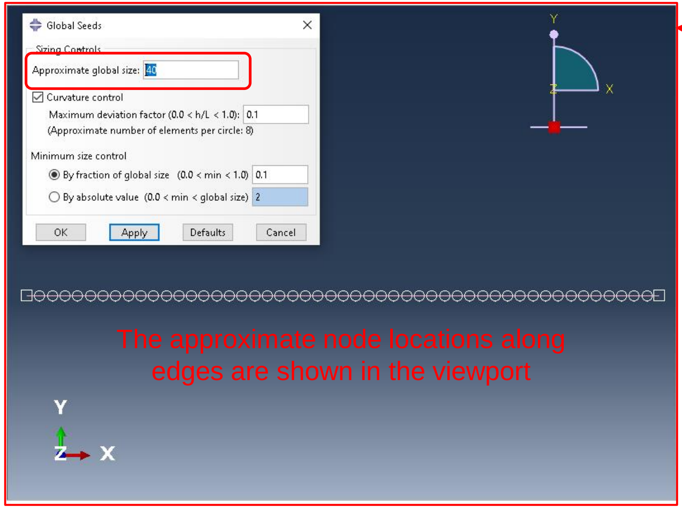
Step 7: Mesh the part

- The geometry is divided into finite elements for the calculation

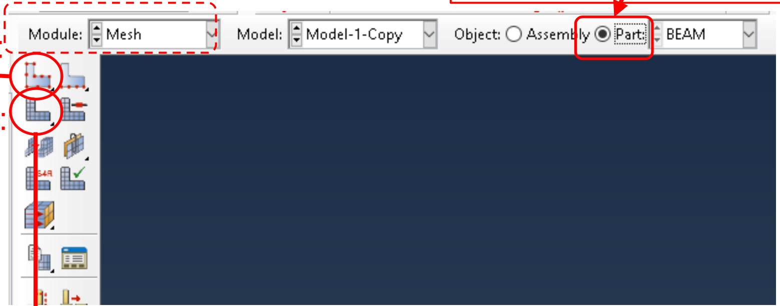
Tip 1: When you see this error, switch to the Part level in the mesh module and try again.



Assign global seed size. This is the approximate distance between nodes of the mesh.



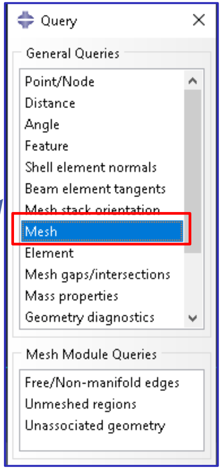
Mesh module



Mesh the model

Tip 2: Use *Query Information* to get the details of the mesh.
Info is printed in the message are under the main window.
Remember: Student Edition has the limit of 1000 nodes

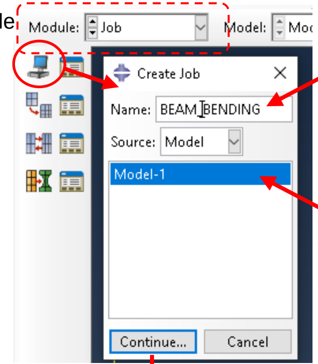
```
Total number of nodes: 51
Total number of elements: 50
     50 linear line elements of type B31
```



Step 8: Create a Job and submit the analysis

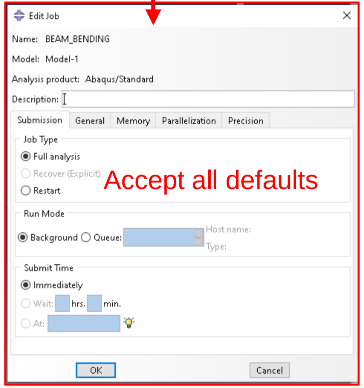
Job module

Create Job

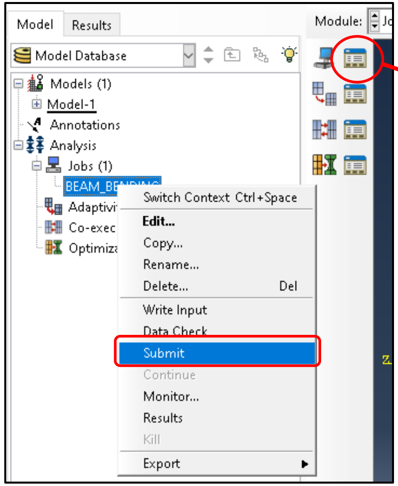


Give a name to the job. All analysis and output files will have this name.

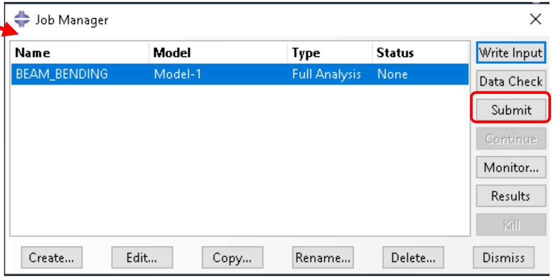
If you have multiple models in your `.cae` database, select the one you want to analyse.



Once the Job has been created, it can be submitted for analysis.



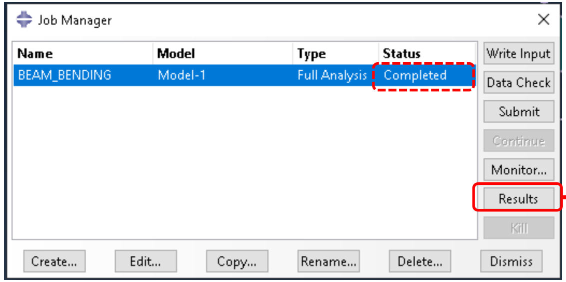
Option 1: Right click the Job in the Model tree and select *Submit*

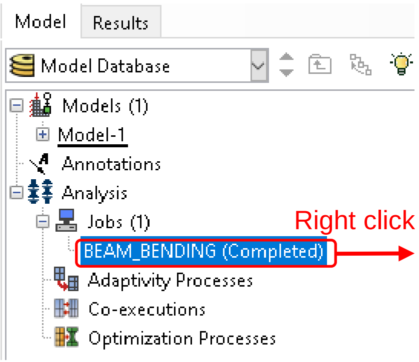


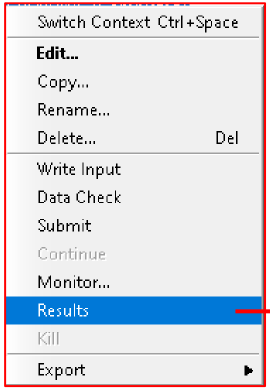
Option 2: Open the Job manager and select *Submit*

Step 9: Visualize the results

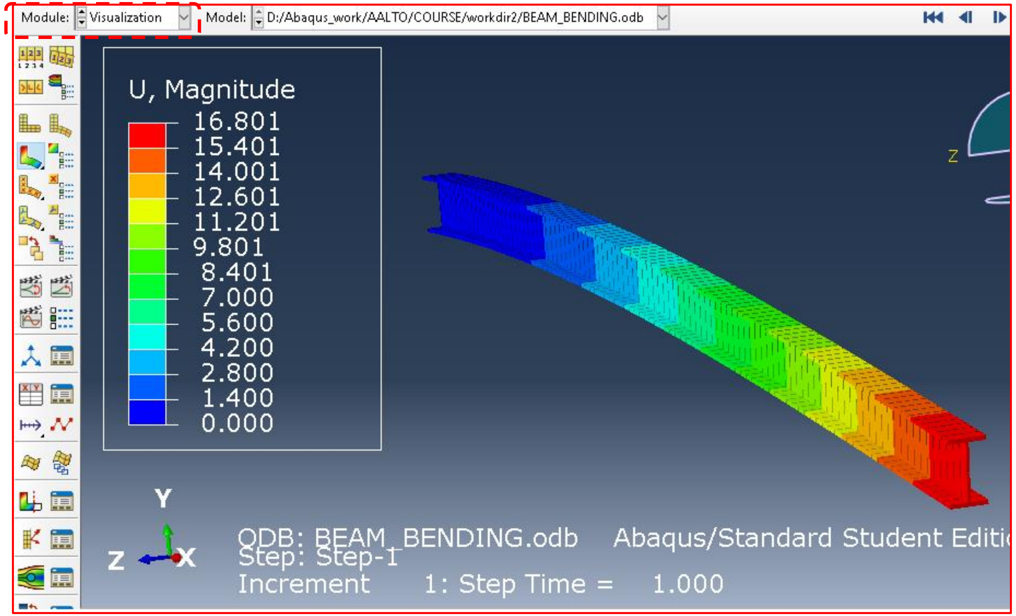
Check that the job has completed from the Model tree or Job manager







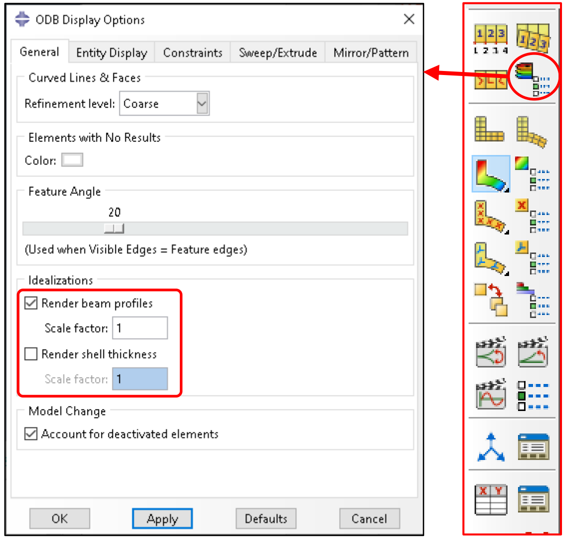
Visualization module



Simulation
Let **Knowledge** be your guide

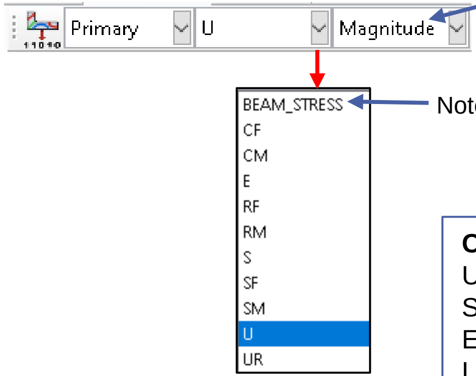
Tips for result visualization

Rendering beam profiles and shell thickness



Selecting variable to show in contour plot

Vector or Tensor component or scalar (magnitude, equivalent stress,.. etc.) is selected here.

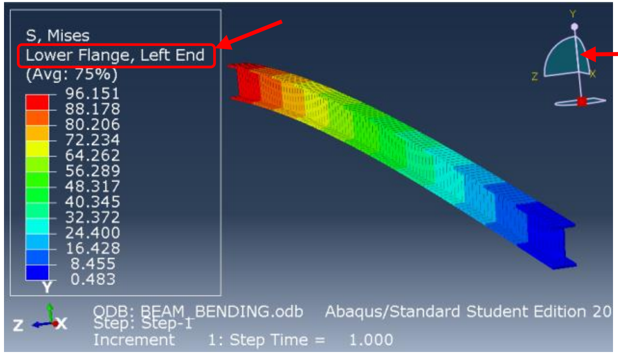


Note: Output variable SF must be included in outputs to get BEAM_STRESS

Common output variables
U/UR: Displacement (translations/rotations)
S: Stress
E: Linear strain (for linear problems)
LE: Logarithmic strain
RF/RM Reaction forces (force/moment)
CF/CM: Concentrated loads (force/moment)
SF/SM: Beam section forces (force/moment)

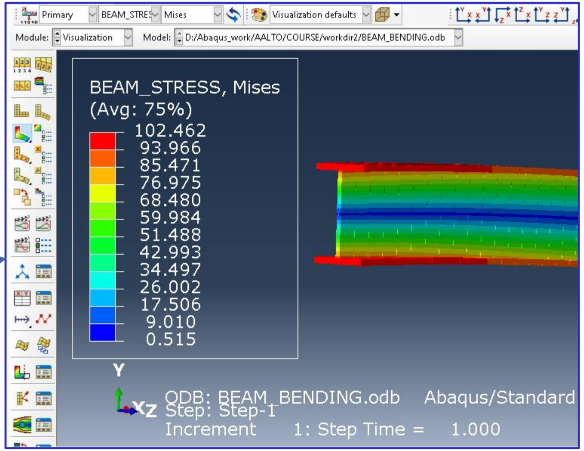
Tips for result visualization

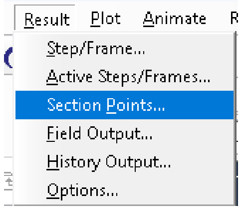
- For solid elements visualization is trivial: Result value at each point is unambiguous
- For shell elements the result value can be from various locations in the thickness of the shell
- For beam elements the result value can be from various locations of the profile cross section



Example of beam element output. Location of stress visualized by colours is given in the legend. Note that the Mises stress in each cross section is constant because the Lower flange left corner value is plotted for the whole visualized cross section.

Exception: BEAM_STRESS output plots the stress correctly across the beam cross section. However, this output variable is valid only for linear elastic material. Do not use with plasticity.





Location in the shell thickness direction or the position in beam cross section used for output is selected from Main menu > Result > Section Points

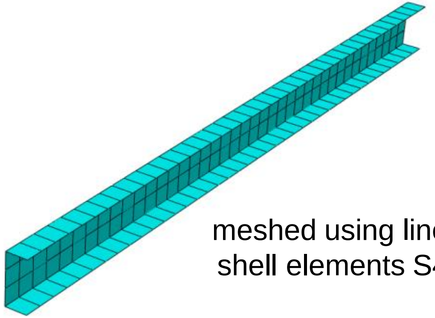
Static analysis of U-profile cantilever beam

U-profile 60x140x6

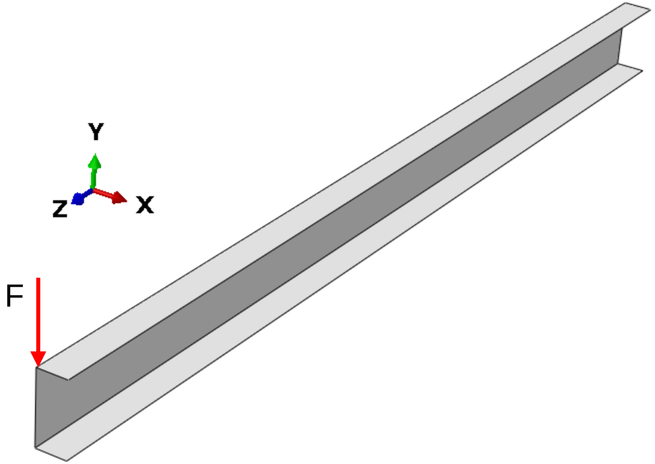
L = 2000 mm

E = 206 GPa, ν = 0.3

F = 1000 N



meshed using linear shell elements S4R

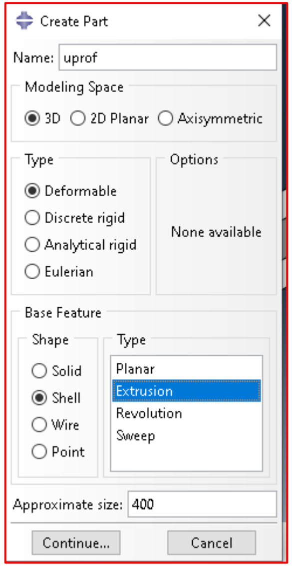


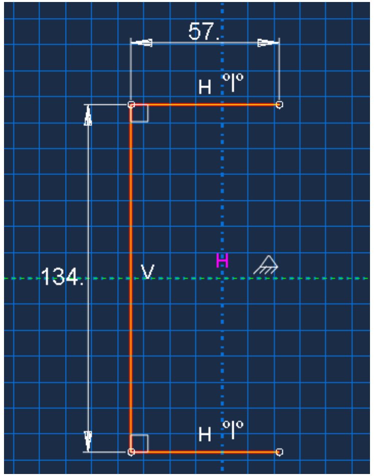
This end is fixed

Step 1: Create a shell part

- This time shell elements are used to model the beam
  - The cross section is modelled as geometry
  - Material and thickness are given as section properties

Create part with base feature *Shell, Extrusion*



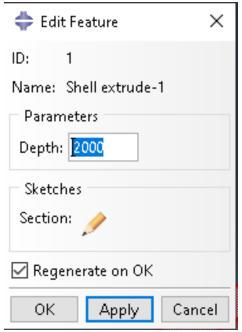


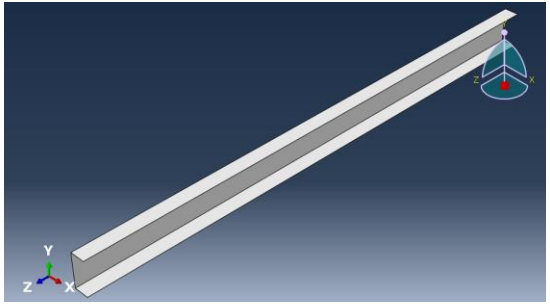
Cross section for extruded shell.

