MEC-E1005 Abaqus workshop

PLM Goup
Gate8 Business Park
Äyritie 8E
01510 Vantaa
Finland

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

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

Overview of Abaqus/CAE and Abaqus solvers

WE MAKE
YOU INNOVATE

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

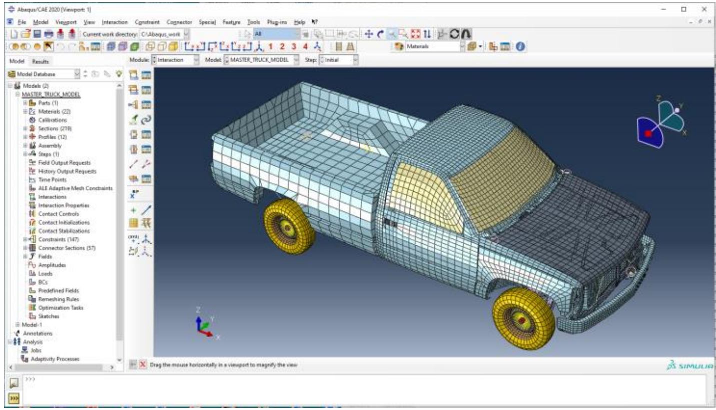
- feature-based, parametric modelling
- interactive and scripted operation
- GUI customization

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

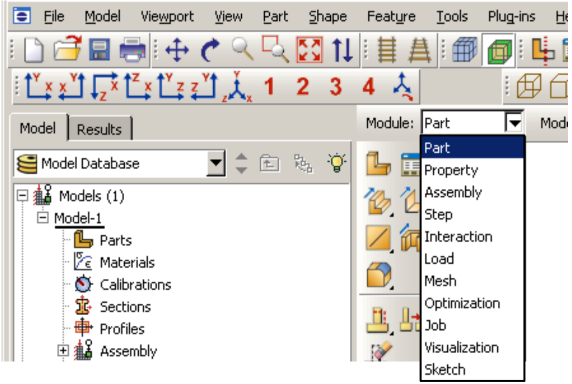


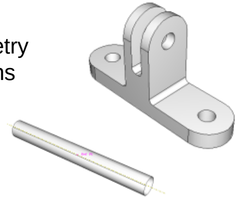
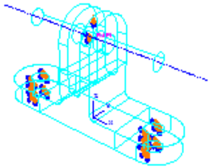
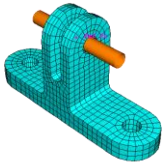
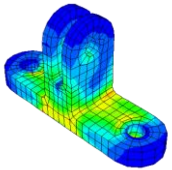
Overview of Abaqus/CAE

Functionality is presented in modules.

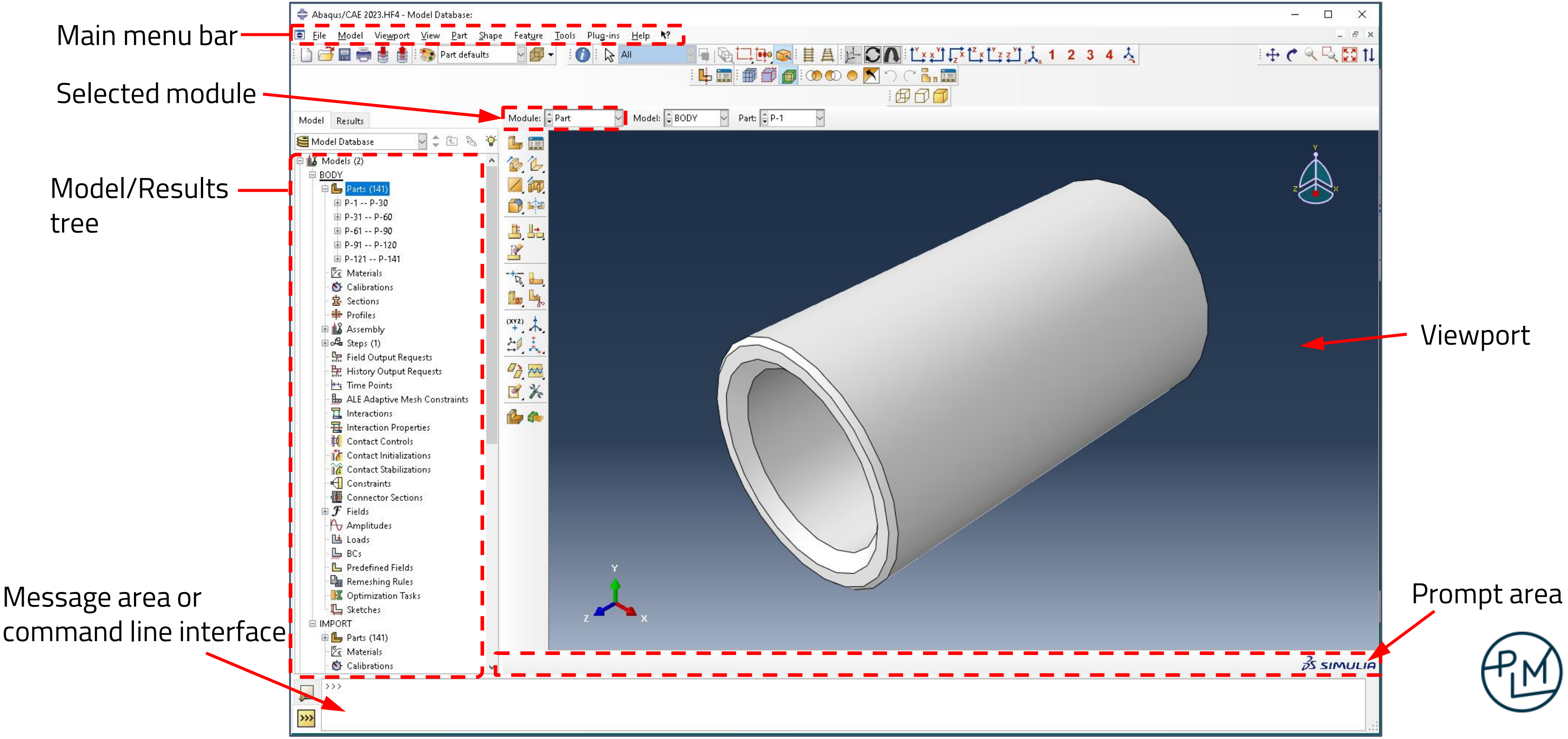
Each module contains a logical subset of the overall functionality.

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 

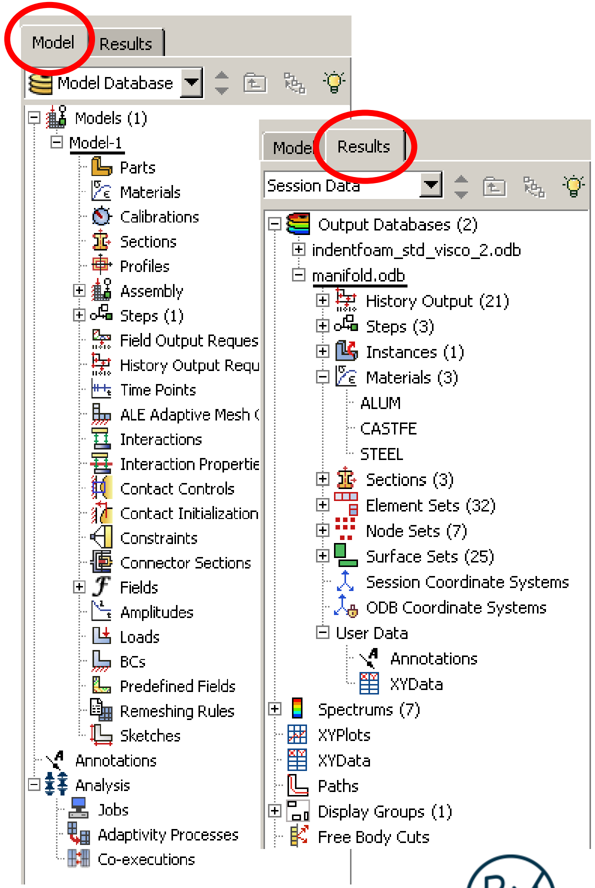
Overview of Abaqus/CAE



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.



Overview of Abaqus/CAE

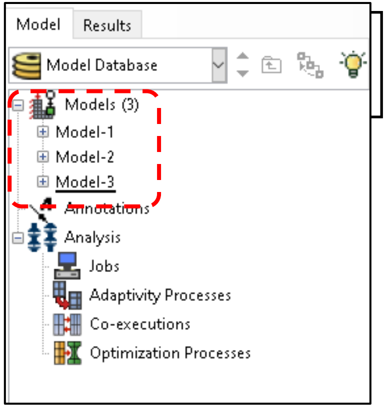
What is a model database file (extension `.cae`)?

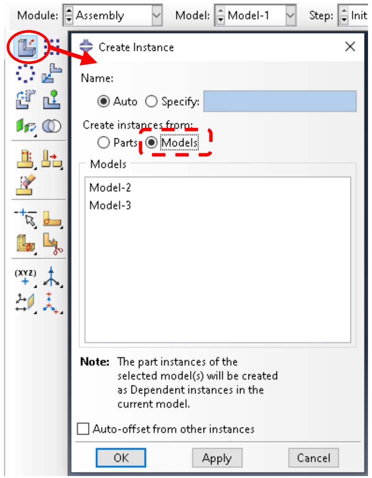
- Contains all the information for any number of models.
- Typically contains one model or several related models.
- Only one model database can be opened in Abaqus/CAE at a time.

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, unless an instance of another model is created in the assembly
  - Objects such as parts and materials can be copied between different models in the same database: Main menu > Model > Copy Objects
- Contains a single assembly including the associated contact interactions, loads and boundary conditions, mesh, and analysis history.

Model database (**.cae**)





Starting Abaqus/CAE

1. Command line

   abaqus cae

   Opens most recently installed release of Abaqus/CAE in current directory

   abq2023HF2 cae

   Opens specific release of Abaqus/CAE in current directory

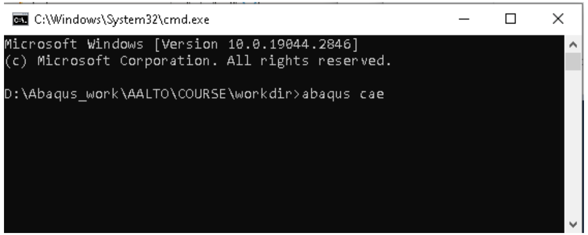
2. Windows Start menu

   Opens Abaqus/CAE in startup directory set during installation

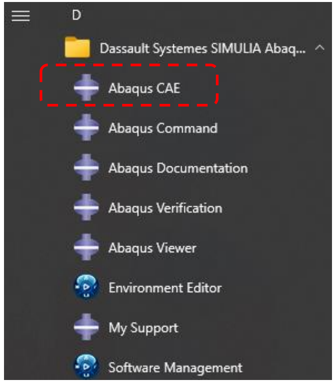
3. Double-click **.cae** or **.odb** file in Windows folder

   Opens Abaqus/CAE in current directory

Option 1



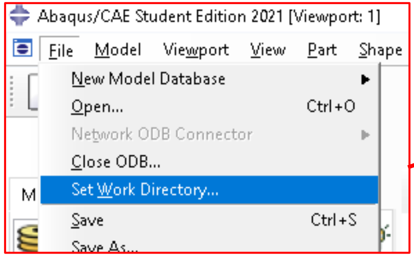
Option 2

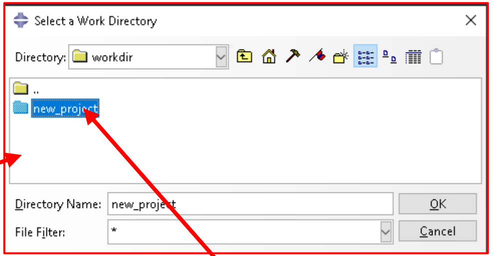


Suggested workflow when starting a new project

- Create a new directory for the project
- Start a command prompt in the new directory and start Abaqus/CAE from the command line

or Start Abaqus/CAE from Start menu and then select the new directory created for the project



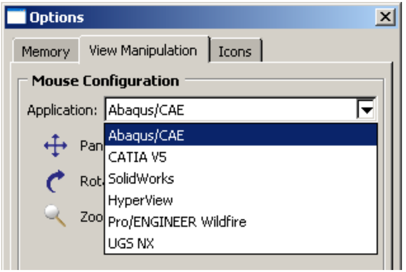


Select directory (Do not open/go into directory)

Now all analysis and results files are written in this directory and things stay organized

Otherwise, the default work directory given during installation is used for all files