Note: Dimensions correspond to the middle surface of the profile cross section.

Outer dimensions: 60x140x6

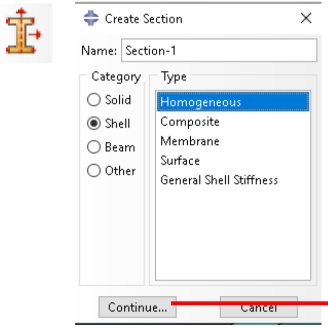
After sketching, exit the sketcher and give 2000 for the extrusion depth.

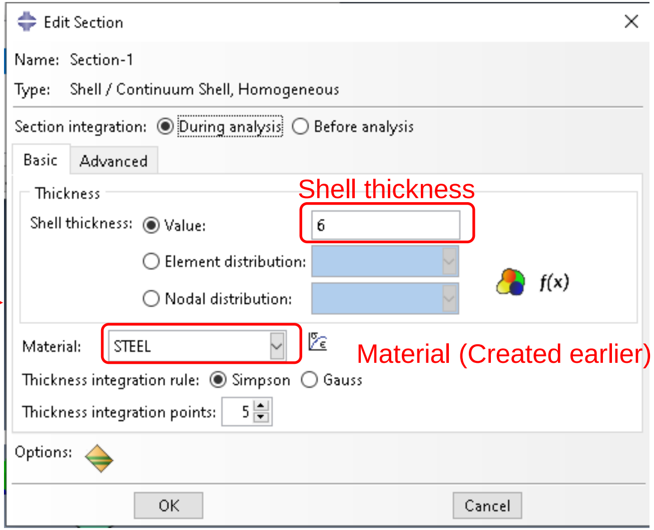


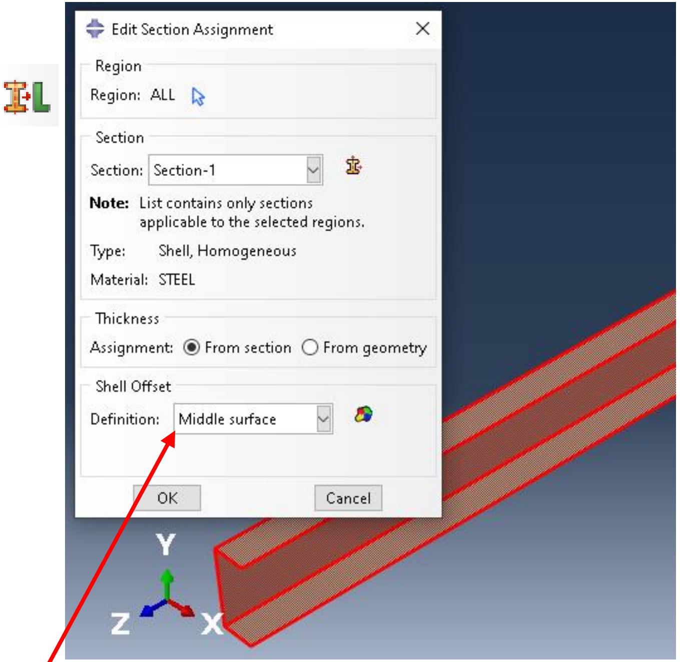


Step 2: Create and assign section

Create a shell section





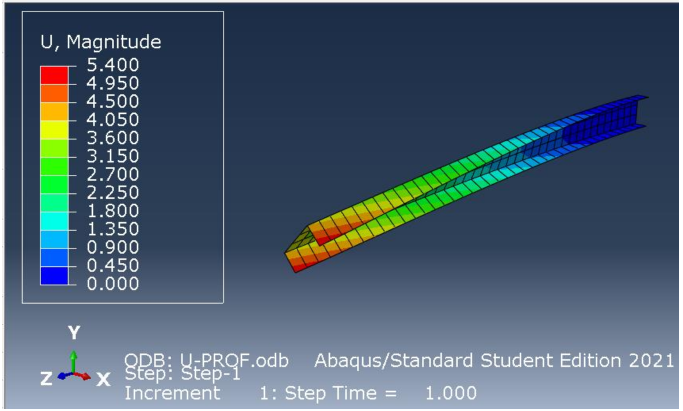


Tip: After Section assignment the shell thickness can be visualized:
Main menu > Part Display Options > Render Shell Thickness

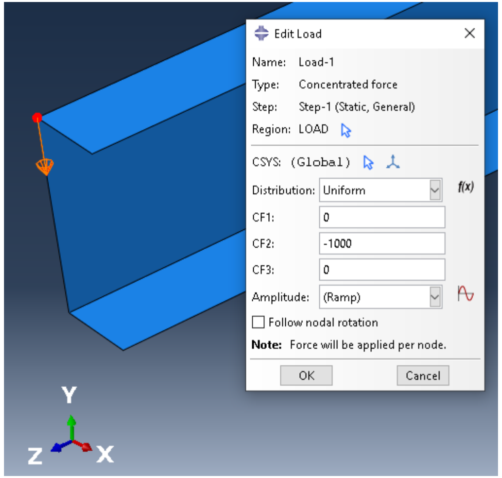
Sketch of cross section was done using the middle surface dimensions, so *Middle surface* is selected. Optionally the thickness can be offset with reference to the modelled geometrical surface

Next steps

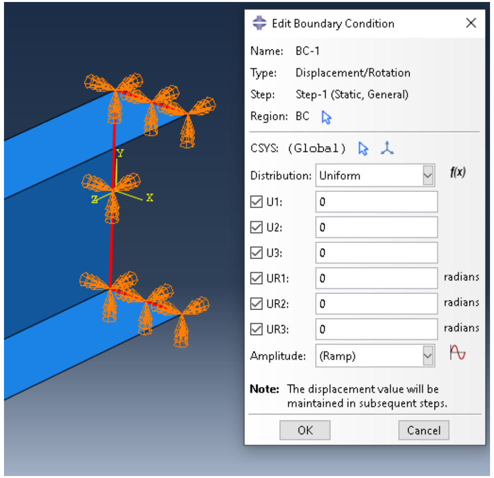
- Similarly as in the first example:
  - Create the assembly
  - Create an analysis step (Static, General)
  - Apply loads and boundary conditions
  - Mesh the part
  - Create an analysis job
  - Submit the analysis



Concentrated force at one end of the beam



Fixed boundary conditions at the other end of the beam

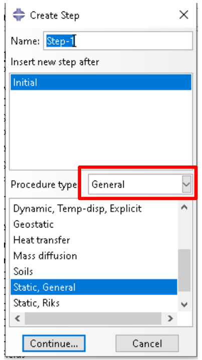


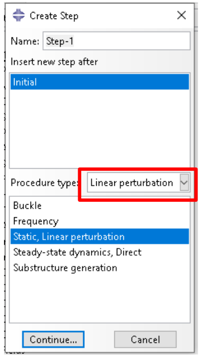
www.rand.fi | Rand Simulation Oy

Simulation
Let **Knowledge** be your guide

General and Linear perturbation steps

- Abaqus has General steps and Linear perturbation steps
- Linear perturbation = small displacements around a base state
  - Linear analysis $\Rightarrow$
    - Small displacements
    - Only elastic material response considered
    - Contact status remains the same as in base state
- Base state:
  - Initial state of model, if linear perturbation is the first step of the analysis
  - State at the end of the preceding General step where NLGEOM is on
- Linear perturbation steps do not effect the time history of a sequence of General steps
  - Duration of linear perturbation step is 1e-36





General and Linear perturbation steps

- Example: Vibration modes of tensioned string
- Sequence of steps

Step Manager

	Name	Procedure	Nlgeom	Time
✔	Initial	(Initial)	N/A	N/A
✔	TENSION-1	Static, General	ON	1
✔	Step-2	Frequency	ON	0
✔	TENSION-2	Static, General	ON	1
✔	Step-4	Frequency	ON	0

Create... Edit... Replace... Rename... Delete... Nlgeom... Dismiss

Apply tension 1 N
Calculate eigenmodes
Apply tension 2 N
Calculate eigenmodes

Step/Frame

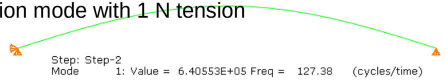
Step Name	Description
TENSION-1	
Step-2	
TENSION-2	
Step-4	

Frame

Index	Description	
0	Increment	0: Base State
1	Mode	1: Value = 6.40553E+05 Freq = 127.38 (cycles/time)
2	Mode	2: Value = 2.56251E+06 Freq = 254.77 (cycles/time)
3	Mode	3: Value = 5.76677E+06 Freq = 382.20 (cycles/time)
4	Mode	4: Value = 1.02548E+07 Freq = 509.66 (cycles/time)

Density: 7.85E-9 Ton/mm**3
E: 200 000 MPa
Length: 500 mm
Diameter: 0.1 mm
Tension: 1 N & 2 N
Element type: B23

1st vibration mode with 1 N tension



Analytical: 127.36

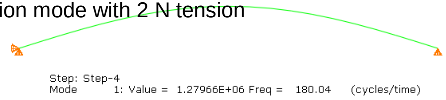
Step/Frame

Step Name	Description
TENSION-1	
Step-2	
TENSION-2	
Step-4	

Frame

Index	Description	
0	Increment	0: Base State
1	Mode	1: Value = 1.27966E+06 Freq = 180.04 (cycles/time)
2	Mode	2: Value = 5.11892E+06 Freq = 360.09 (cycles/time)
3	Mode	3: Value = 1.15187E+07 Freq = 540.16 (cycles/time)
4	Mode	4: Value = 2.04805E+07 Freq = 720.26 (cycles/time)

1st vibration mode with 2 N tension



Analytical: 180.11

Applied force

Ux & Uy fixed

Uy fixed

Agenda day 2

- Geometric nonlinearity
- Material nonlinearity
- Constraints and contacts
- Nonlinear static analysis Workshops
- Eigenmode buckling analysis Workshop
- Dynamic analysis Workshop

Linear and non-linear problems

- Properties of the linear problem
  - Load scaling
    - Calculated result U (can be displacement, stress,...) due to load F
    - If load F is increased 4x, the result U will be also 4x bigger
  - Load superposition
    - Load F_1 causes result U_1
    - Load F_2 causes result U_2
    - If loads F_1 and F_2 act at the same time, the result $U = U_1 + U_2$
  - The final state is independent of the loading order.
    - If the load F_1 is applied first and then the load F_2, the result U will be the same as in the case when first load F_2 and then load F_1 would be applied.
- **If the problem in non-linear the above-mentioned statements do not hold true.**

Linear and non-linear problems

- The most common causes of non-linearity in structural analysis:
  - Large displacements; geometric non-linearity
  - Non-linear material behavior; material non-linearity
    - Plasticity
    - Hyperelasticity
    - Material damage model
  - Contact
    - Opening and closing of contact
    - Change in the friction contact state: slip/stick
  - Non-linear connectors
    - Non-linear springs
    - Locking/opening/damage connectors

Geometric nonlinearity

Some examples of geometric non-linearity:

- Effect on load carrying mechanism
  - Shell begins to carry load in the membrane mode
  - Directions of support forces change (rope)
- Effect on loading
  - Change of load orientation or length of moment arm
    - crank mechanism
  - Surface area of pressure load changes
    - inflating a balloon

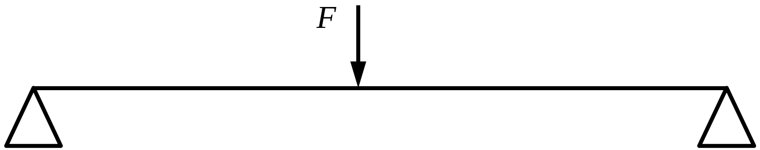
Geometric nonlinearity

When must geometric nonlinearity be considered?

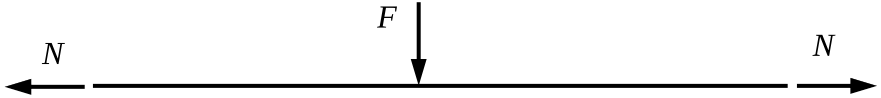
- **Good rule of thumb**: If the deformation is not visible by naked eye, there is likely no need to consider geometric nonlinearity.
- Some cases cannot be calculated without considering the geometric nonlinearity.
  - For example, a rope loaded in a transverse direction.

Geometric nonlinearity

- Example: rope, which has no bending stiffness
  - Carries only tension force in the longitudinal direction
  - Rope force N

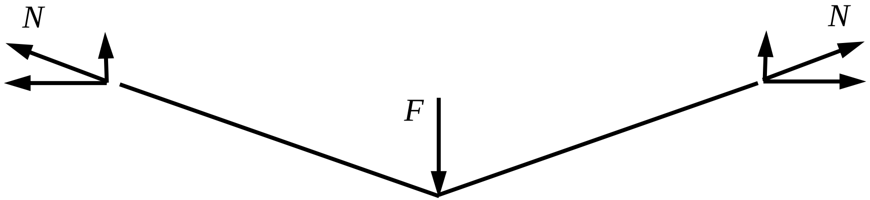


Free body diagram



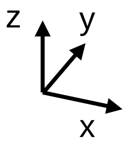
In the initial state, when the rope is straight, there is no solution for the equilibrium problem.

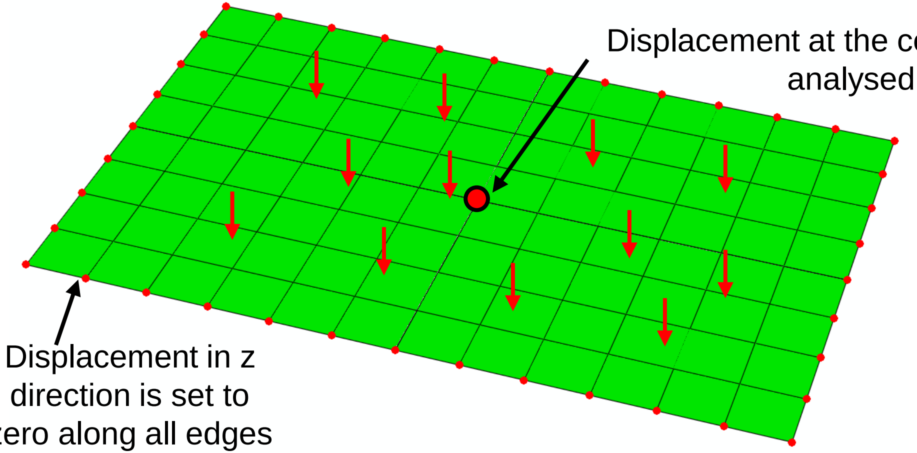
The equilibrium is found only, when the rope force N gets the vertical component at the support points due to deformation.



Geometric nonlinearity: Example

- Steel plate loaded on one side by evenly distributed pressure
  - dimensions: 600 mm x 400 mm
  - thickness: 1 mm
  - material: E = 200 GPa





Displacement at the center point is analysed

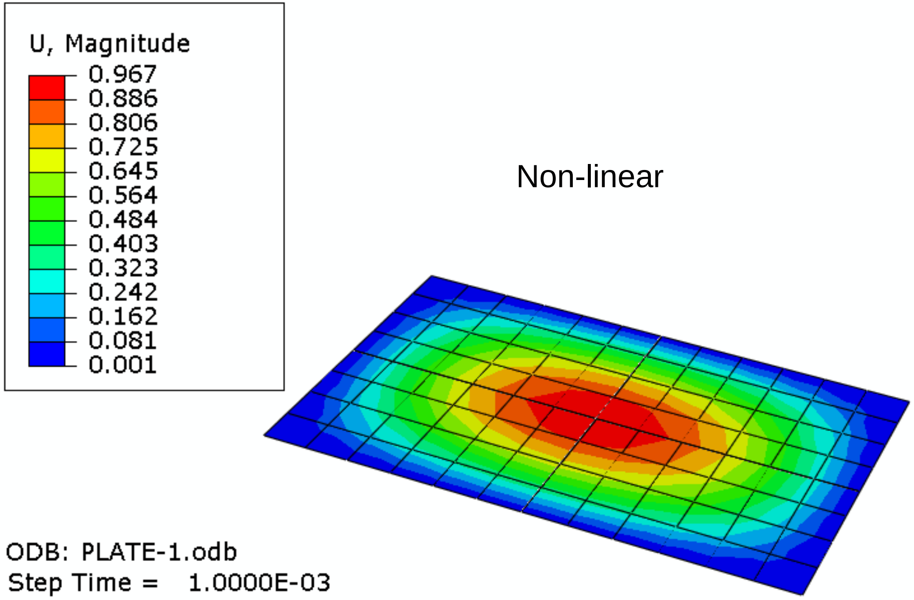
Displacements in xy plane constrained at 3 corners only to fix rigid body motions

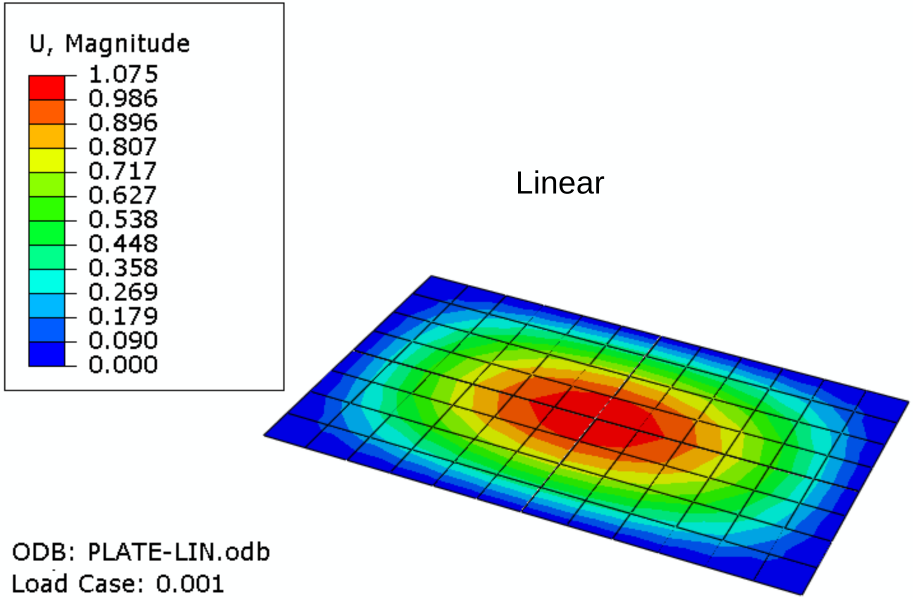
Displacement in z direction is set to zero along all edges

Geometric nonlinearity: Example

Loading 100 Pa

Displacement [mm]

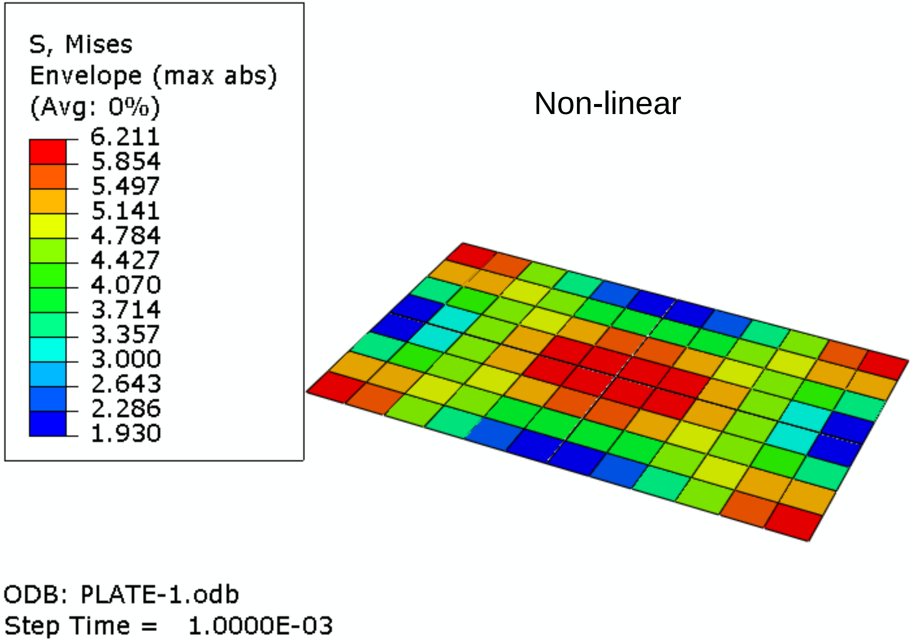


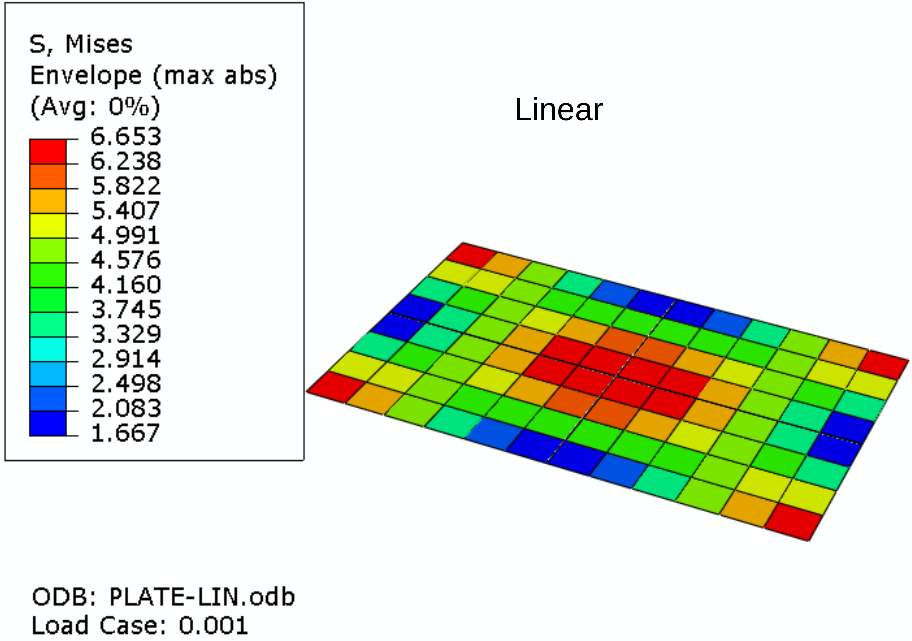


Geometric nonlinearity: Example

Loading 100 Pa

Stress [MPa]

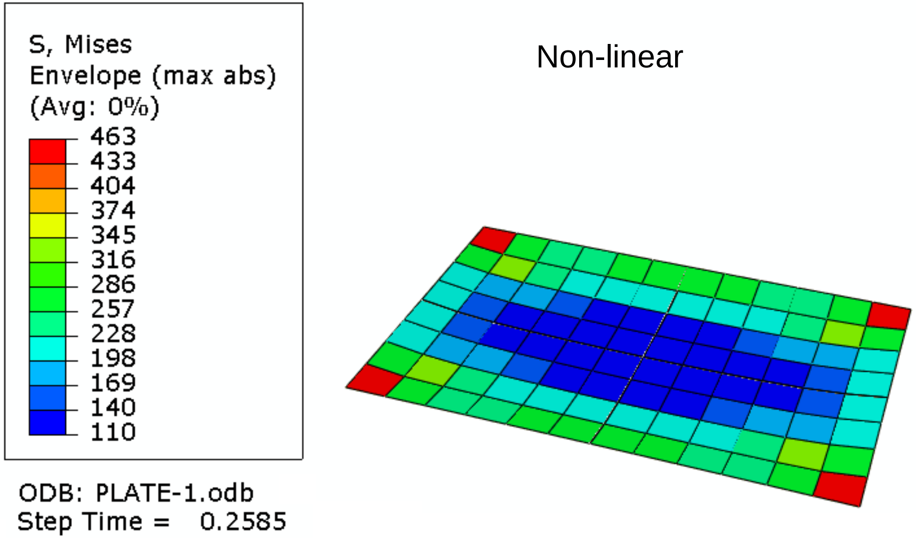


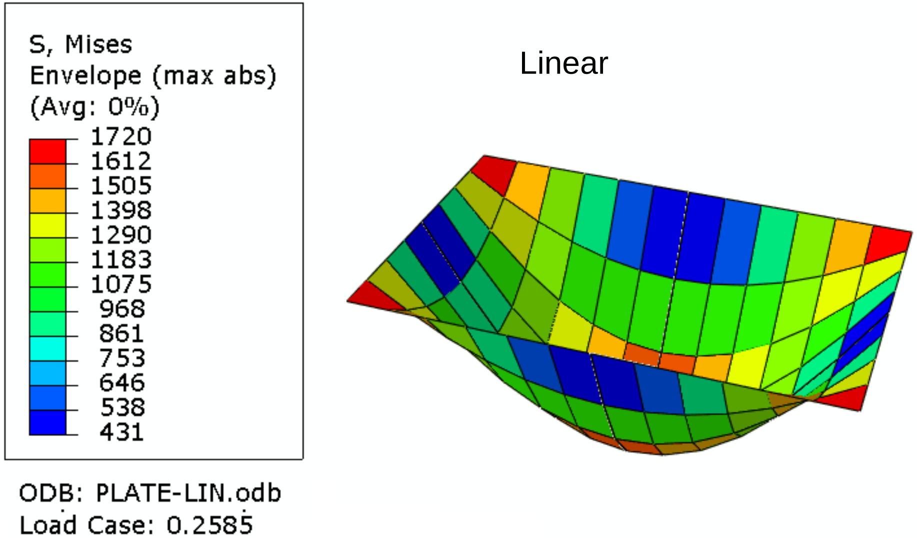


Geometric nonlinearity: Example

Loading 25850 Pa = 0.2585 bar

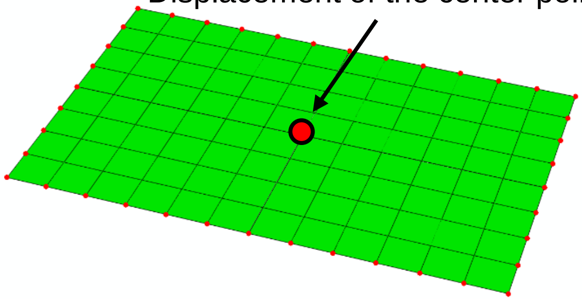
Stress[MPa]