Basic use of Abaqus/CAE



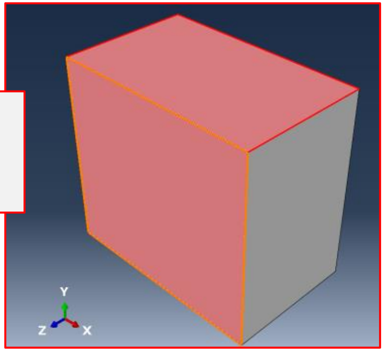
View manipulation

- Toolbar to control view (pan, zoom, rotate, etc.).
- Alternatively, can use a combination of keyboard and mouse actions:
  - Rotate: [Ctrl]+[Alt]+MB1. (MB = Mouse Button)
  - Pan: [Ctrl]+[Alt]+MB2.
  - Zoom: [Ctrl]+[Alt]+MB3.
- You can reconfigure these combinations to mimic the view manipulation command used by other common CAD applications- Main menu: Tools > Options

Selecting entities from main window

- Add to selection: [Shift] + MB1
- Remove from selection: [Ctrl] + MB1

Example: Selecting faces from model in Main window



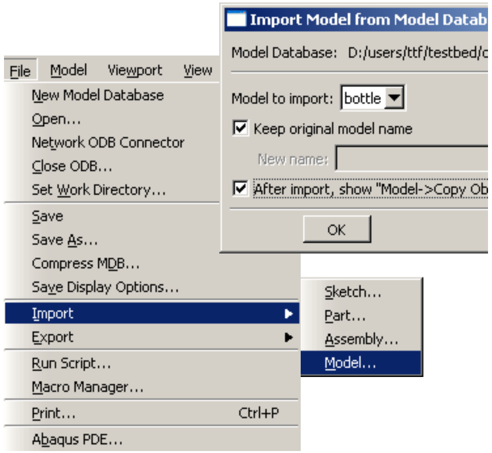
Basic use of Abaqus/CAE

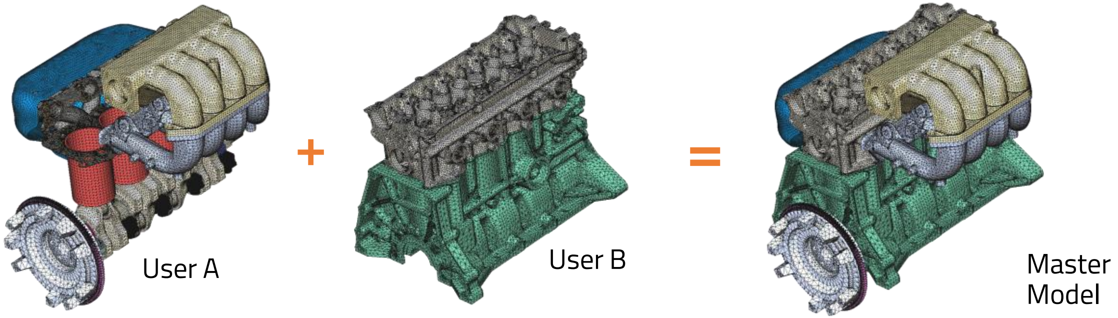
Models can be imported into one database from another

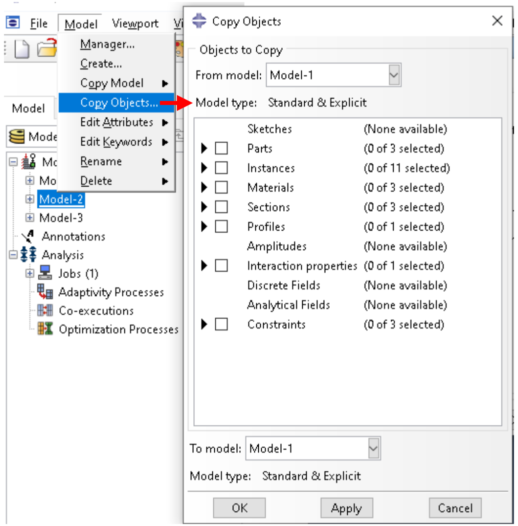
Model data from the imported database is copied into the current database.

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 structural analysis types

Static stress/displacement analysis:

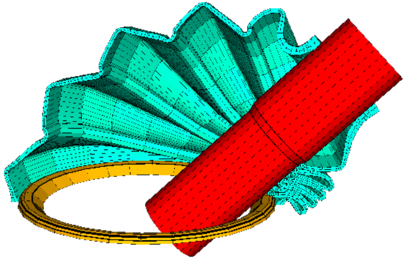
- Linear and nonlinear analysis
- Rate-dependent or rate-independent response
- Eigenvalue buckling load prediction

Linear dynamics:

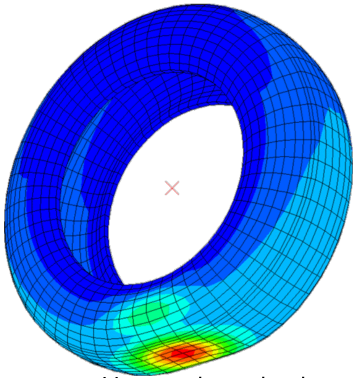
- Natural frequency extraction
- Modal superposition
- Harmonic loading
- Response spectrum analysis
- Random loading

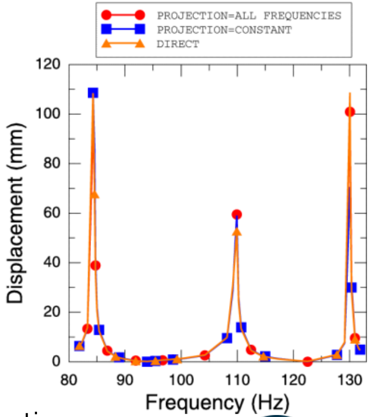
Linear/Nonlinear dynamics:

- Transient dynamics, Implicit time integration



Articulation of an automotive boot seal





Harmonic excitation of a tire

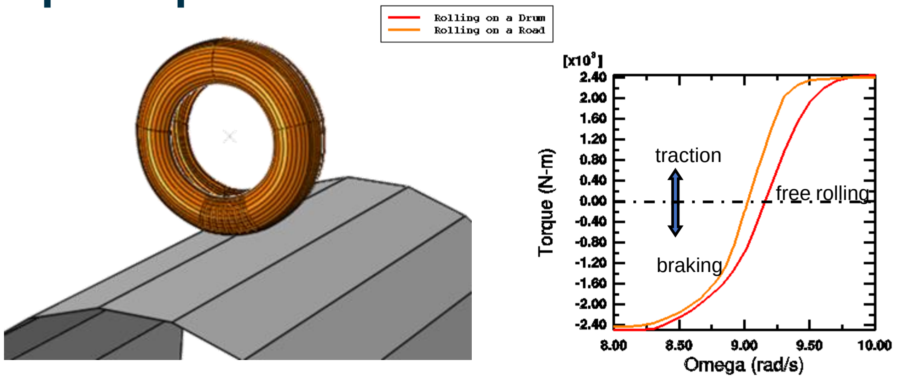
Overview of Abaqus/Standard and Abaqus/Explicit

Other analysis types available in Abaqus/Standard:

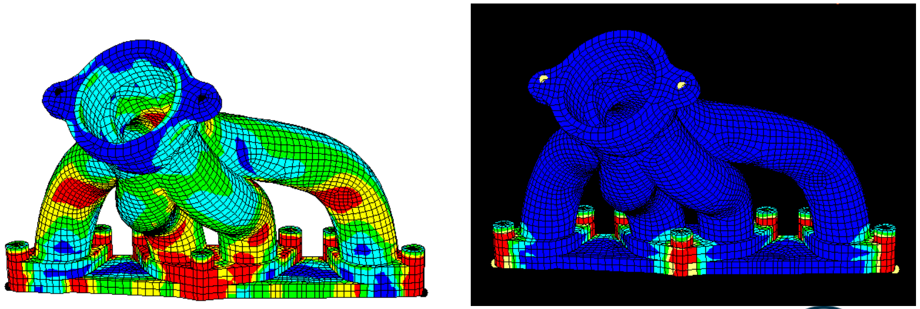
- Heat transfer
- Acoustics
- Mass diffusion
- Steady-state transport

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

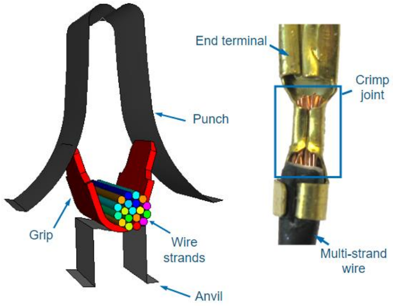
Abaqus/Explicit

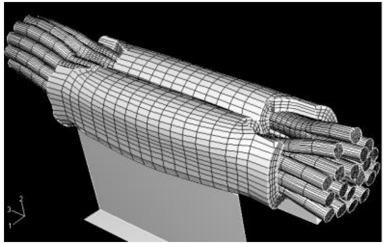
Simulates high speed dynamic events such as drop tests.

- Explicit algorithm for updating the mechanical response.

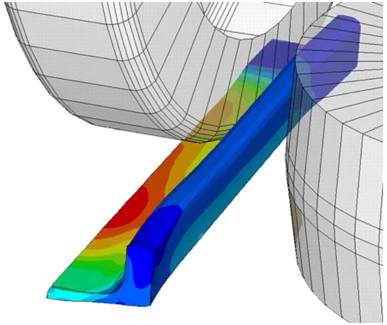
Also, a powerful tool for highly nonlinear quasi-static analyses

- Material plasticity & damage, multiple compilated contacts, large deformations,...
- Annealing is available for multistep forming simulations

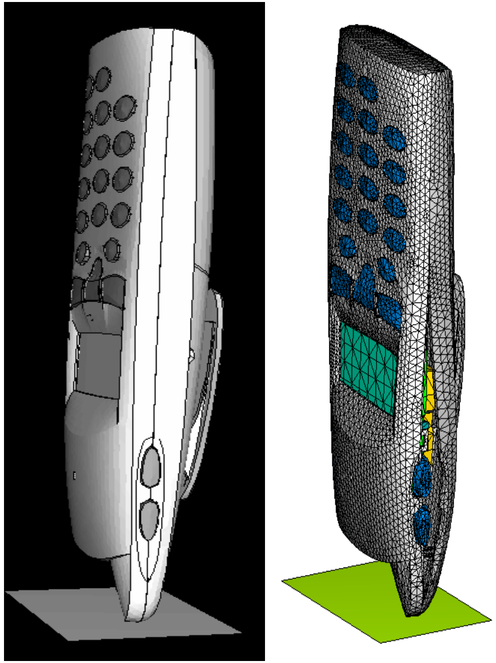




Crimping of terminal



Rolling of a symmetric I-section

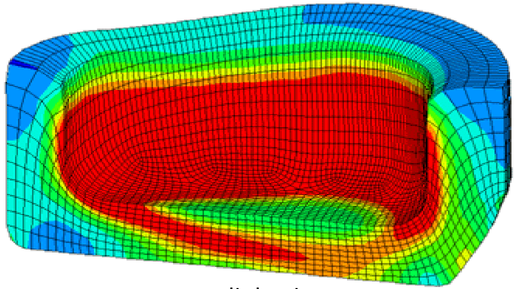


Drop test of a cell phone

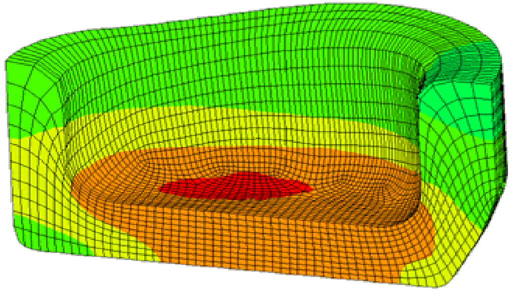
Overview of Abaqus/Standard and Abaqus/Explicit

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



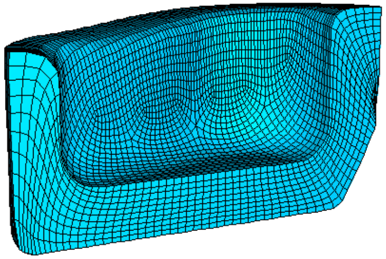
fully coupled temperature-displacement

Two-stage forging, using ALE—contours of temperature

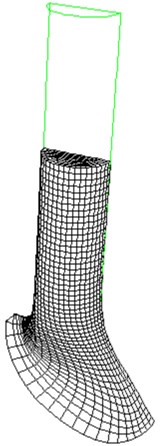
Overview of Abaqus/Standard and Abaqus/Explicit

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.