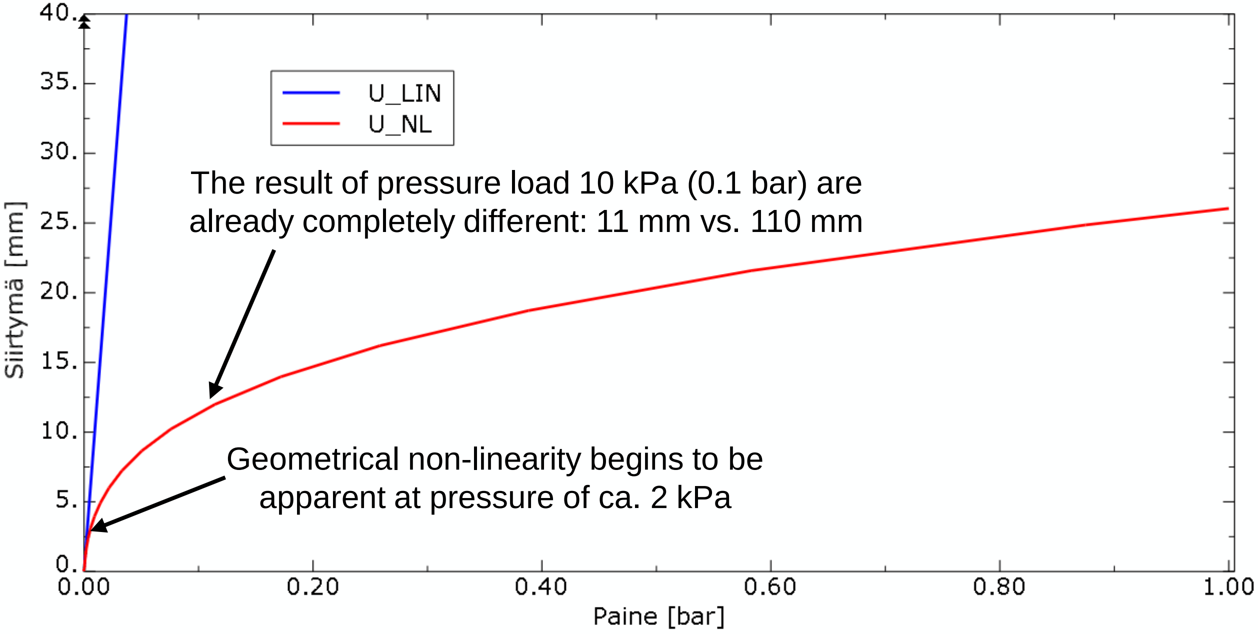




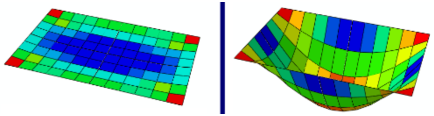
Geometric nonlinearity: Example

Displacement of the center point





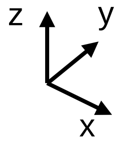
Geometric nonlinearity: Example

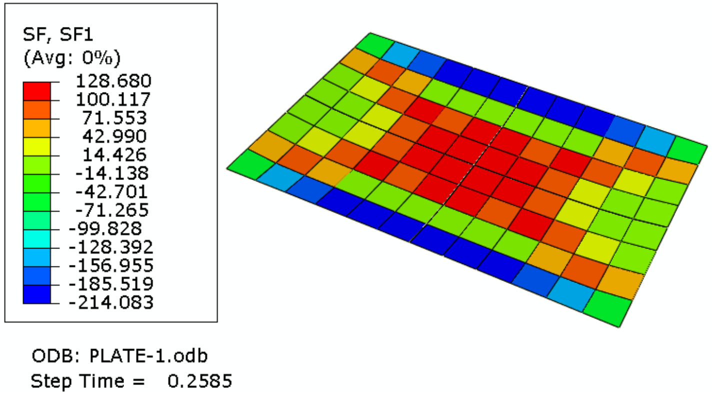


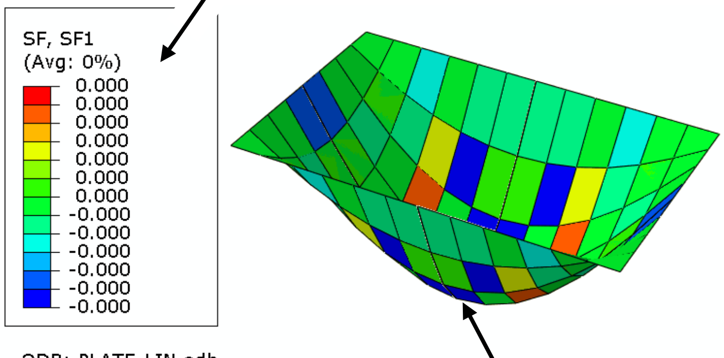
- Question: What causes this big difference?
- Answer: As the displacement increases, the plate begins to carry in the membrane mode.

SF1 = membrane force in plate [N/mm] in direction x

In the linear model, the membrane force is not generated. It is generated only from the geometrical non-linearity.







Colors are only decimal noise, magnitudes 1.e-13 to 1.e-15.

Geometric nonlinearity

Computing time

- In a linear problem the system of equations is solved only once
- Geometrically non-linear analysis requires iteration
  - The system of equations is solved in each iteration
  - Thus, each iteration cycle corresponds to roughly one linear analysis
  - Geometrically non-linear analysis computing time ~ 2 - 100 times the linear analysis
  - If there are other non-linearities, e.g., contact, yielding material, the counting time will increase even more
    - Iterations for material model equations and contact forces

Geometric nonlinearity

- Example of iteration history of the plate analysis involving only geometric nonlinearity

SUMMARY OF JOB INFORMATION:

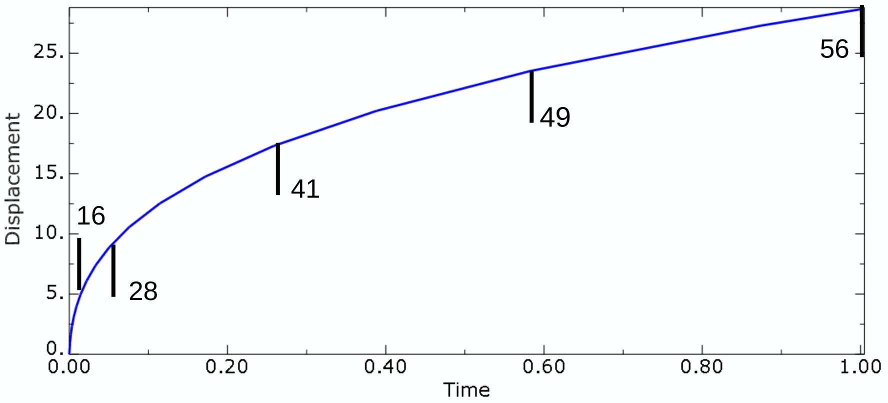
STEP	INC	ATT	SEVERE DISCON ITERS	EQUIL ITERS	TOTAL ITERS	TOTAL TIME/FREQ	STEP TIME/LPF	INC OF TIME/LPF	DOF MONITOR	IF RIKS
1	1	1	0	4	4	0.00100	0.00100	0.001000		
1	2	1	0	3	3	0.00200	0.00200	0.001000		
1	3	1	0	3	3	0.00350	0.00350	0.001500		
1	4	1	0	3	3	0.00575	0.00575	0.002250		
1	5	1	0	3	3	0.00913	0.00913	0.003375		
1	6	1	0	3	3	0.0142	0.0142	0.005063		
1	7	1	0	3	3	0.0218	0.0218	0.007594		
1	8	1	0	3	3	0.0332	0.0332	0.01139		
1	9	1	0	3	3	0.0503	0.0503	0.01709		
1	10	1	0	3	3	0.0759	0.0759	0.02563		
1	11	1	0	3	3	0.114	0.114	0.03844		
1	12	1	0	3	3	0.172	0.172	0.05767		
1	13	1	0	4	4	0.258	0.258	0.08650		
1	14	1	0	4	4	0.388	0.388	0.1297		
1	15	1	0	4	4	0.583	0.583	0.1946		
1	16	1	0	4	4	0.875	0.875	0.2919		
1	17	1	0	3	3	1.00	1.00	0.1252		

THE ANALYSIS HAS COMPLETED SUCCESSFULLY

Number of iterations

Time instant of the analysis

Displacement of the slab center point and required iteration rounds



Material nonlinearity

- Plastic material (more details later)
  - When the yield limit is exceeded, permanent deformations arise
  - Scope of the linear analysis: yield limit is not exceeded
- Large strains
  - Elastic strains are so large that the linear material model is not realistic
  - Elastomers, plastic materials, rubber
  - Scope of the linear analysis: maximum strain ca. 5%

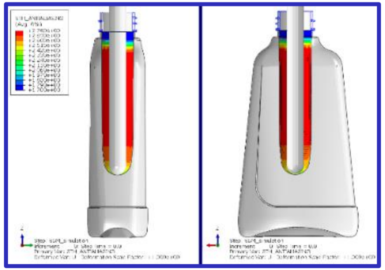
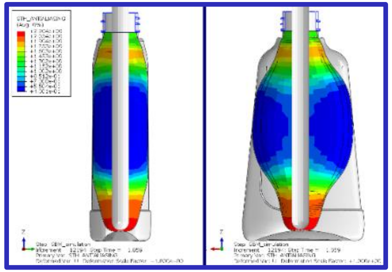
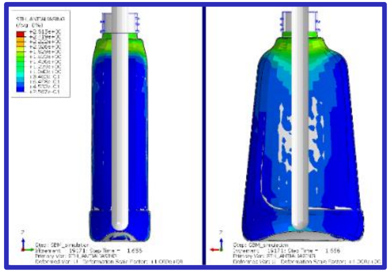
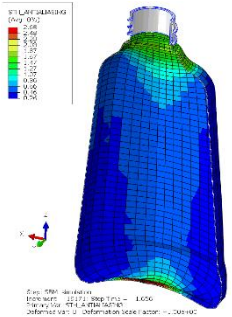
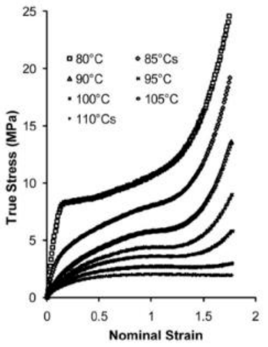
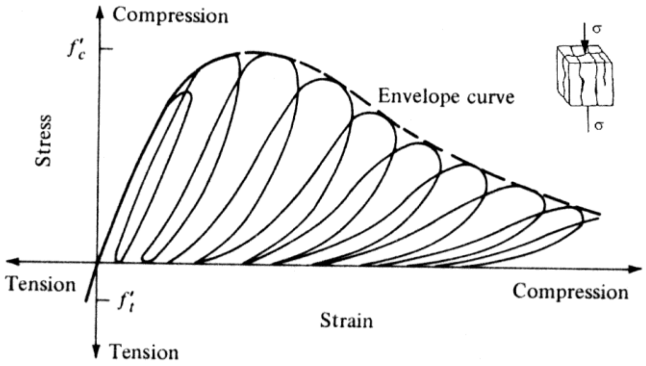
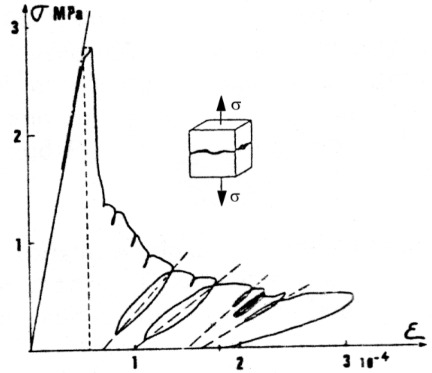






Figure 1. Influence of temperature on the behavior of poly(ethylene terephthalate) (PET) under equal biaxial (EB) deformation at a nominal strain rate of 1/s.

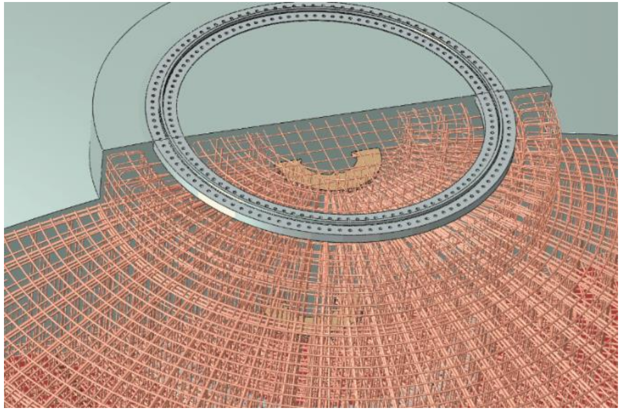
Material nonlinearity

- Damage models
  - The yield limit and the elastic stiffness change
    - E.g., concrete, figures below
  - Damage cannot be modeled by linear models





Reinforced concrete structure
- Reinforced elastic-plastic model
- Elastic-plastic concrete damage model

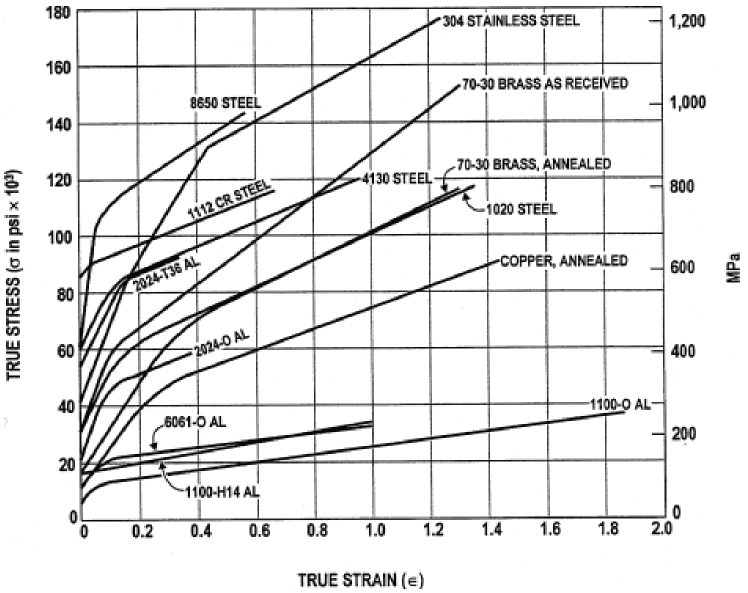


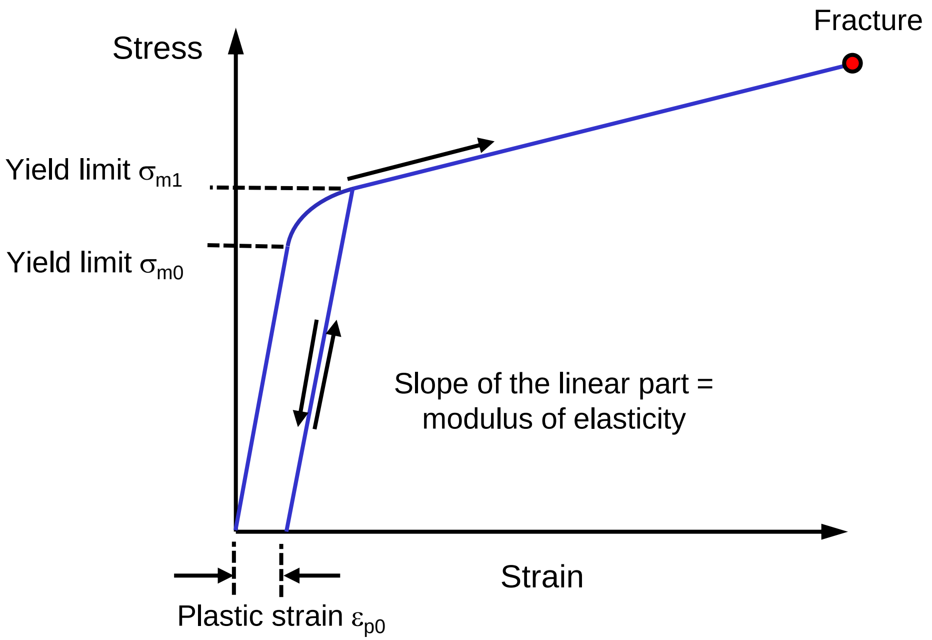
Plasticity

- When the load is high enough, permanent deformation begins to occur in the material
  - The deformation will not disappear even if the load is removed
- For metals von Mises plasticity (J2-theory) is most commonly used
- Use in Abaqus:
  - The program is provided with a stress-strain relation, obtained from a tensile test
  - Abaqus calculates the stresses, and based on them, the equivalent von Mises stresses
  - If the yield limit is exceeded, the program iterates until the stress is obtained at the yield limit
    - Iteration: Searching for a deformation state at which the stress is at most at the yield limit and the structure in balance
  - Iterations always increase the computational cost
    - From the calculation effort point of view, each iteration cycle corresponds to approximately one static linear analysis