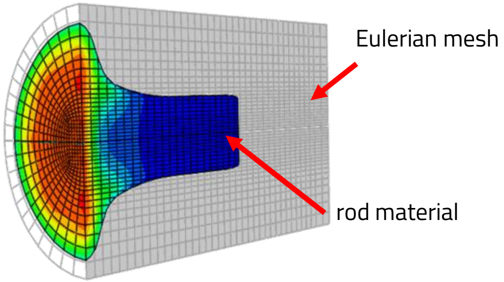
Bulk metal forming



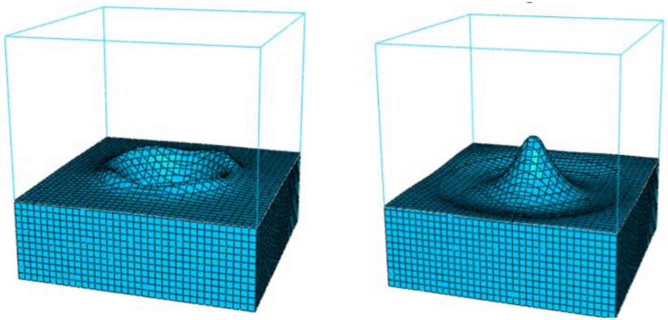
High speed impact

Special features of Abaqus/Explicit: Coupled Eulerian-Lagrangian (CEL)

- Define a domain in which material can flow for a Eulerian analysis
- Flow problems
- Structural problems with extreme deformation



High speed impact of copper rod



Fluid drop

Starting the Abaqus solvers

Option 1: Submitting the analysis from Abaqus/CAE

- From Job manager
- Right mouse button on job in model tree

Option 2: Submitting the analysis from the command line

- Create an input file
    - Write input file from Abaqus/CAE and/or edit input file in text editor
- Start a command prompt in the working directory containing the input file
- Issue the start command on the command line, examples below:

abaqus job=*input_file_name* cpus=8 Uses most recently installed Abaqus release, analysis is run using 8 CPU cores

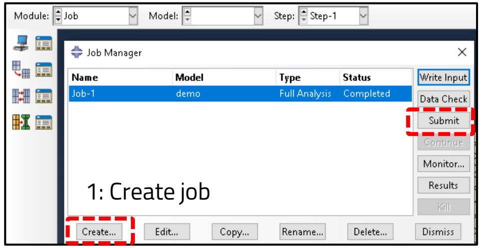
abq2023hf4 job=*jobname* input=*input_file_name* cpus=8 Uses Abaqus release 2023HF4, files are named *jobname,* input is read from *input_file_name,* uses 8 CPU cores

- Commonly used command line parameters

datacheck goes through the input file and writes odb and all data needed by the solver. Does not start the actual calculations. Useful for checking the model

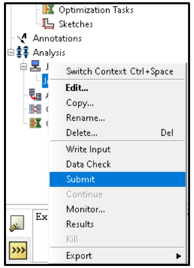
syntaxcheck goes through the input file and writes the odb file. Useful for checking the model. **NOTE: does not require a license**

gpus=*n* number of graphics cards to use in parallel computing. Note: GPU calculation is not supported for all analysis procedures.



2: Submit job

Job manager



Model tree

Abaqus conventions

WE MAKE
YOU INNOVATE

Abaqus conventions

WE MAKE
YOU INNOVATE

Abaqus Conventions

Abaqus uses no inherent set of units.

User only 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

SI (mm, ms): gram, mm, ms ⇒ Force: N, Stress: MPa

Common unit systems and units shown in table below

Most common, and generally recommended

Used in very short duration events like drop tests, shock tests, blast loads

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

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

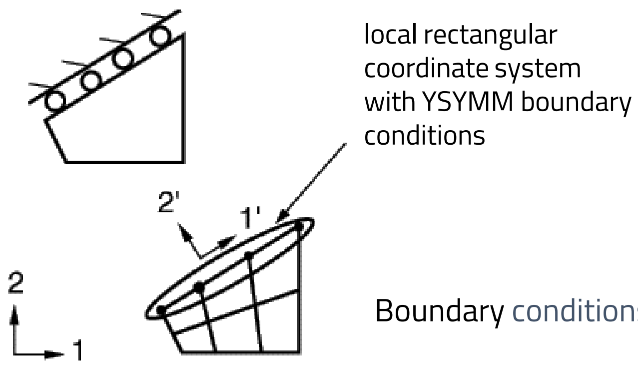
Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2 /in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	tonne/mm^3	slug/ft^3	lbf s^2 /in^4

Common systems of consistent units

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.



Boundary conditions on a skew edge

Abaqus Conventions

Degrees of freedom; DOFs

- Primary solution variable 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.

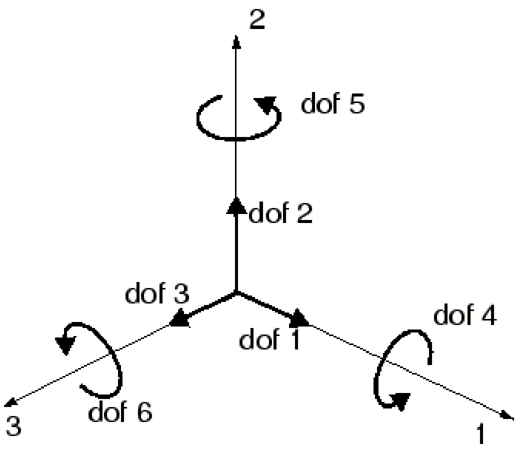
2 Translation in the 2-direction.

3 Translation in the 3-direction.

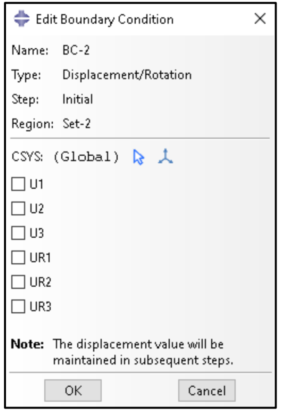
4 Rotation about the 1-direction.

5 Rotation about the 2-direction.

6 Rotation about the 3-direction.



Numbering of displacement DOFs



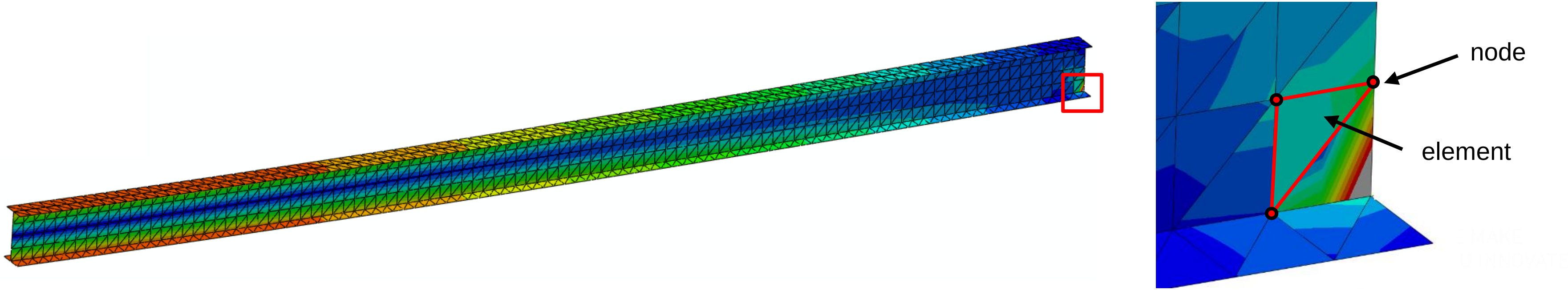
Numbering is used, for example when defining boundary conditions

Elements

WE MAKE
YOU INNOVATE

Finite Element Method basics

- The studied domain (1D:line, 2D:surface, 3D:volume) is discretized into smaller regions; **elements**
- The unknown quantities at the **nodes** are the **Degrees of Freedom (DOFs)**
  - In structural analyses the displacements and in some elments also the rotations
- The displacements at any point within the elements are obtained using the nodal values interpolated using **shape functions**
  - Typically polynomials
  - Other quantities like stress and strain are obtained from the displacements
- Prescribed values for selected DOFs ance be given; **Boundary conditions**
  - At a support the displacements can be set to zero, for example
- In addtion to boundary conditions, **loads** can be applied; force, pressure,...
- A system of equations is formed, from which the values of the unknown DOFs are calculated



Structural elements (shells and beams) and solid elements

Geometric representation of a beam element is a line

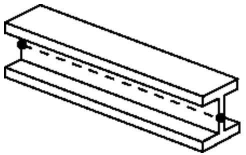
- Material, Profile cross section, and orientation information is given as parameters
- Cannot describe stress in the thickness or width direction of the beam,
- The beam cross-section does not deform

Geometric representation of a shell element is a surface

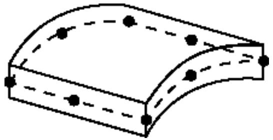
- Material and thickness are given as a parameters
- Cannot describe stress in the thickness direction
- Generally, no deformation in the thickness direction
  - Assumptions on the thinning caused by significant plastic strain in forming simulations can be made.

Geometric representation of a solid element is a solid cell

- Material is given as parameters
- Can describe the full 3D stress and deformation state



Beam element

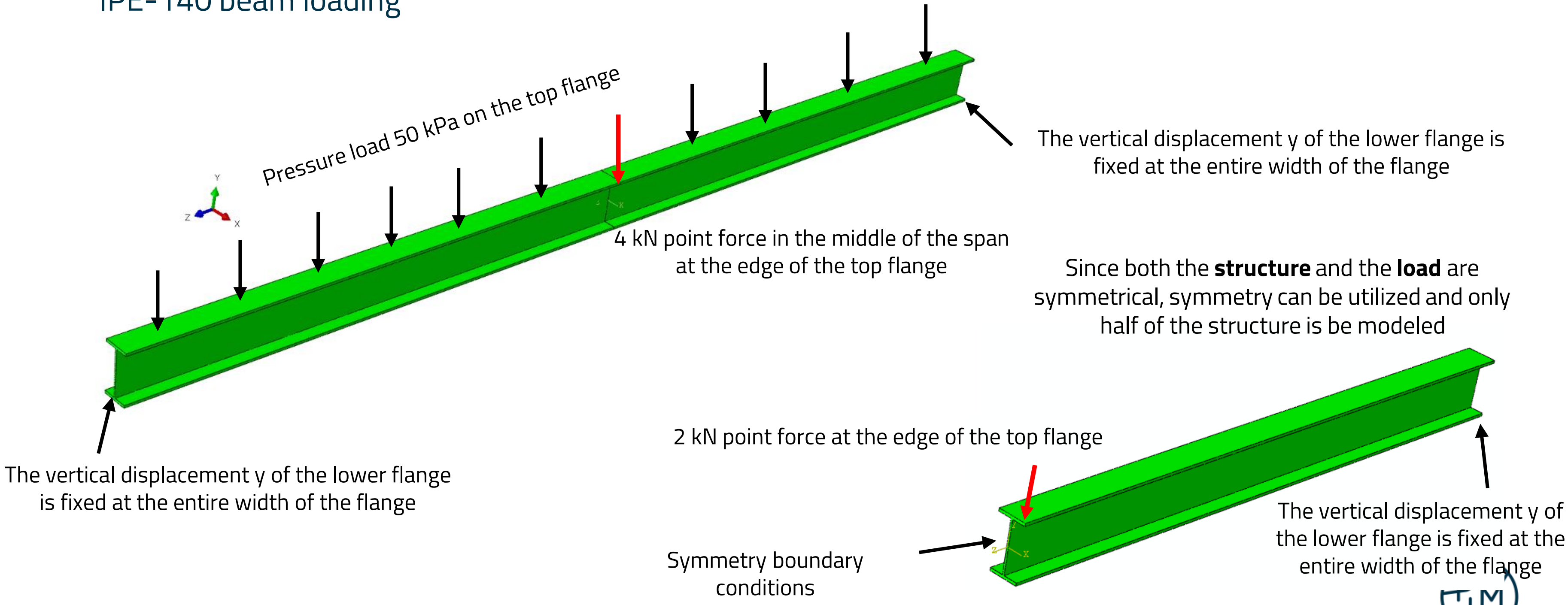


Shell element

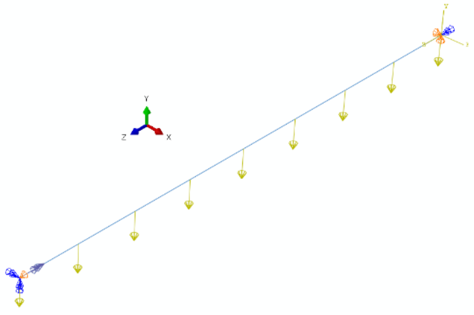
Solid element

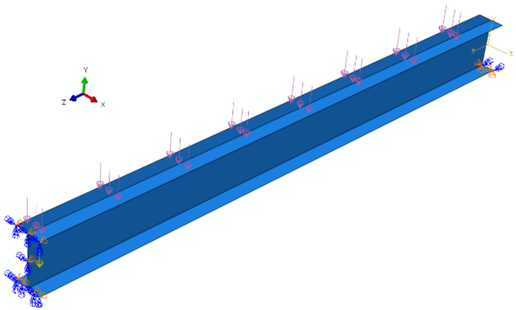
Example: Beam, shell and solid elements comparison