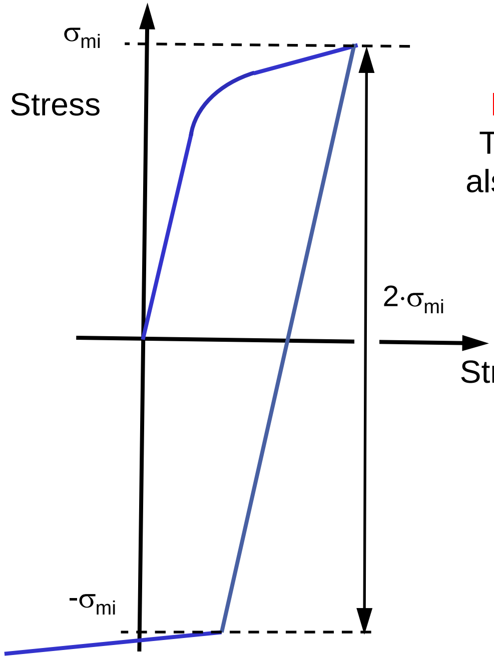
Plasticity, hardening

- Hardening: Yield limit grows with plastic deformation

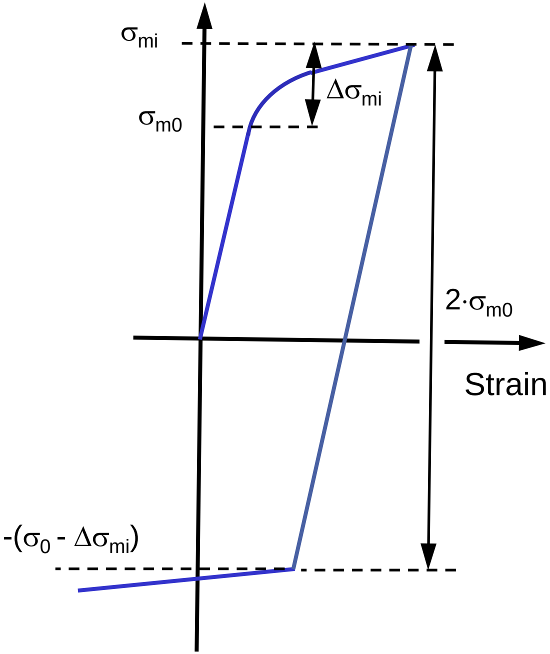




Plasticity, hardening



Isotropic hardening:
Tensile yield increases also on the compression side yield strength



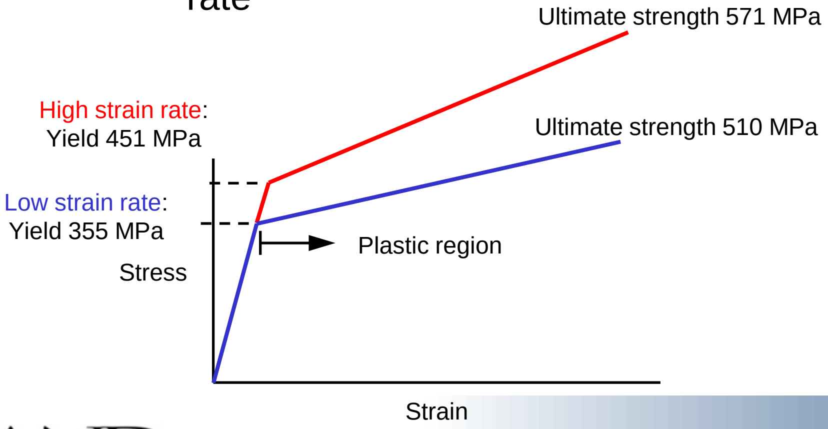
Kinematic hardening:
If the yield limit increases on the tension side, it decreases on the compression side.

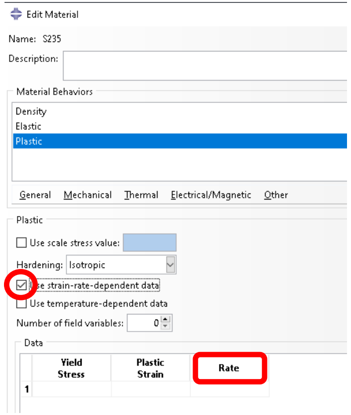
Bauschinger efect

Plastic material, strain rate dependence

*) Ref.: P.Soroushian, K-B.Choi, *Steel mechanical properties at different strain rates*, J. Struct. Eng. 1987, 113(4):663-672.

- The plastic behavior of structural steel depends on the loading rate *)
  - The yield and ultimate strength increase as the load speed increases
- Can be added to the model, *rate dependent plasticity*
  - The material parameters are given as a function of the strain rate





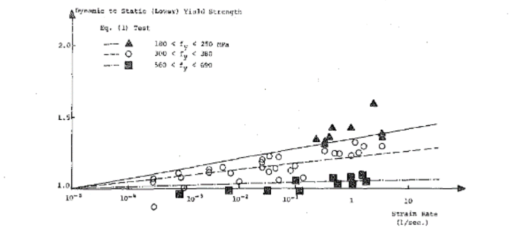


FIG. 2.—Strain Rate Effects on Lower Yield Strength of Steels with Different Yield Strengths

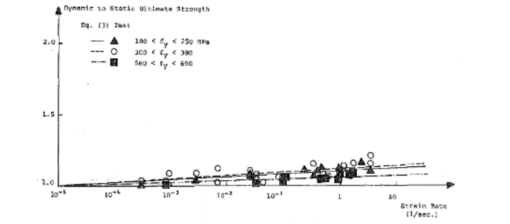
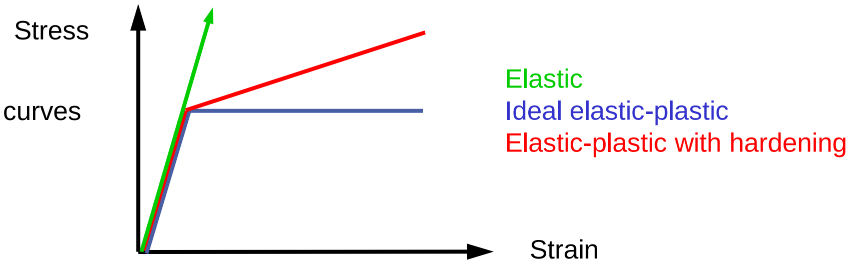


FIG. 4.—Strain Rate Effects on Ultimate Strength of Steels with Different Yield Strengths

Plasticity, usage recommendations

- If the load is monotonous (steadily increasing) or the load direction does not change, then
  - At simplest, give only one yield limit σ_{m0}
  - At the simplest, give only one ideal elastic-plastic material, no hardening
    - For numerical reasons, it is generally advisable to use a small hardening
  - Use isotropic hardening, yield limit is given as a function of plastic strain
  - These are easy to calibrate and to implement into the model, computationally light
- When the load is changing (tension-compression) use:
  - Either kinematic
  - Or combined isotropic+kinematic hardening
  - These are more difficult to calibrate, computationally heavy

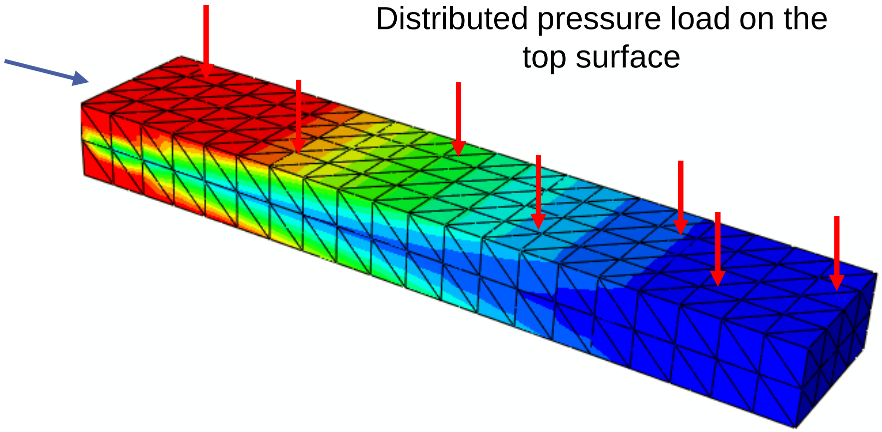
Stress-strain curves



Plasticity, example

- Cantilever beam, material S355 (yield limit 355 MPa, small hardening)

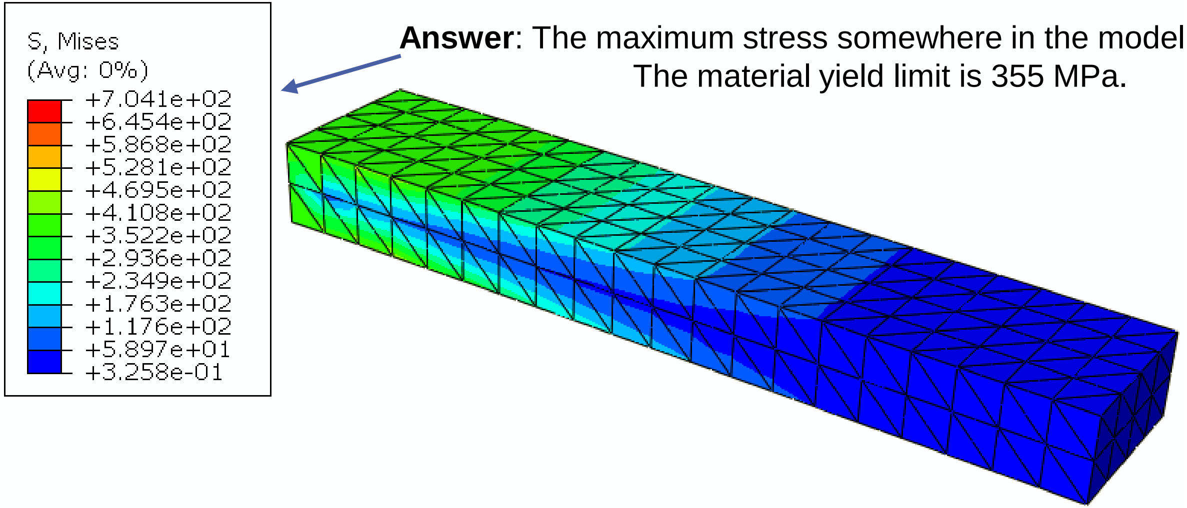
This end is rigidly fixed



Distributed pressure load on the top surface

Plasticity, example

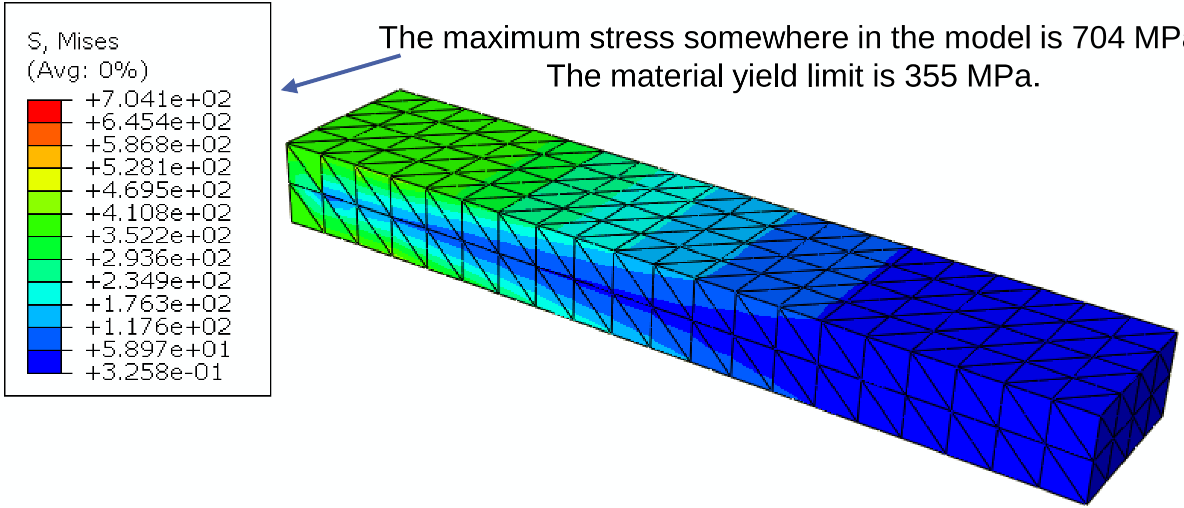
- Calculated with the second-order solid tetra elements (Abaqus C3D10)
- Von Mises [MPa] in the figure.
- **Question**: Is there something strange?



Answer: The maximum stress somewhere in the model is 704 MPa.
The material yield limit is 355 MPa.

Plasticity, example

- **Question**: Why is the stress higher than the yield limit?
- **Answer**: The yielding due to bending begins on the surface, however, the material model is evaluated at the integration points inside the element. The stress at the integration points has maximum value of 355 MPa, but when extrapolated, the stress on the surface becomes too large.



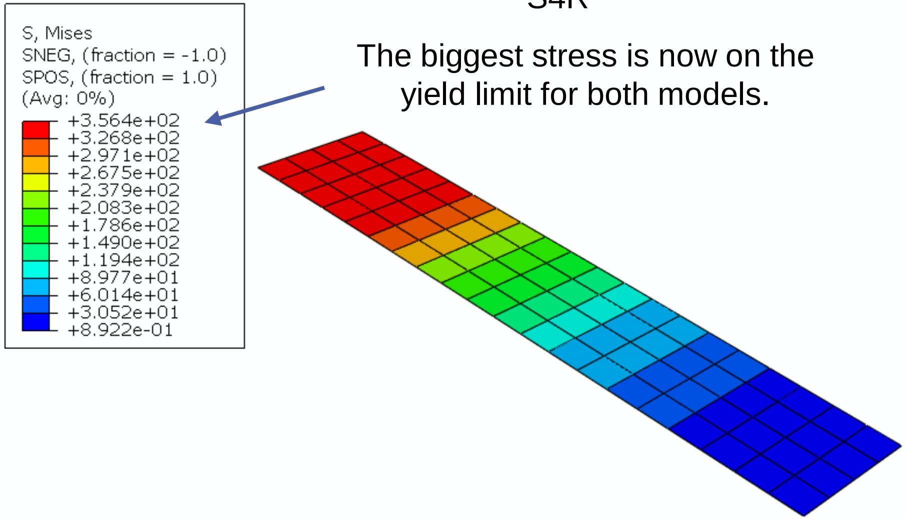
The maximum stress somewhere in the model is 704 MPa.
The material yield limit is 355 MPa.

Plasticity, example

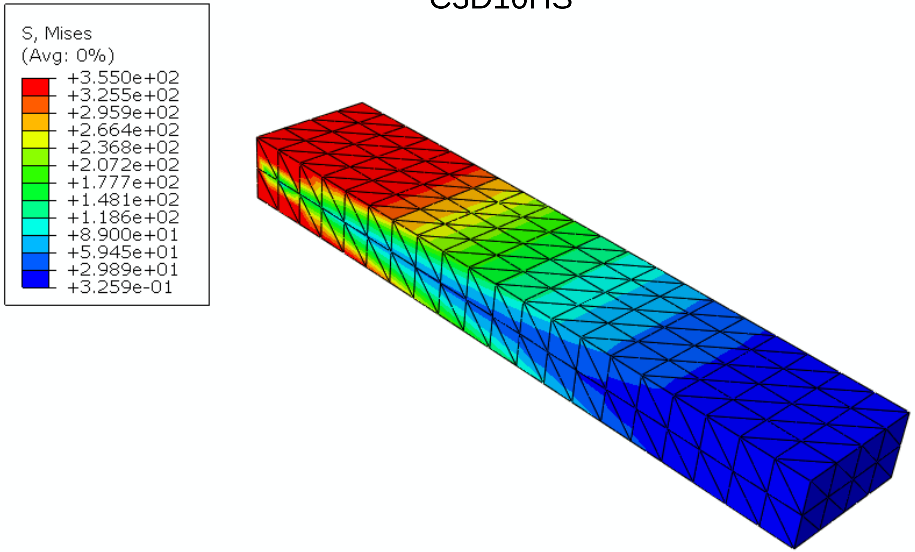
- When calculated with linear shell elements S4R and solid tetrahedrons C3D10HS (integration points on the surface)

S4R

The biggest stress is now on the yield limit for both models.