IPE-140 beam loading

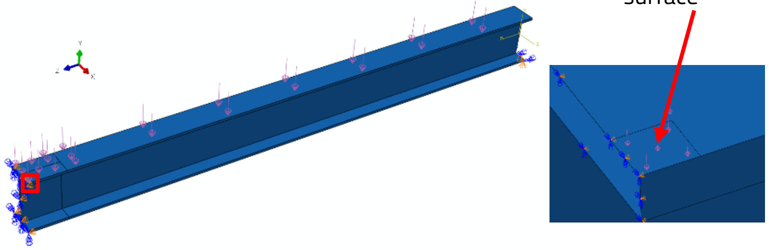


Example: Beam, shell and solid elements comparison





For the load, an additional region is defined on the solid surface



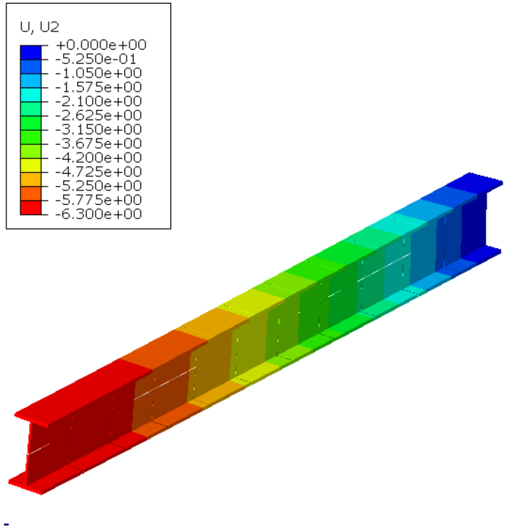
	Beam	Shell	Solid
Geometry	line	surface	volume
Cross-section parameters	Profile dimensions, direction, material	thicknesses, material	material
Pressure load	As line load on the beam elements; pressure×flange width [F/L]	As pressure [F/L^2] on top flange	As pressure [F/L^2] on top flange
Point force	Point force and point moment at the end node	Point force at the top flange edge node	As pressure at the additional region at the flange edge. Pressure resultant equals point force.

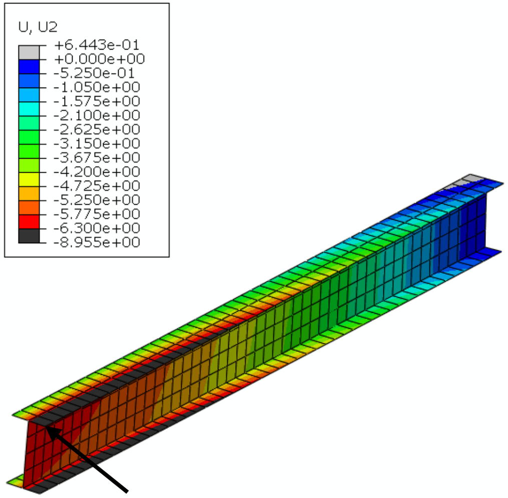
Example: Beam, shell and solid elements comparison

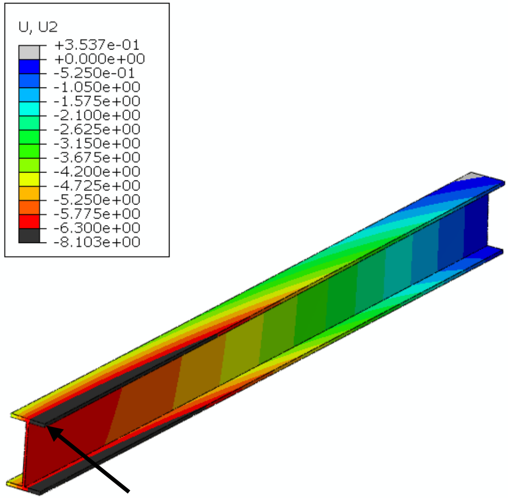
Vertical displacement is practically the same for all models

In the beam model, only the center line displacement and rotation are obtained. The vertical displacement of the flange edge should be calculated separately

In the shell and solid model, the displacement caused by the torsion of the profile is seen directly as translation





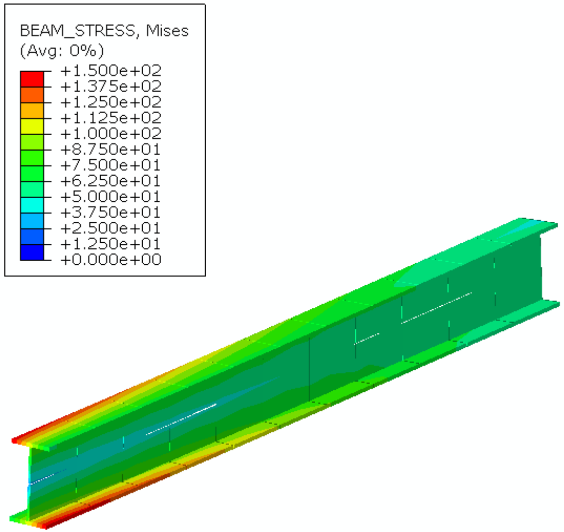


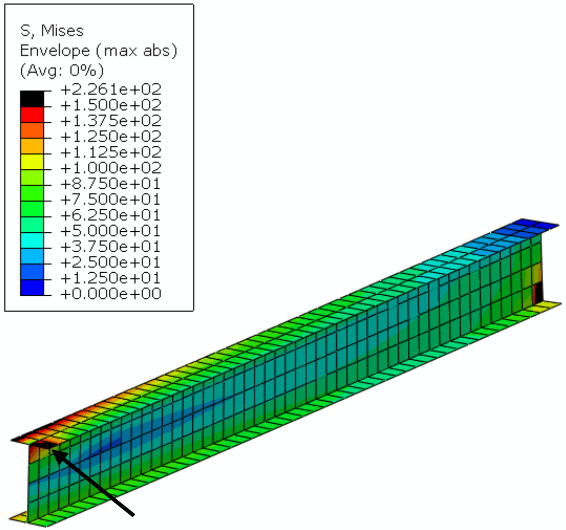
Example: Beam, shell and solid elements comparison

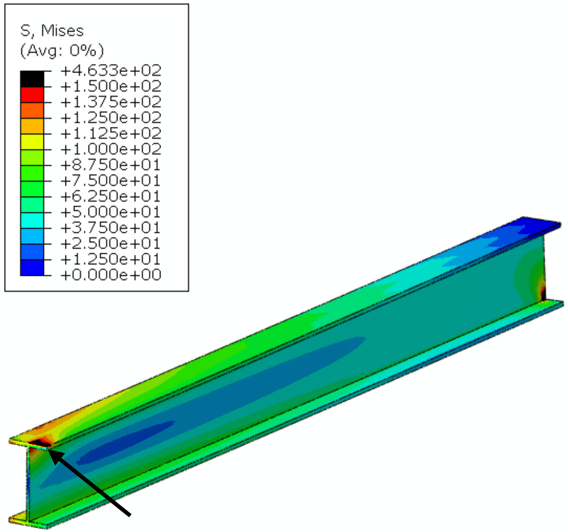
Global stresses are practically the same in all models

The beam model is unable to capture the stress concentration caused by the point force at the flange edge.

In the shell and solid models, the stress concentration caused by the point force is apparent.





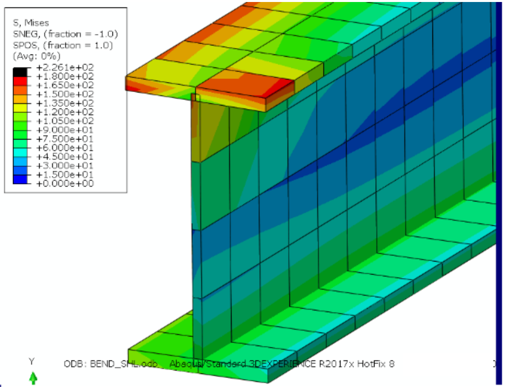


Example: Beam, shell and solid elements comparison

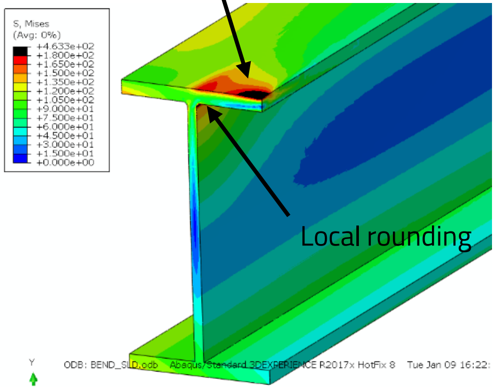
Differences in stress concentration in the shell and solid models

There is a 3D stress state near the force acting on the flange, which can only be captured using a solid model

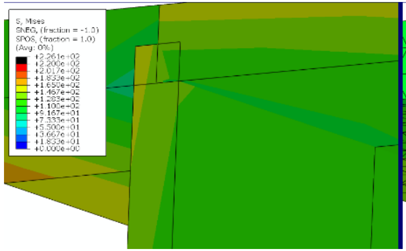
Local rounding between flange and web



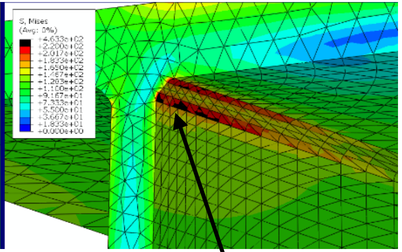
Shell



Solid



Shell



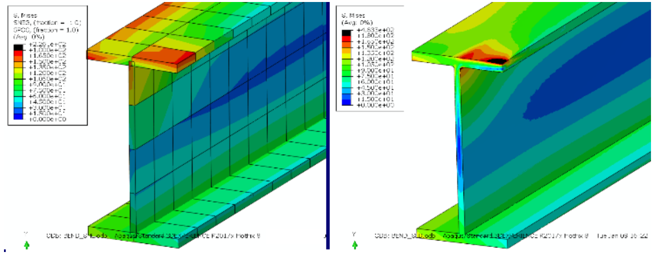
Solid

Local details can only be analyzed using a solid model

Example: Beam, shell and solid elements comparison

Required computing resources

- Beam model: 1 second with one processor
- Shell model: 2 seconds with one processor
- Solid model: 173 seconds with three processors



Results summary

Global phenomena, e.g., bending and stress for the whole beam

- These are already apparent with a very light beam model

Cross-section deformations

- These cannot be calculated using a beam model (1D model)
- Can be calculated with a shell model with cross-section modeling (2D model)

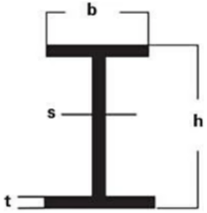
Local phenomena, e.g., stress in rounding or at local load region

- Analysis requires a full 3D model
- Computationally the heaviest by far

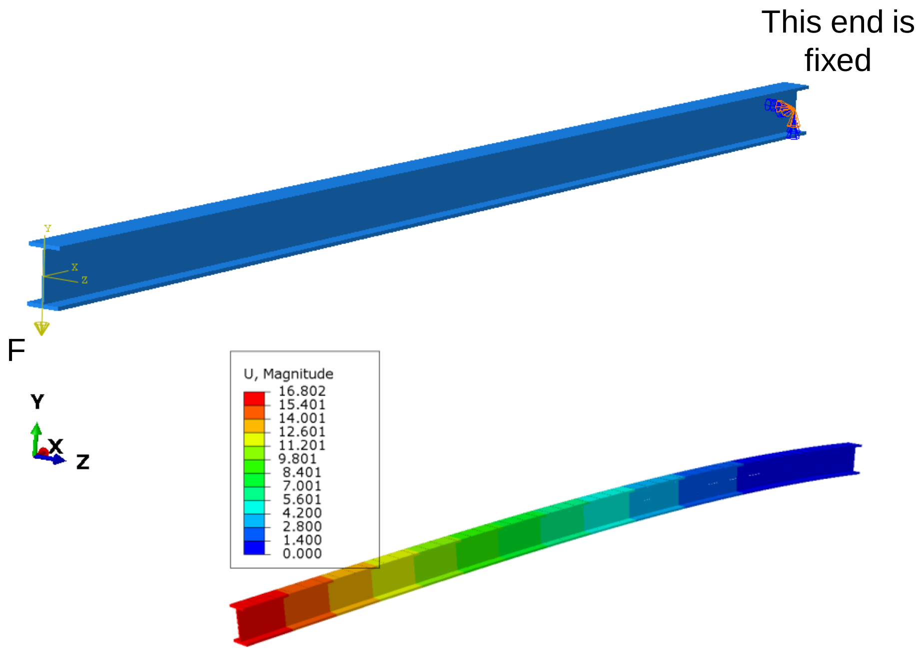
Demonstration of Abaqus/CAE functionality

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



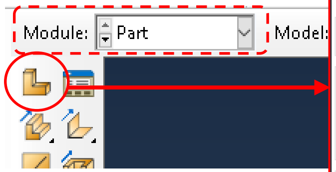
Step 1: Create a part

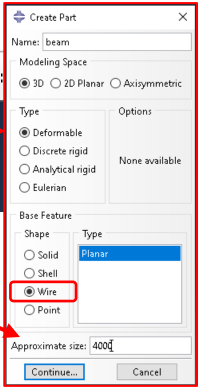
For **beam** elements

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

Part module

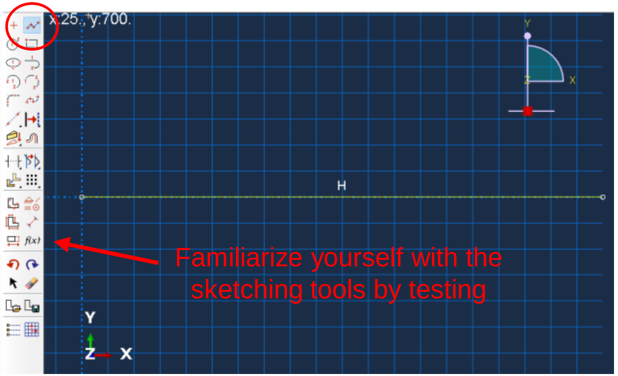
Create part





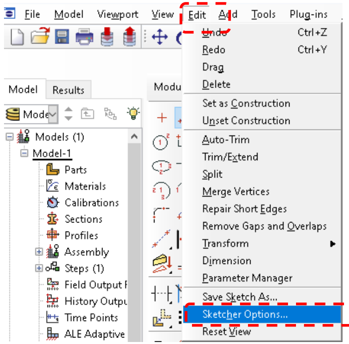
Give approximate size of drawing canvas

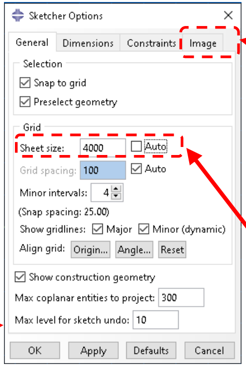
Sketch line



Familiarize yourself with the sketching tools by testing

Sketcher options

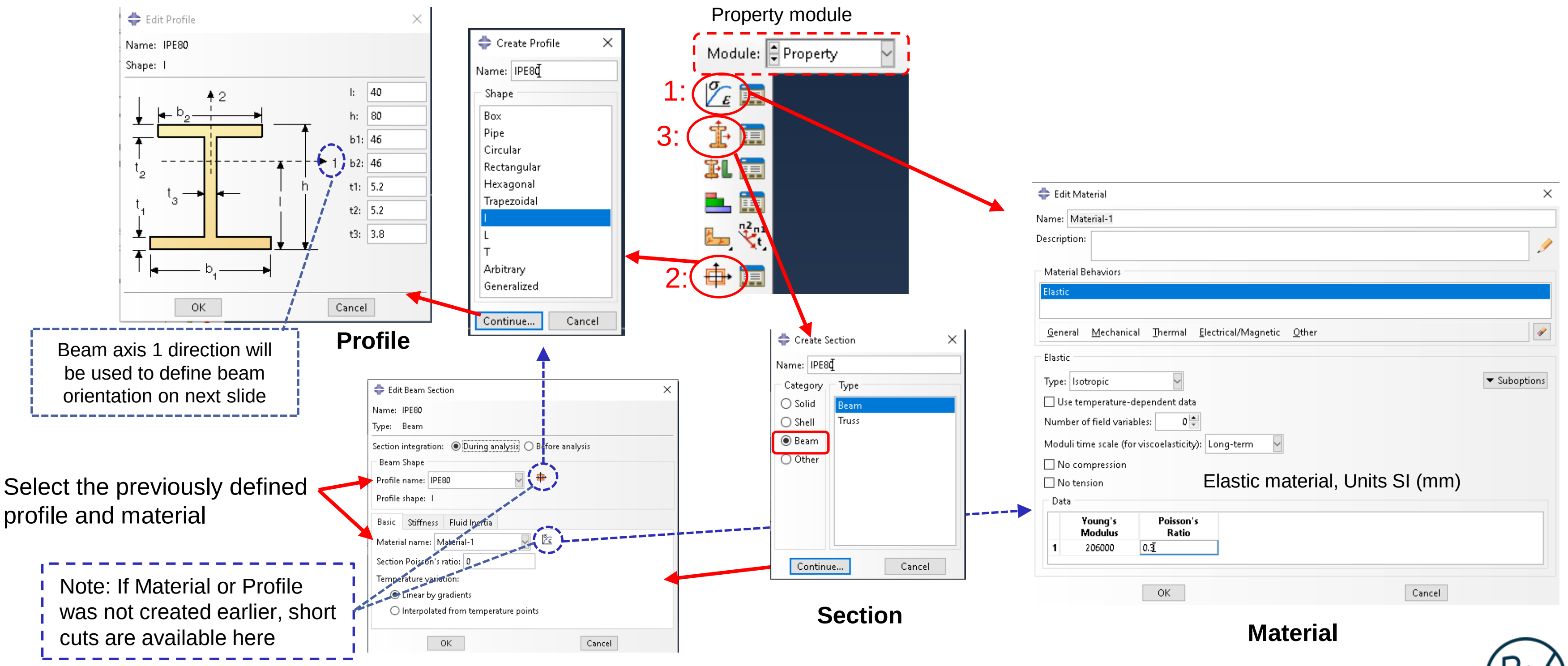




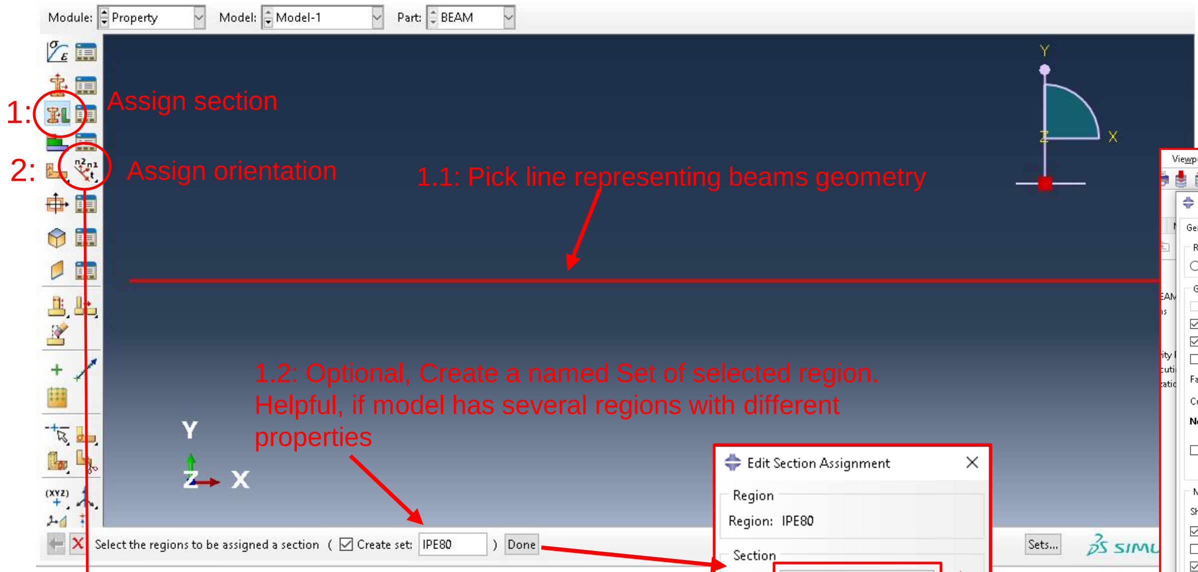
An image can be imported, scaled, and set on the background of the canvas. This can help in modelling complex shapes: take a photo, set is as background image, and draw lines according to the photo

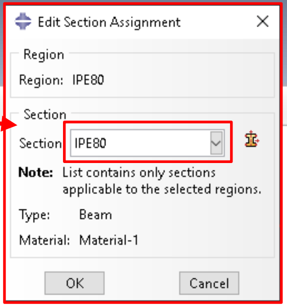
Modify size of drawing canvas

Step 2: Create material, profile, and beam section



Step 3: Assign beam orientation

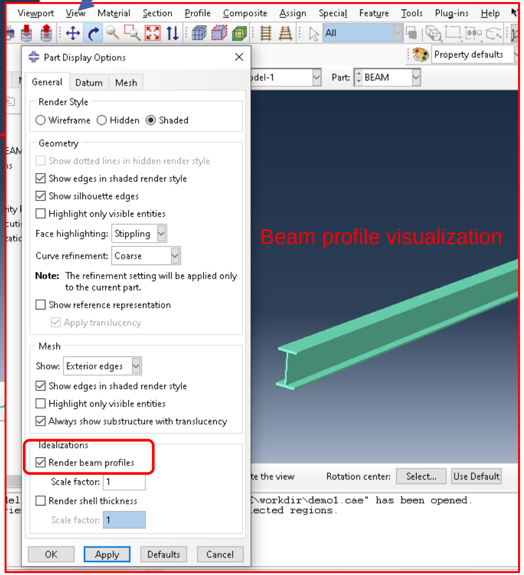




1. Select region (the line representing the beam)
2. Abaqus/CAE shown arrows along beam tangent. Give approximate direction for beam cross section axis 1 (see profile definition on previous slide for axes 1 and 2 directions)

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

Main menu: View > Part Display Options

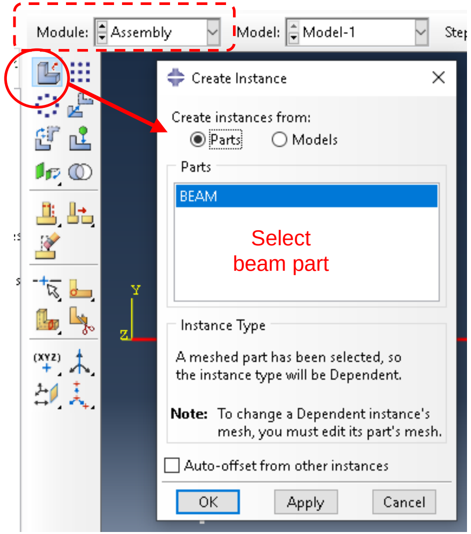


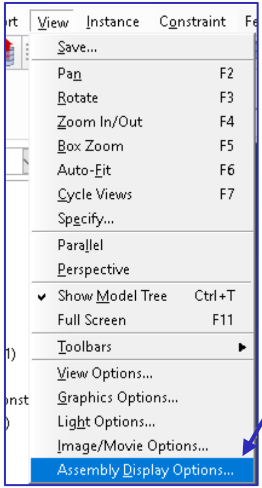
Step 4: Create the Assembly

- 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

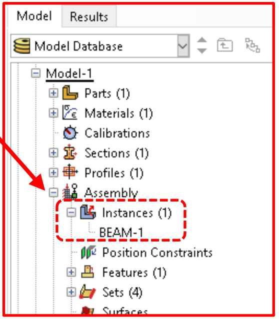
Create Instance





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

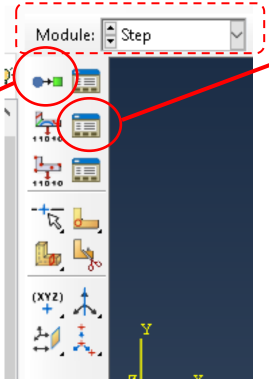


Step 5: Create analysis step

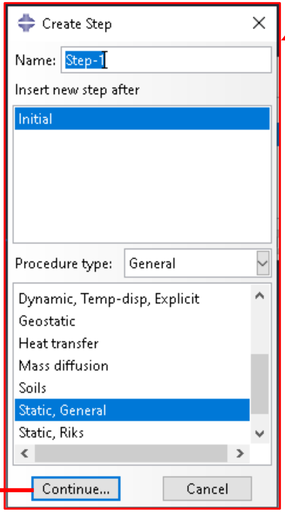
Analysis step defines what is calculated

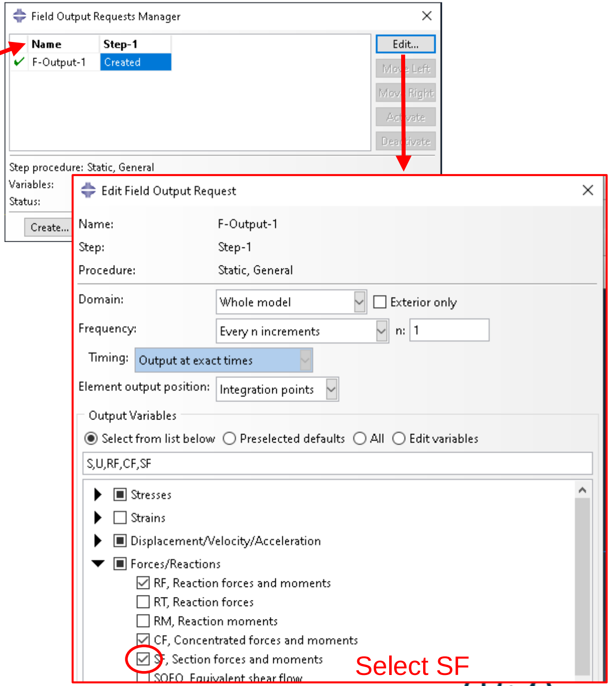
Modify the default output request to add output of beam section forces SF. This is needed to visualize the stress on the beam cross section.