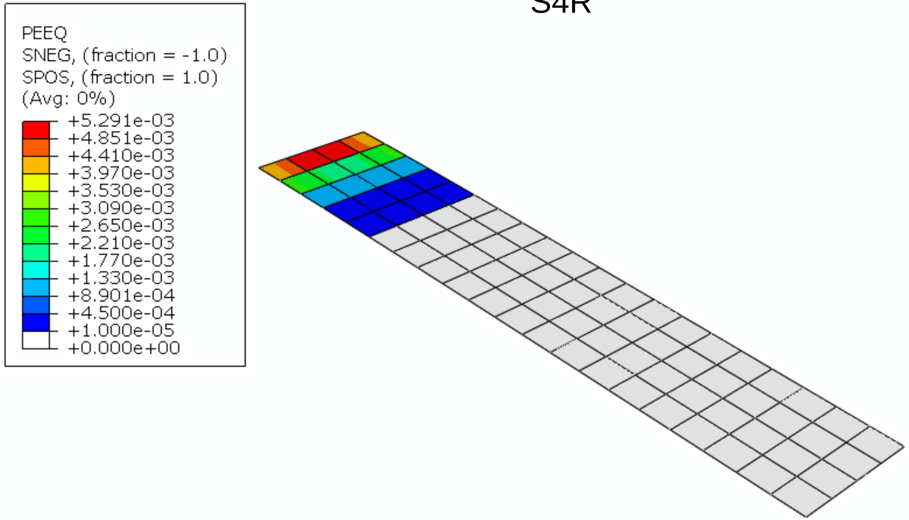
C3D10HS



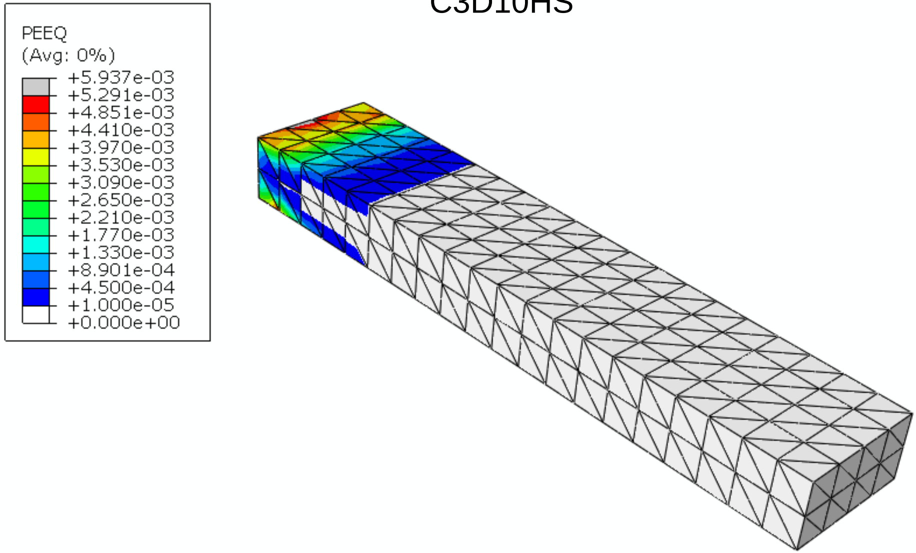
Plasticity, example

- The largest plastic equivalent strain: shell 0.53%, solid 0.59%

S4R



C3D10HS

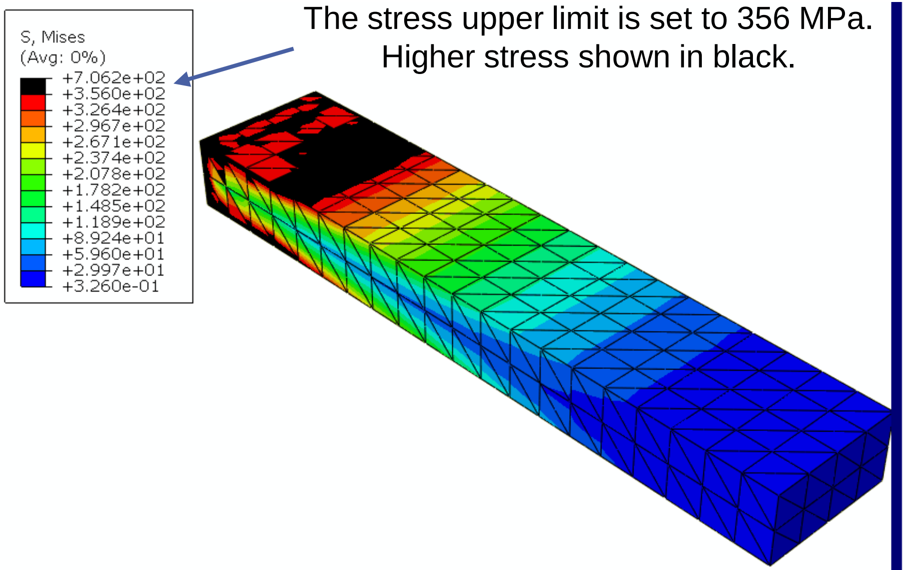


Plasticity, example

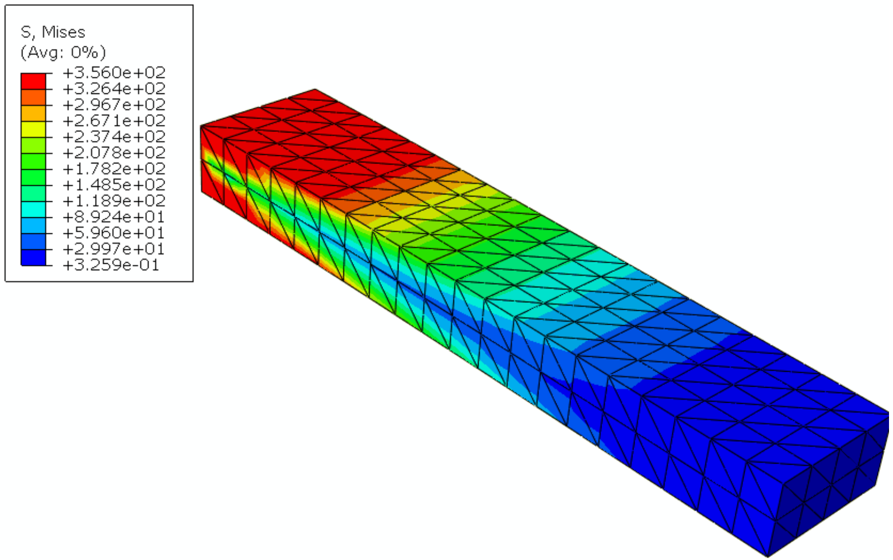
- However, even with ordinary solid tetras, the tension result is not as bad as it first appeared

C3D10

The stress upper limit is set to 356 MPa. Higher stress shown in black.

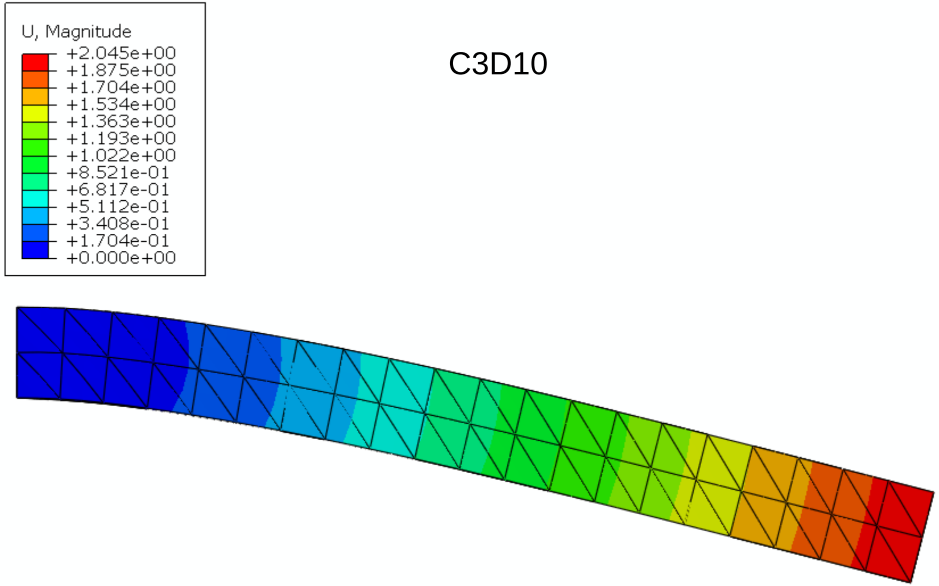


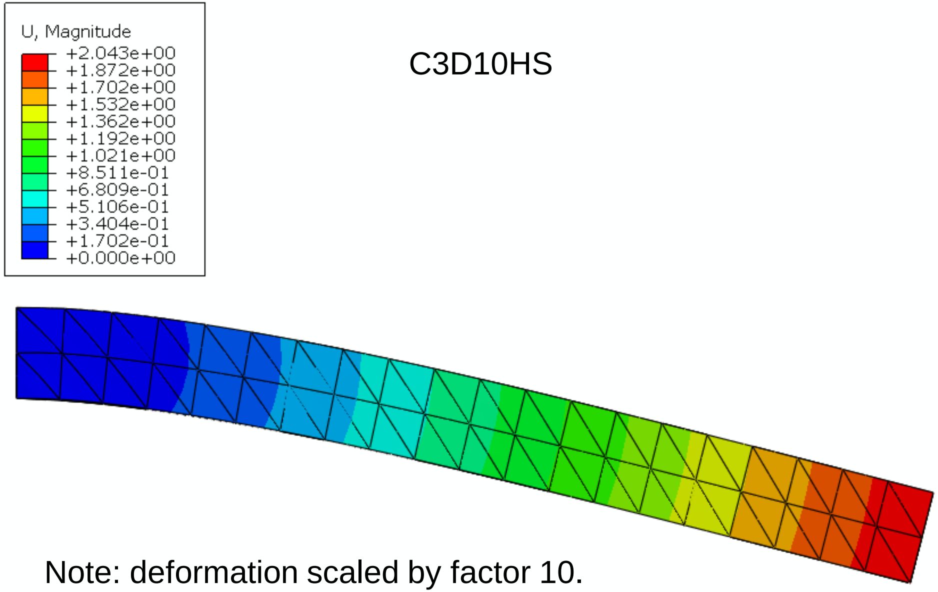
C3D10HS



Plasticity, example

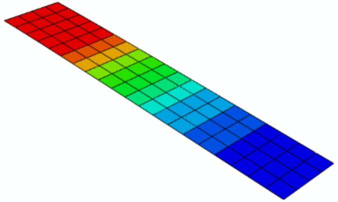
- The displacement result is practically the same in all models.

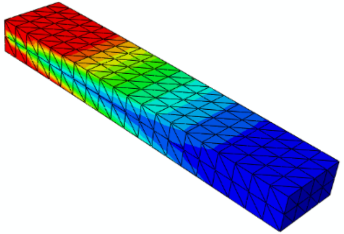




Note: deformation scaled by factor 10.

Plasticity, example





Number of DOF of the model

- Shell S4R: 630 (computationally by far the lightest)
- Solid model C3D10: 7446
- Solid model C3D10HS: 7840 (computationally the heaviest)

Question 1: Why does the shell element give the stress result correctly

Answer 1: The shell element has an integration points on the surface where the greatest bending stress is obtained. The shell elements are very good in describing yielding in bending. By default, Abaqus uses 5 integration points in the thickness direction of the shell.

Question 2: What should be considered, when using traditional solid elements in analyses involving plasticity?

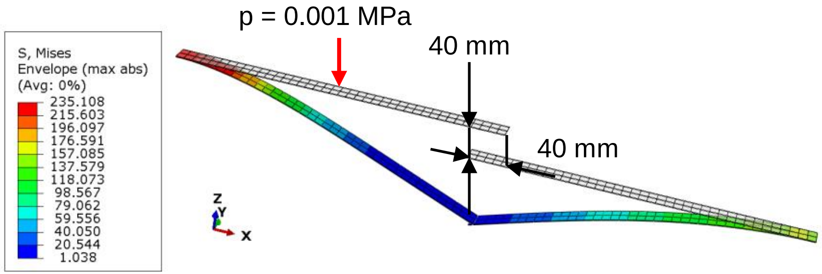
Answer 2: The mesh must be file enough close to the surface to get integration points close to the surface.
If being cautious, the higher stresses than yield limit can be accepted. The color scale should be changed in the pictures.

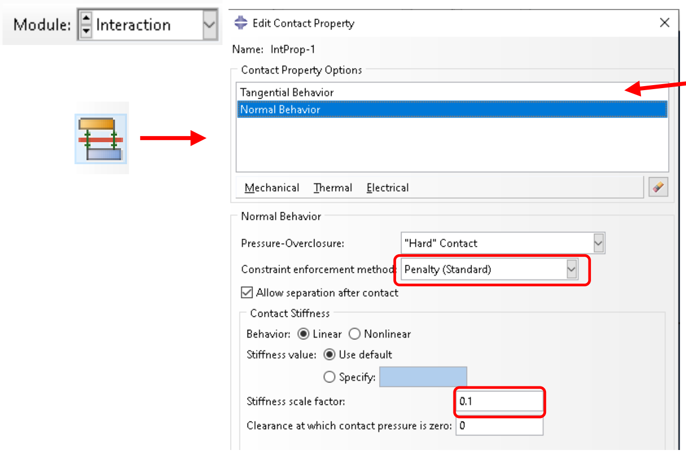
Question 3: Why do C3D10HS elements have more DOF than the C3D10, even though both models have the same number of elements?