Step module

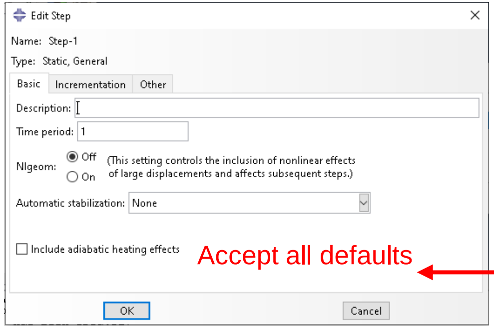


Create Step





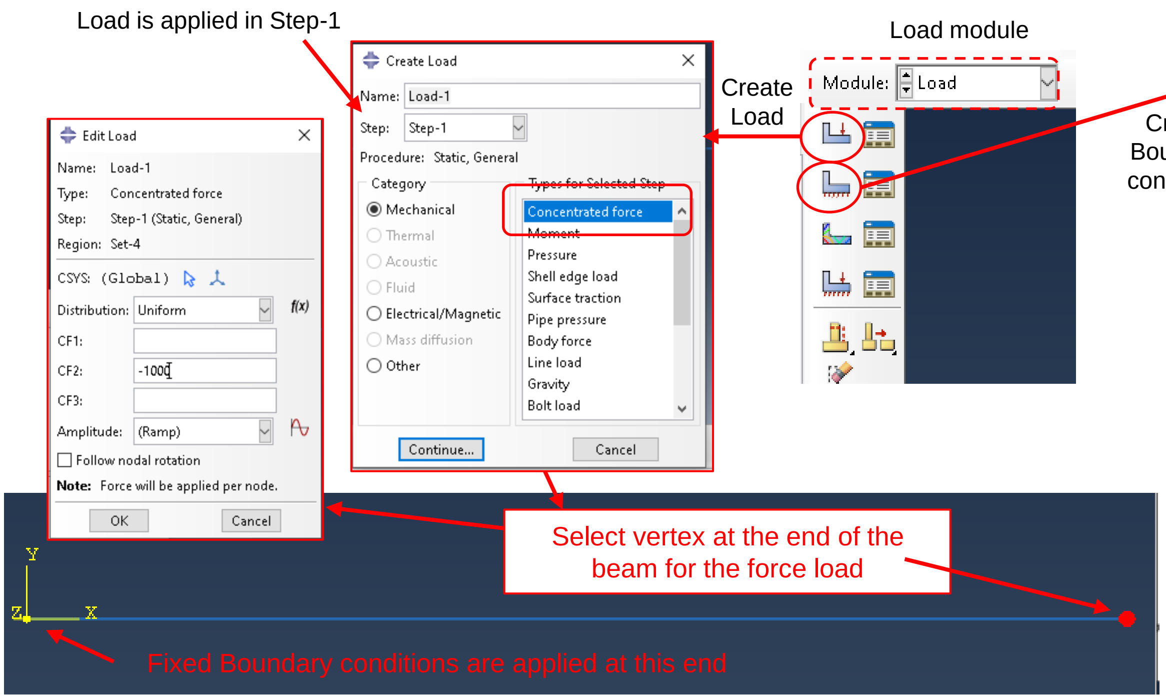
Select SF

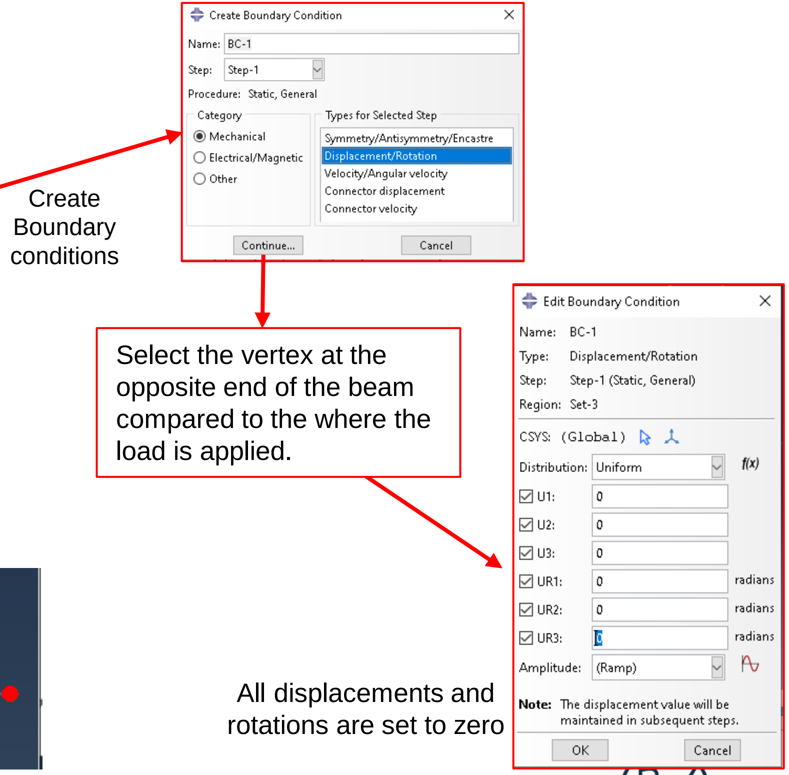


Accept all defaults

Step 6: Define loads and boundary conditions

The analysis step defines what is calculated

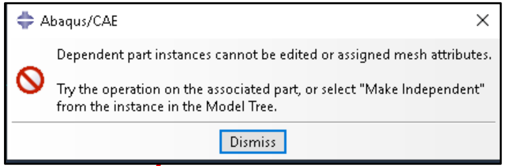




Step 7: Mesh the part

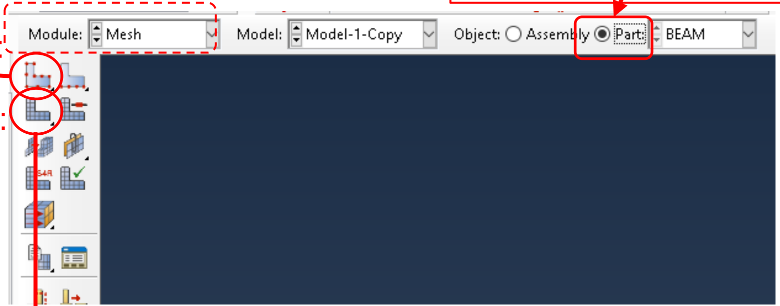
- The geometry is divided into finite elements for the calculation

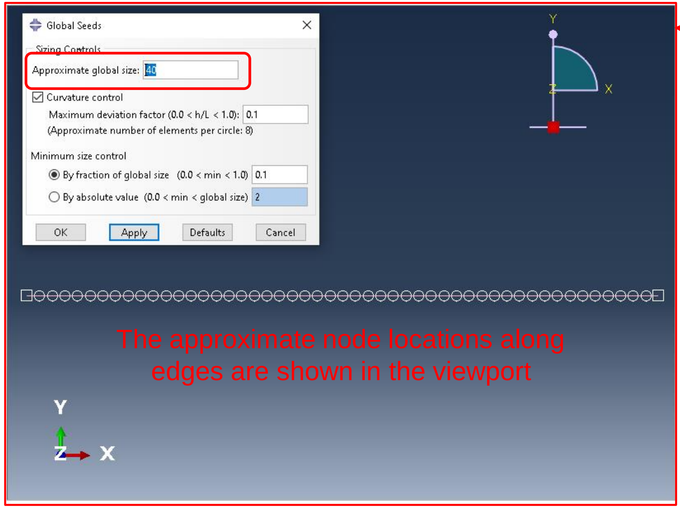
Tip 1: When you see this error, switch to the Part level in the mesh module and try again.



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

Mesh module

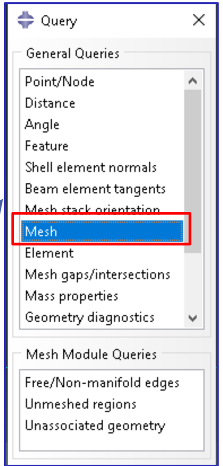




Mesh the model

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

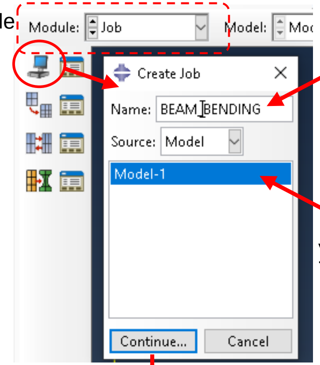
```
Total number of nodes: 51
Total number of elements: 50
      50 linear line elements of type B31
```



Step 8: Create a Job and submit the analysis

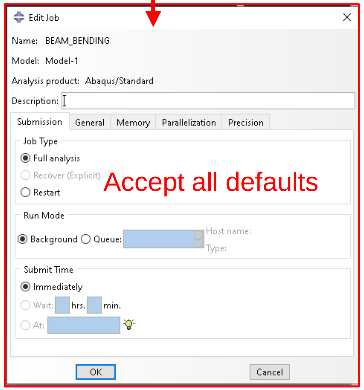
Job module

Create Job



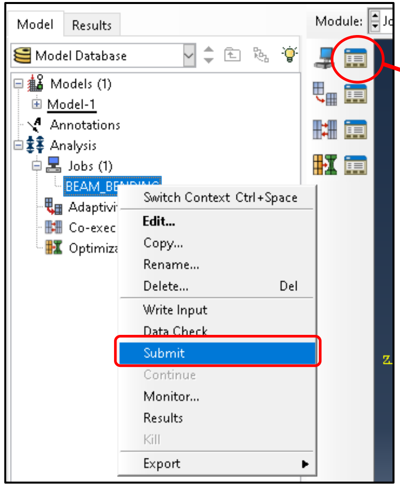
Give a name to the job. All analysis and output files will have this name.

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

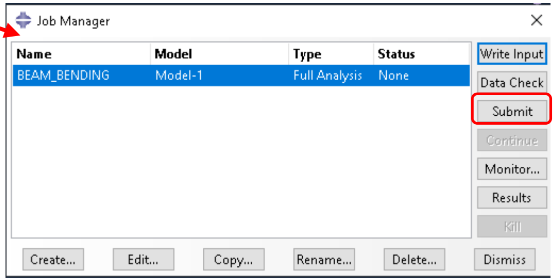


Accept all defaults

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



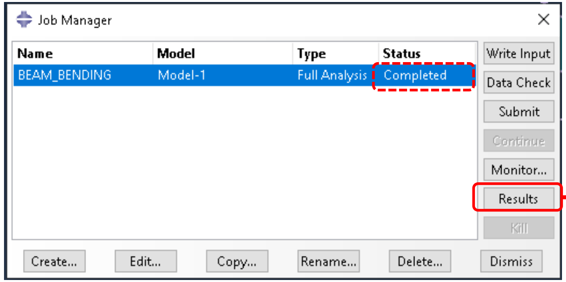
Option 1: Right click the Job in the Model tree and select *Submit*



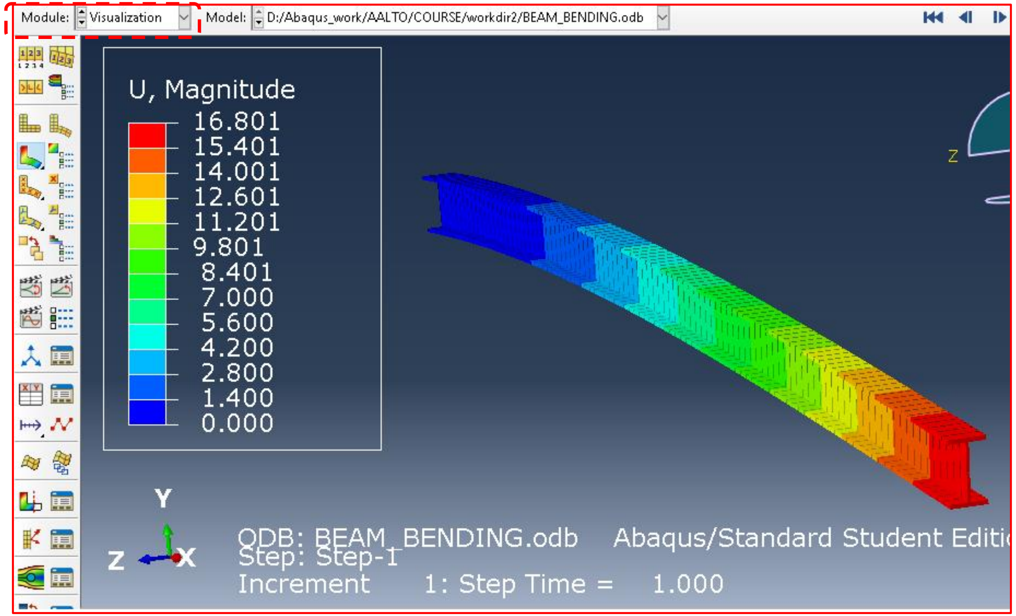
Option 2: Open the Job manager and select *Submit*

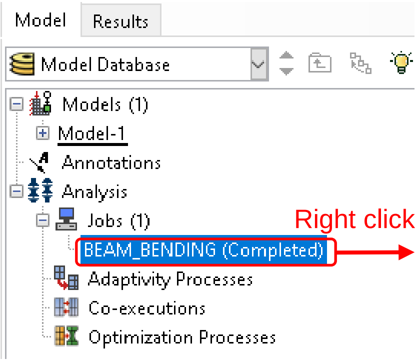
Step 9: Visualize the results

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

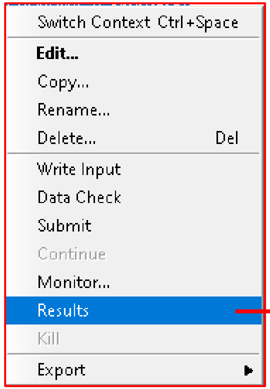


Visualization module



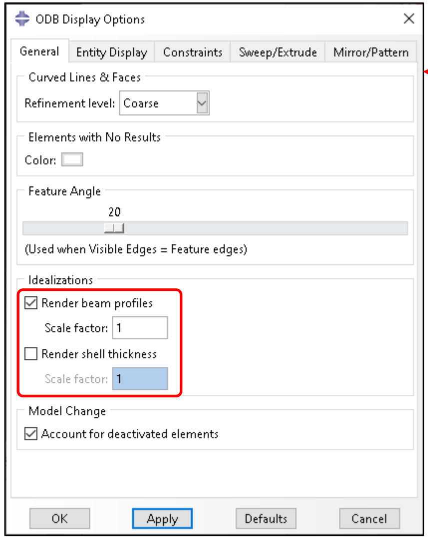


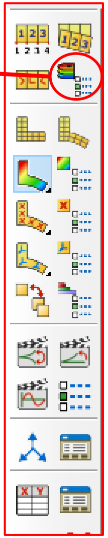
Right click



Tips for result visualization

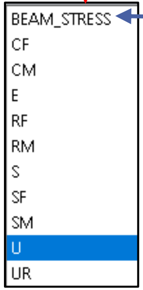
Rendering beam profiles and shell thickness





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

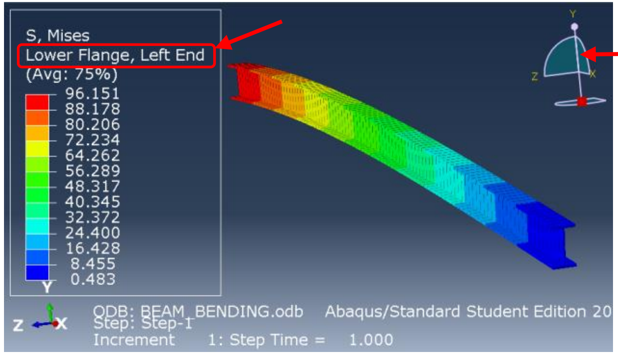
RF/RM Reaction forces (force/moment)

CF/CM: Concentrated loads (force/moment)

SF/SM: Beam section forces (force/moment)

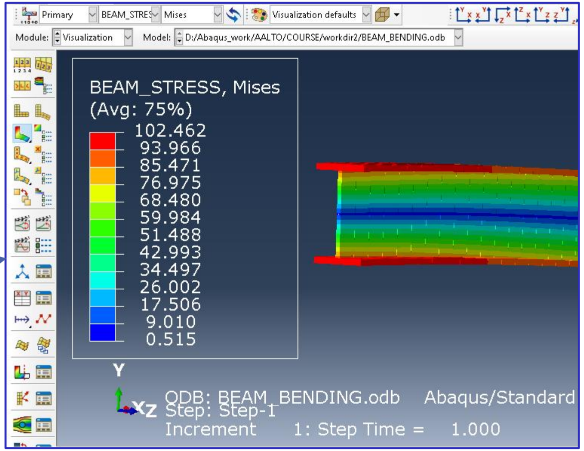
Tips for result visualization

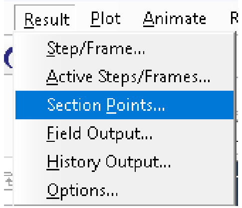
- For solid elements visualization is trivial: Result value at each point is unambiguous
- For shell elements the result value can be from various locations in the thickness of the shell
- For beam elements the result value can be from various locations of the profile cross section



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

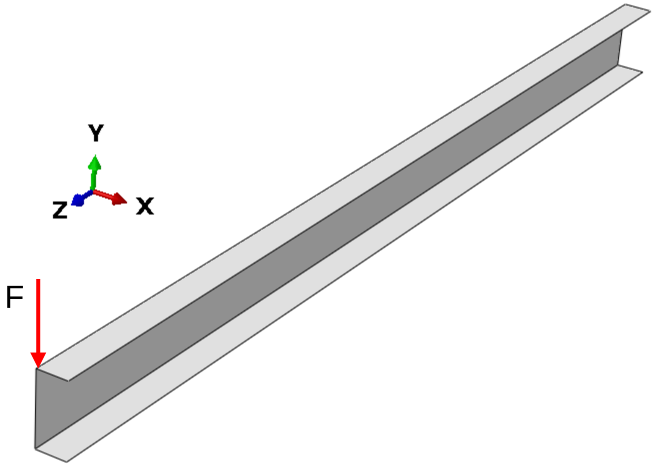
Static analysis of U-profile cantilever beam

U-profile 60x140x6

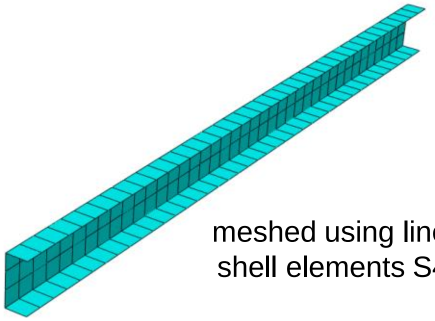
L = 2000 mm

E = 206 GPa, ν = 0.3

F = 1000 N



This end is fixed



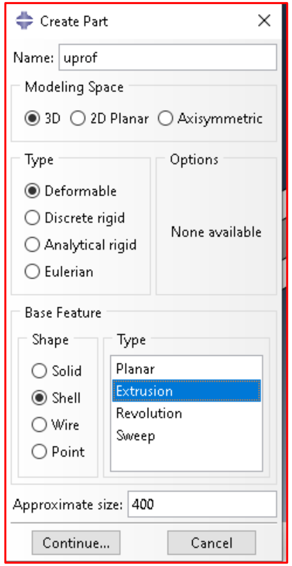
meshed using linear shell elements S4R

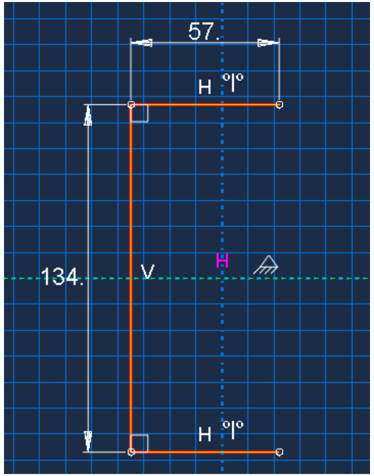
Step 1: Create a shell part

- 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

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

Create part with base feature *Shell, Extrusion*

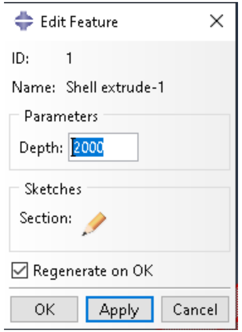


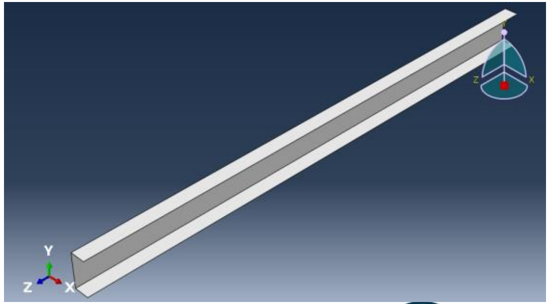


Cross section for extruded shell.

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

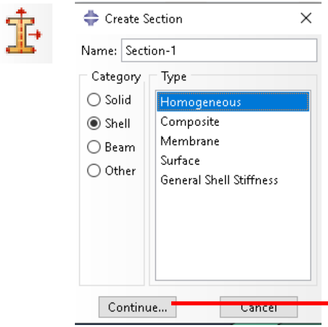
Outer dimensions: 60x140x6

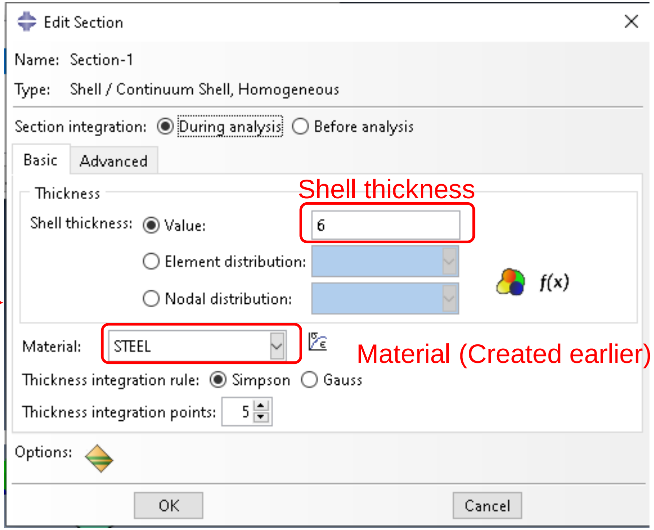


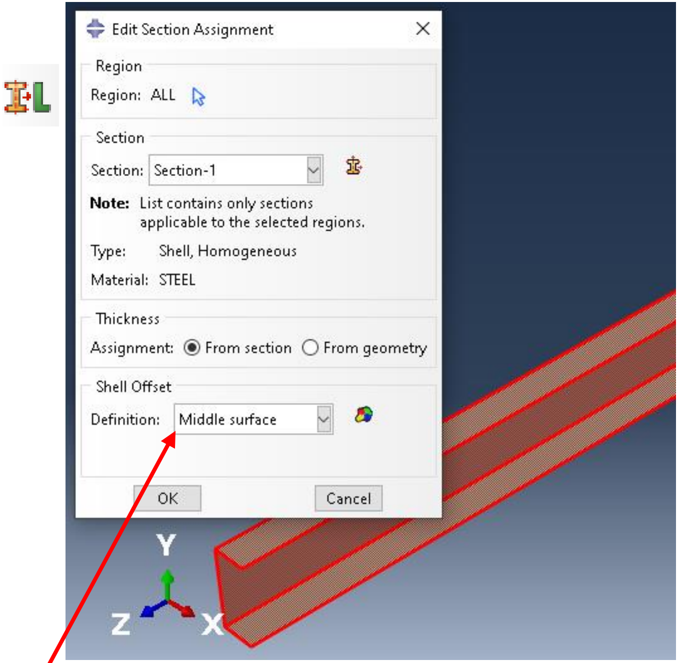


Step 2: Create and assign section

Create a shell section





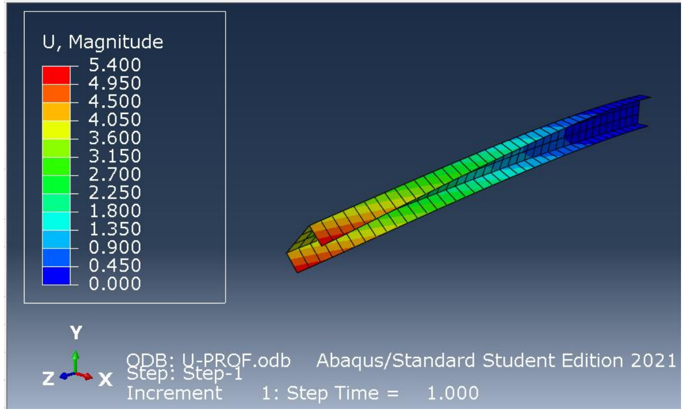


Tip: After Section assignment the shell thickness can be visualized:
Main menu > Part Display Options > Render Shell Thickness