Answer 3: The C3D10HS elements incorporate the internal pressure DOF in the calculation. H = Hybrid element

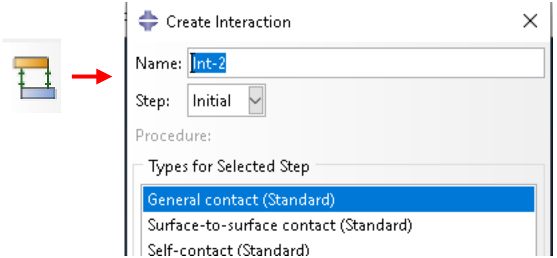
Workshop; Contacting beams

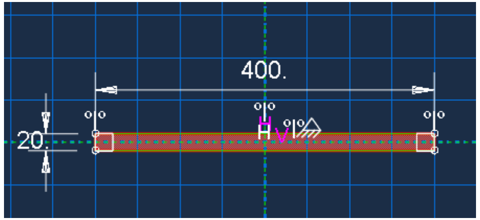
- Use a shell element model
- Dimensions [mm], L x W x t = 400 x 20 x 1
- E = 200 000 MPa, ν = 0.3
- Pressure load on top beam: 0.001 MPa
- Interaction properties for contact





Tangential behaviour:
Frictionless



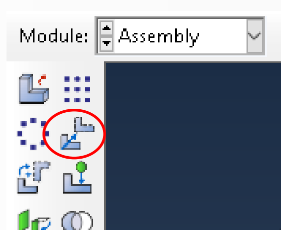


Sketch of planar shell

Interaction for contact: *General contact, All with Self*

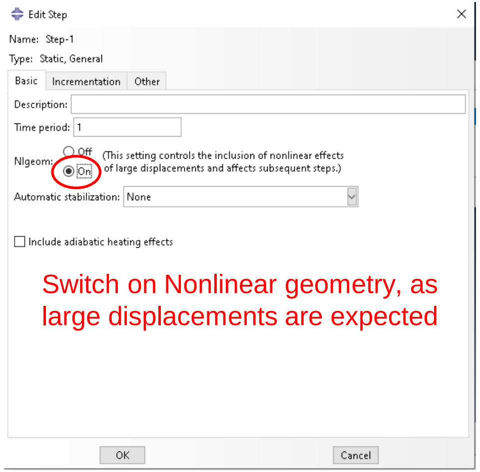
Steps

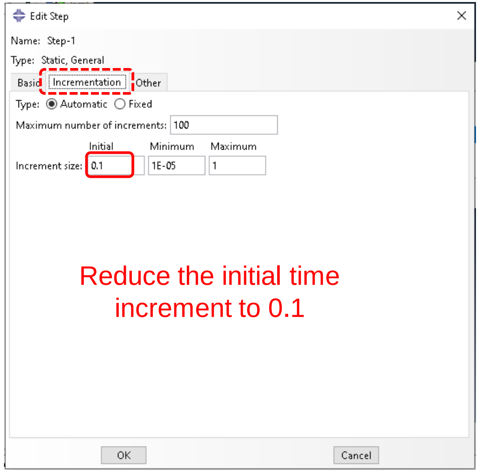
- Create a shell part. Two options for base feature
  - Planar shell
  - Extruded shell
- Create material and shell section
- Assign shell section
- Mesh part
- Create two instances of the same part in the Assembly
  - Both beams are identical, no need to define two identical parts. Instead, two instances of the same part are created in the assembly
- Position the two beams using the *Translate Instance* tool



Steps, continued

- Create an analysis step: *Static, General*





In static analysis time does not have a physical meaning. It is used to increment the loads and describe the order in which thing happen.

The time period of the step is set to 1. The initial increment is set to 0.1. This means, that Abaqus will try to apply 10% of the load and find an equilibrium state for the structure. After this Abaqus increases the load and again finds an equilibrium state. This process is repeated until the full load has been applied.

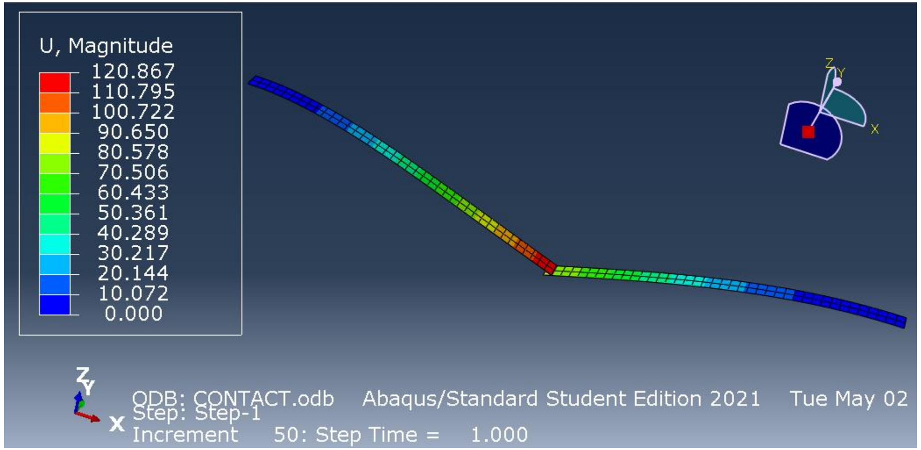
The incrementation of the load is automatically adjusted by Abaqus based on the convergence history.

- In the *Interaction module* create a *Contact property* and *Contact interaction* as shown on the previous slide
  - By default parts in the assembly do not interact with each other in any way
  - The analyst must define contacts or constraints between the parts

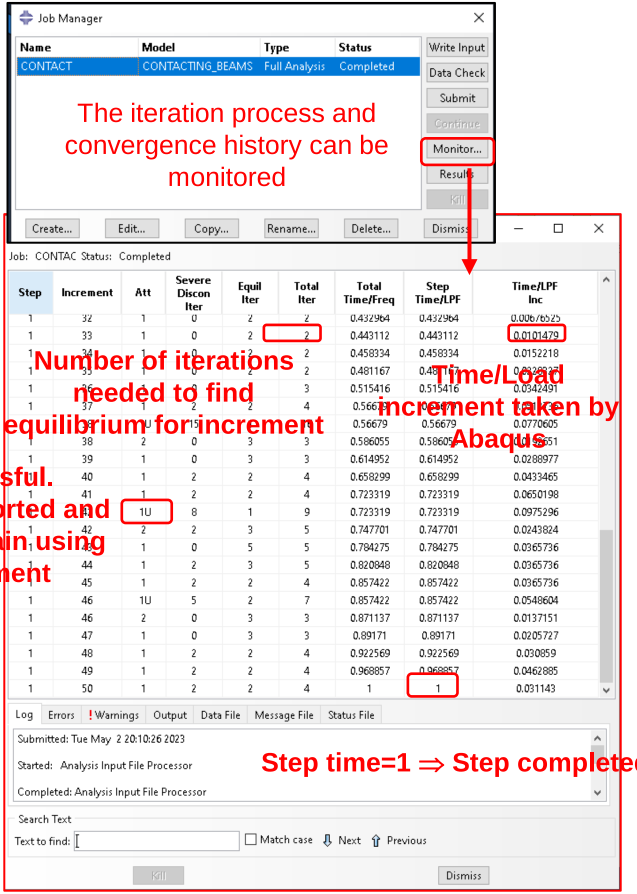
Steps, continued

- Create an analysis Job and submit the analysis
- Check the results

Displacements of contacting beams



U = Unsuccessful. Increment aborted and attempted again using smaller increment



Material plasticity

- Plasticity data are entered using true stress and logarithmic plastic strain.
- If necessary, convert nominal strain to log strain using the equation (1) given on the right
- When a material has incompressible behavior (as is the case for rubber and most metals when they deform plastically), the relationship between “true” stress and nominal stress is given by equation (2) on the right
- Use the following relation to determine the log plastic strain:

$$\varepsilon^{pl} = \varepsilon - \varepsilon^{el} = \varepsilon - \sigma / E.$$

(1) $\varepsilon = \ln(1+\varepsilon_{\mathrm{nom}}).$

Log strain

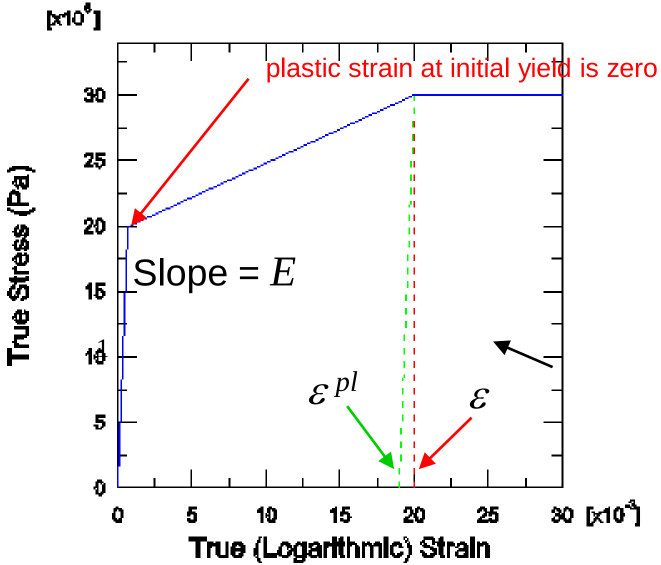
Nominal strain

(2) $\sigma = \sigma_{nom}(1+\varepsilon_{nom}).$

True stress

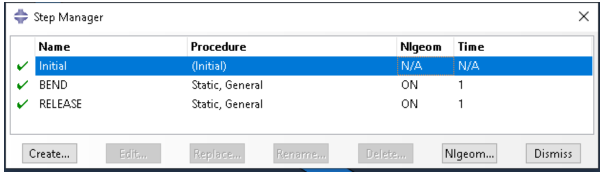
Nominal stress

Nominal strain

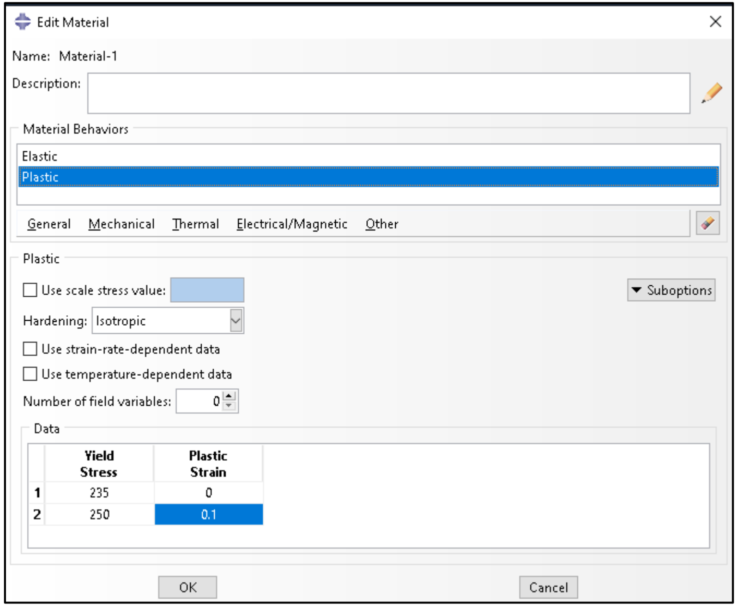


Material plasticity

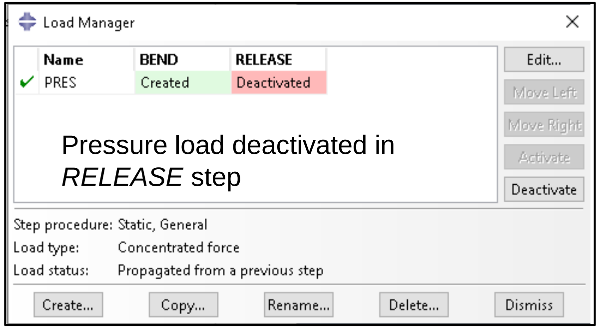
- Adding plasticity to Contacting beams example

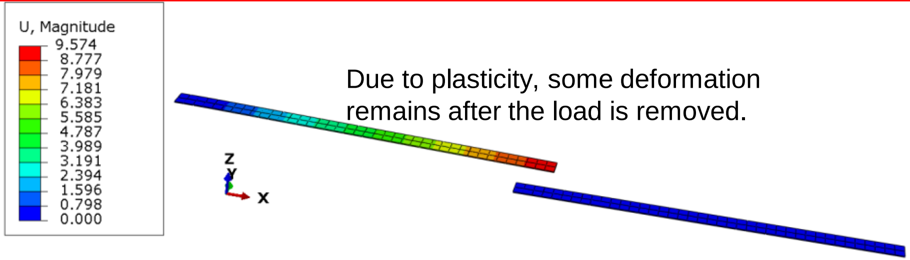


RELEASE step added



Plasticity added into material definition

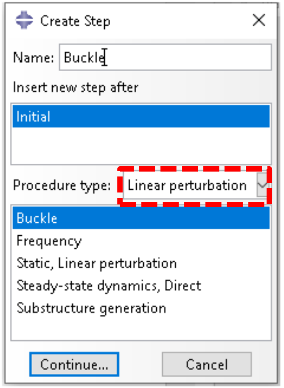


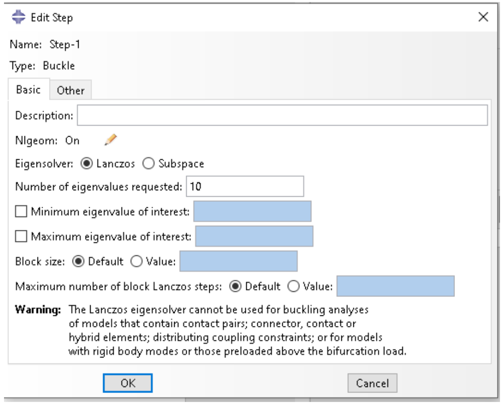


Due to plasticity, some deformation remains after the load is removed.

Linear buckling analysis

- Possible starting points for buckling analysis
  - As first analysis step: Geometry as modelled, Load: Only buckling load
  - As first analysis step with *INITIAL IMPERFECTION: Geometry modified by imperfection, Load: Only buckling load
  - After a general step with NLGEOM=ON: Deformed geometry from previous step, Load: Loads from previous step as “dead loads”, Buckling load with respect to loads applied in Buckling step
- Creating a Buckling analysis step

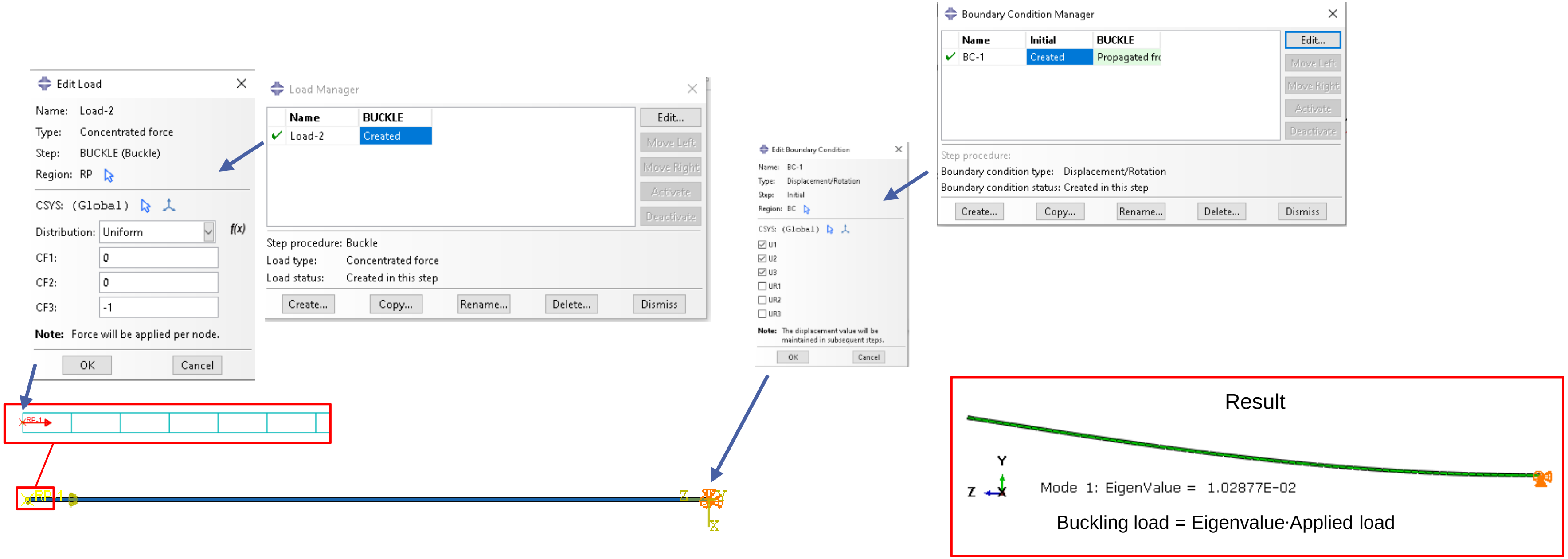




Suggested to start with Lanczos eigensolver with default settings

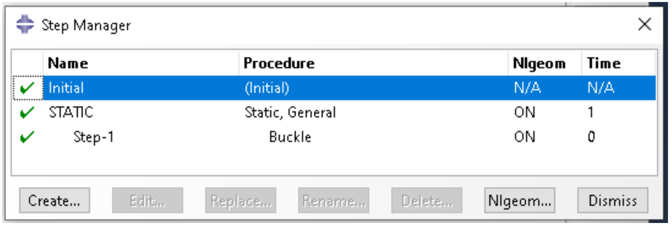
Linear buckling analysis

- Example: Buckling analysis as the first step:



Linear buckling analysis

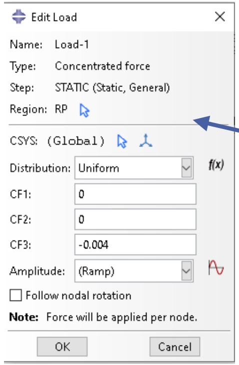
- Example with “dead load”

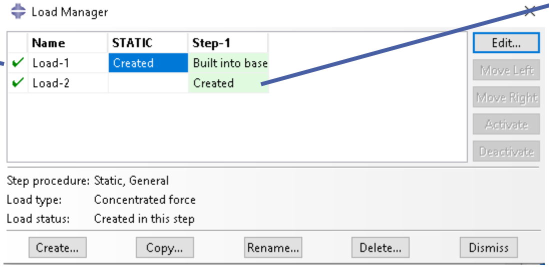


Apply dead loads, Calculate static equilibrium

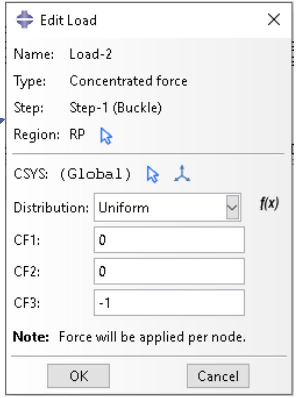
Apply active loads, Calculate buckling

Dead load in static step



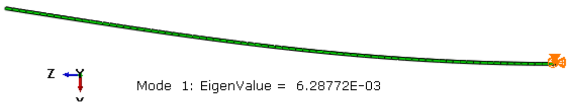


Active load in Buckling step



Result

Buckling load = Dead load + Eigenvalue·Active load



Deads load	0	0.004
Active load · Eigenvalue	1.029E-02	6.288E-03
Total	1.0288E-02	1.0288E-02

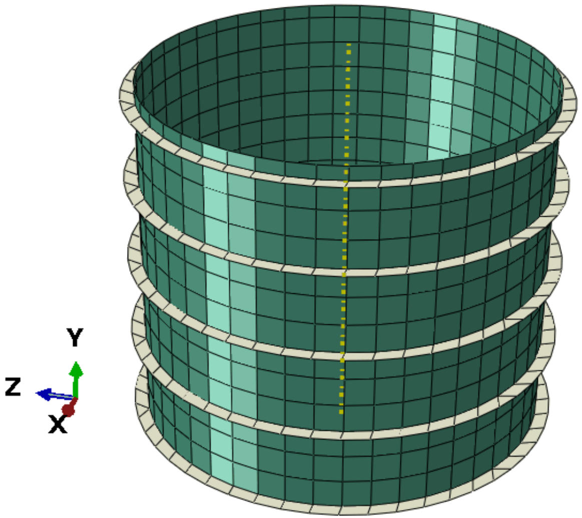
Linear buckling analysis

General notes

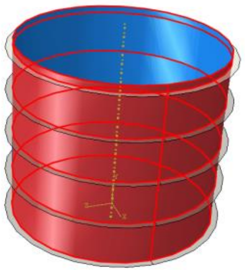
- Eigenvalue is with respect to all active loads acting simultaneously
  - Most load types can be used
  - Nonzero boundary conditions can be applied
- The Lanczos eigensolver cannot be used with
  - a model containing hybrid elements or connector elements
  - distributing coupling constraints
  - contact interactions
  - a model that has been preloaded over the buckling load
  - a model that has rigid body modes

Buckling of a cylindrical shell

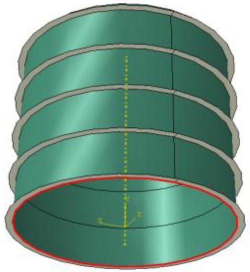
- Material. E = 70 000 MPa, ν = 0.3
- Height = 520 mm, Radius = 300 mm
- Distance between reinforcement rings = 125 mm
- Thicknesses: Cylinder: 5 mm, Rings: 10 mm

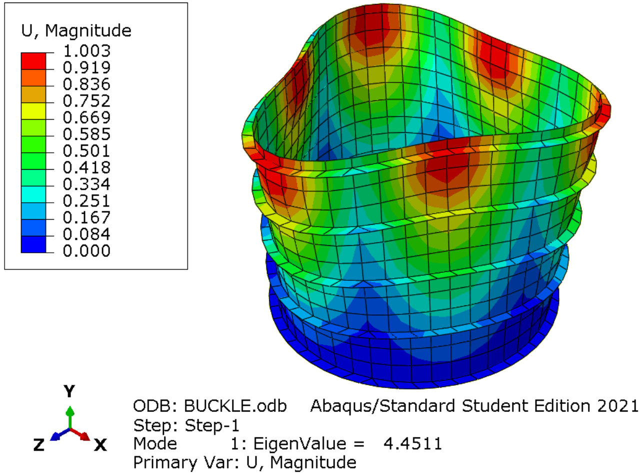


Unit pressure load on outer surface



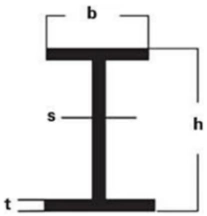
Translations fixed along bottom edge





Dynamic analysis of I-profile cantilever beam

- IPE80
  - h = 80
  - b = 46
  - t = 5.2
  - s = 3.8



- E = 206 GPa, ν = 0.3, ρ = 7.85e-9 Ton/mm^3
- p = 0.05 MPa
- L = 2000 mm
- Use shell elements

Task 1: Static analysis

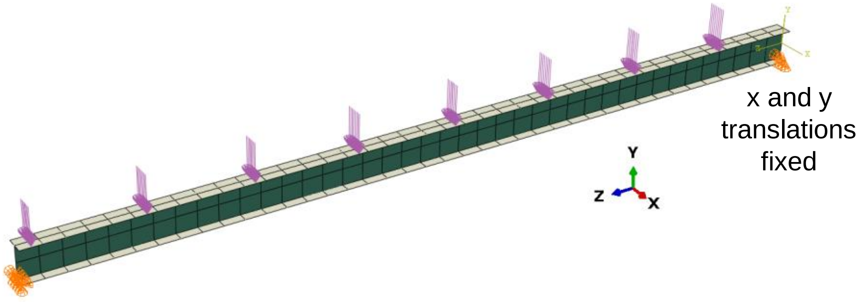
- Calculate static deformation for reference

Task 2: Vibration modes

- Find frequency of vibration mode that is likely to be excited by dynamic loading

Task 3: Transient dynamic analysis

- Calculate transient dynamic response (see next slide)

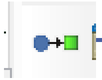


Assumption of static analysis: Load is applied slowly, vibrations are not excited.

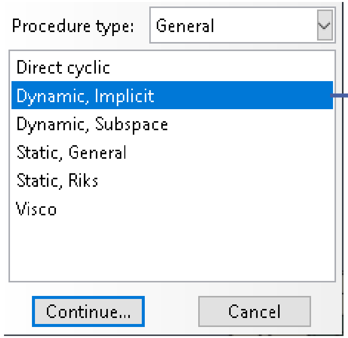
How much higher is the maximum displacement, if the same pressure load is suddenly applied on the beam?

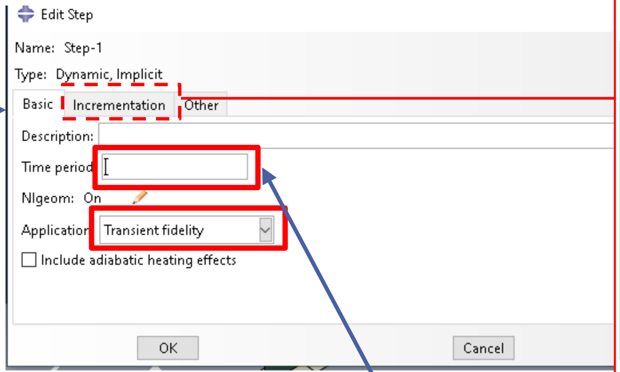
Dynamic analysis of I-profile cantilever beam

- Step definition for transient dynamic step



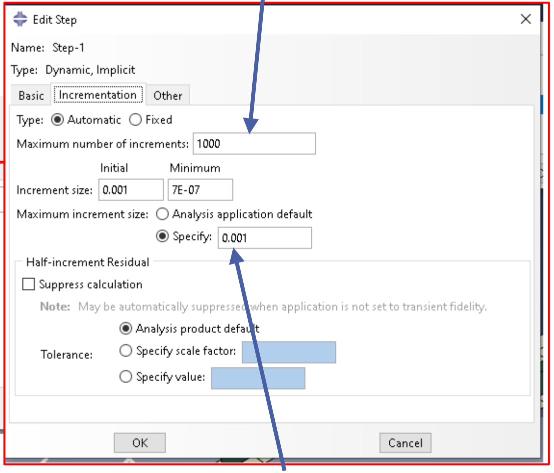
Create step





Based on the frequency of the eigenmode, set the duration of the step to approximately 5 vibration cycles

Increase the maximum number of time increments to ensure that analysis does not end prematurely



Set the maximum time increment to approx. 1/10 of the period of the eigenmode. This ensures that enough time points are used and the time history output is accurate.

Simulation

Let **Knowledge** be your guide

Where to find more information?

- Abaqus *Example Problems*, *Verification,* and *Benchmarks* in documentation
  - Input files and other input data for all examples are available
  - Example:

Abaqus › Example Problems › Static Stress/Displacement Analyses ›

Static and quasi-static stress analyses › Axisymmetric analysis of bolted pipe flange connections

Input files

boltpipeflange_axi_solidgask.inp

Axisymmetric analysis containing a gasket modeled with solid continuum elements.

boltpipeflange_axi_node.inp

Node definitions for boltpipeflange_axi_solidgask.inp and boltpipeflange_axi_gkax6.inp.

boltpipeflange_axi_element.inp

Element definitions for boltpipeflange_axi_solidgask.inp.

boltpipeflange_3d_solidgask.inp

Open and copy/paste to text file or use Abaqus fetch utility:

```
abaqus fetch job=boltpipeflange_axi_solidgask.inp
```

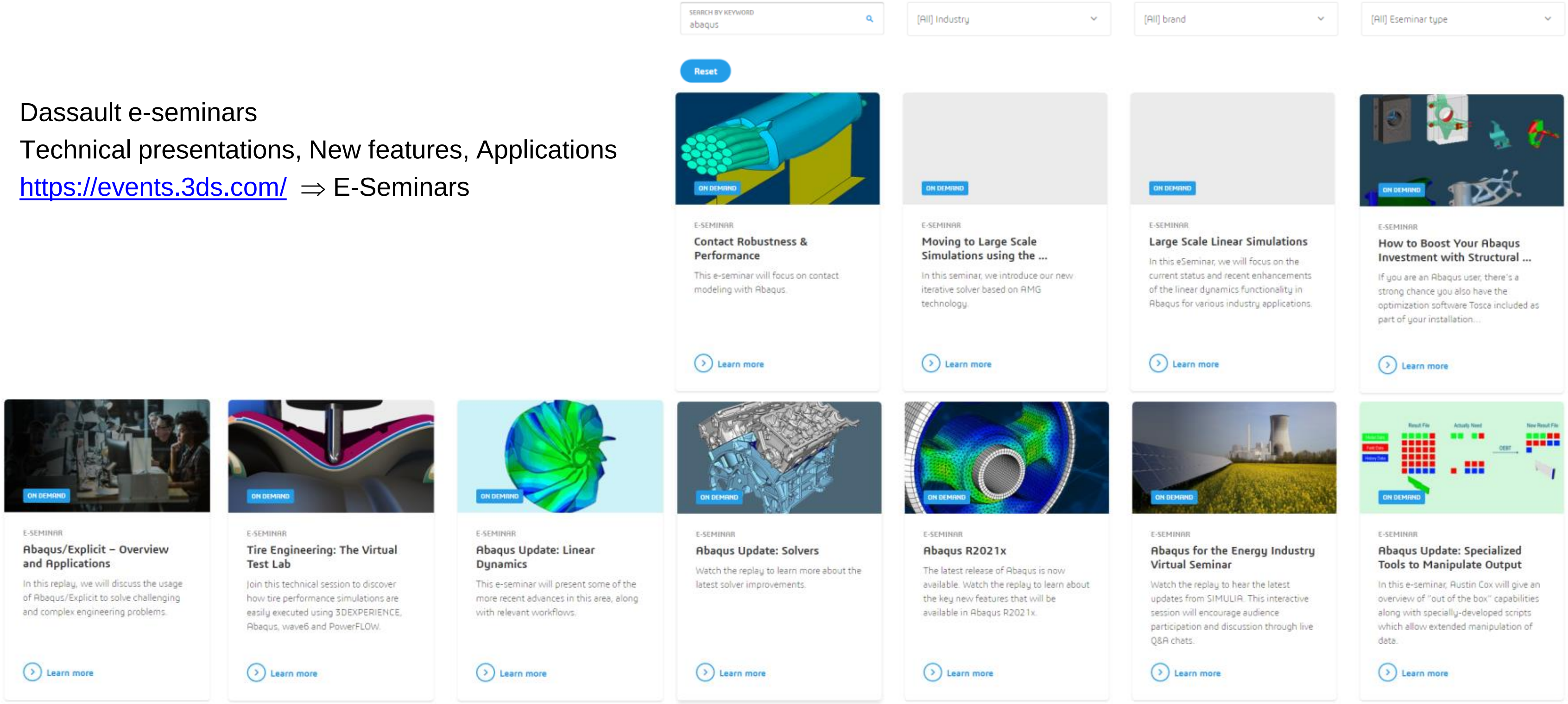
https://help.3ds.com/2020/English/DSSIMULIA_Established/SIMACAEEXARefMap/simaexa-c-boltpipeflange.htm?contextscope=all&id=f88c0b11fa174c71b902ec0a53859db4

Where to find more information?

Dassault e-seminars

Technical presentations, New features, Applications

https://events.3ds.com/ ⇒ E-Seminars



Thank you!