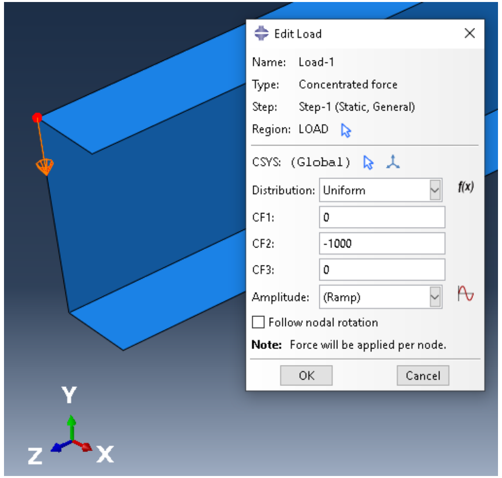
Sketch of cross section was done using the middle surface dimensions, so *Middle surface* is selected. Optionally the thickness can be offset with reference to the modelled geometrical surface

Next steps

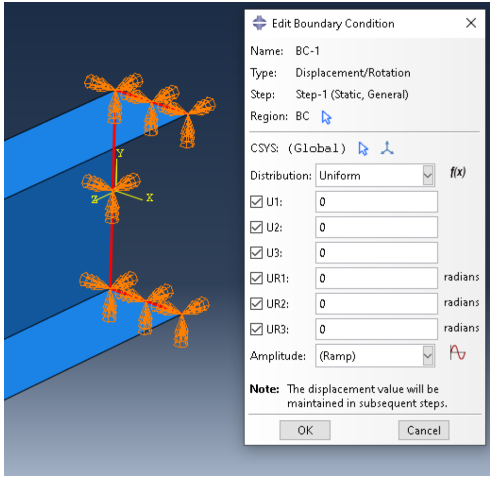
- Similarly, as in the first example:
  - Create the assembly
  - Create an analysis step (Static, General)
  - Apply loads and boundary conditions
  - Mesh the part
  - Create an analysis job
  - Submit the analysis



Concentrated force at one end of the beam



Fixed boundary conditions at the other end of the beam



Agenda day 2

- Geometric nonlinearity
- Material nonlinearity
- Constraints and contacts
- Nonlinear static analysis Workshops
- 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 to $n \cdot F$, the result U will be $n \cdot U$
  - 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 generally 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

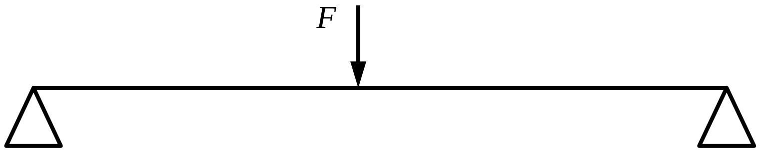
Geometric nonlinearity

When must geometric nonlinearity be considered?

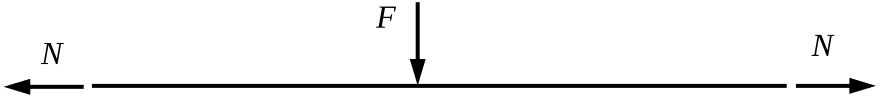
- **A rule of thumb**: If the deformation is not visible by naked eye, there is likely no need to consider geometric nonlinearity.
- Some cases cannot be calculated without considering the geometric nonlinearity.
  - For example, a rope loaded in a transverse direction.

Geometric nonlinearity

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

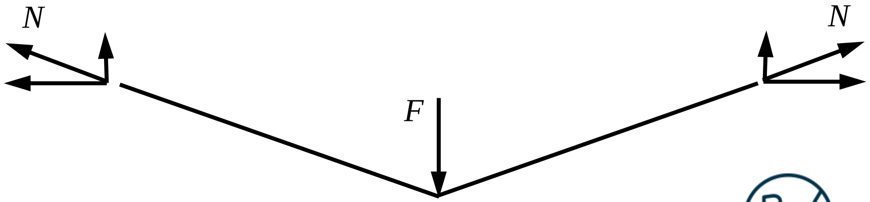


Free body diagram



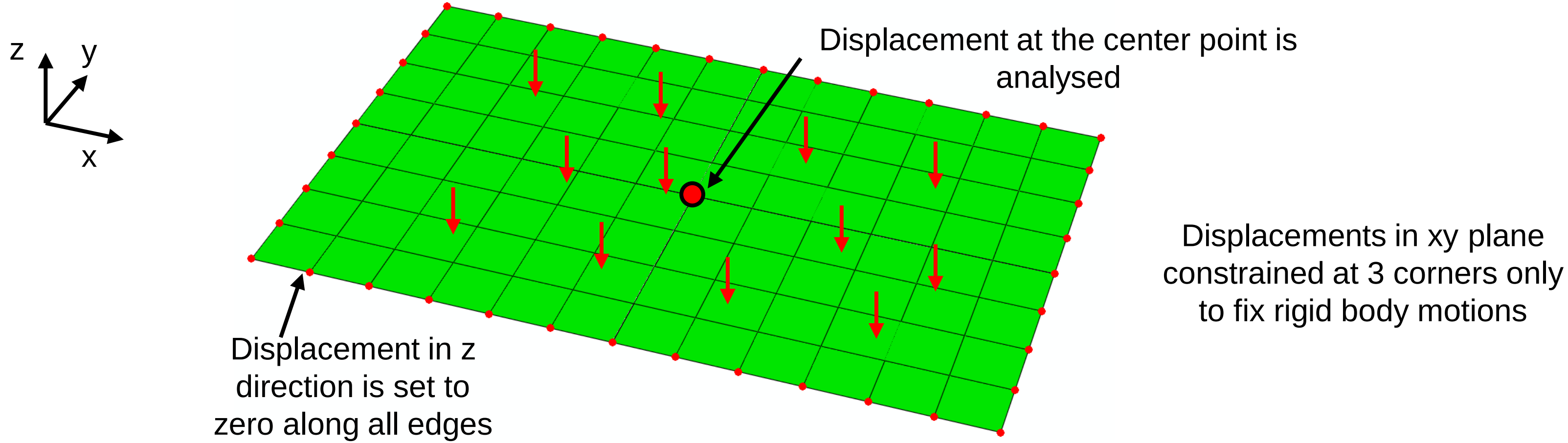
In the initial state, when the rope is straight,
there is no solution for the equilibrium problem.

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

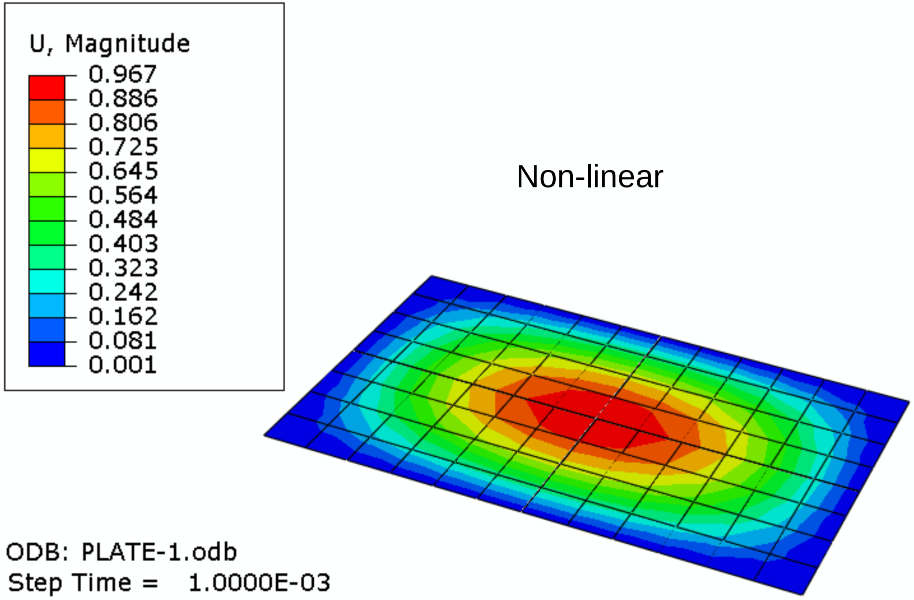


Geometric nonlinearity: Example

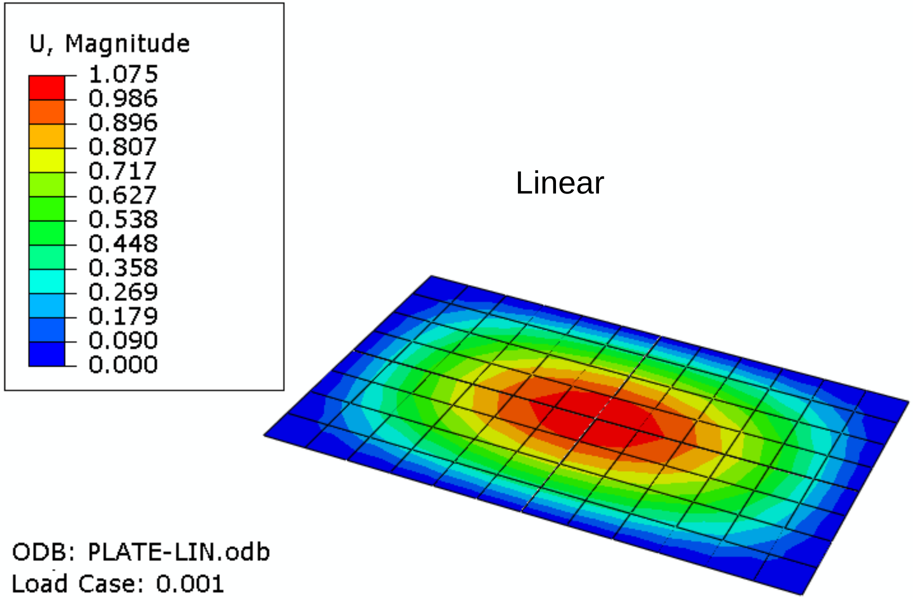
Loading 100 Pa

Displacement [mm]

Non-linear



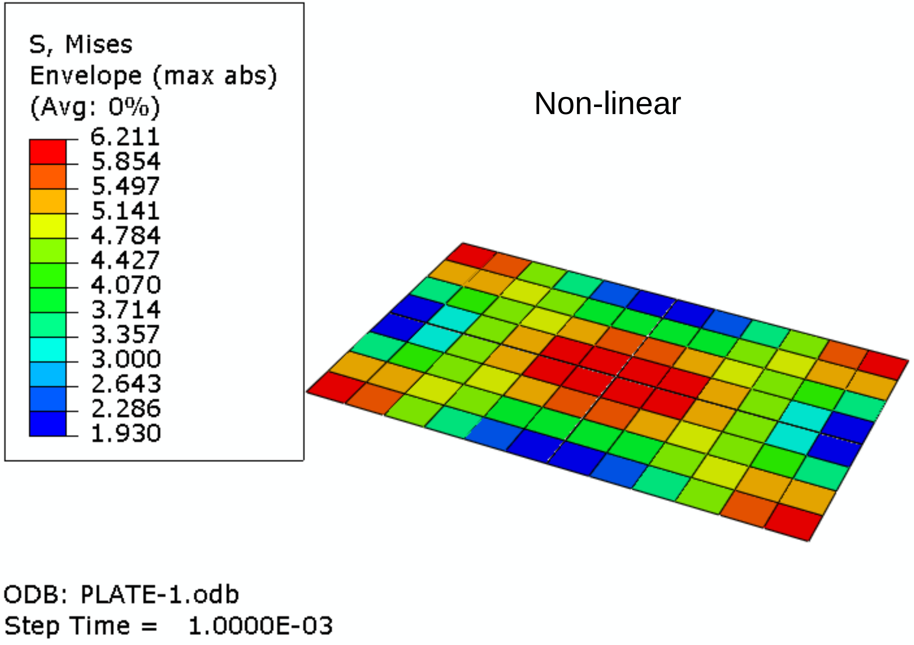
Linear

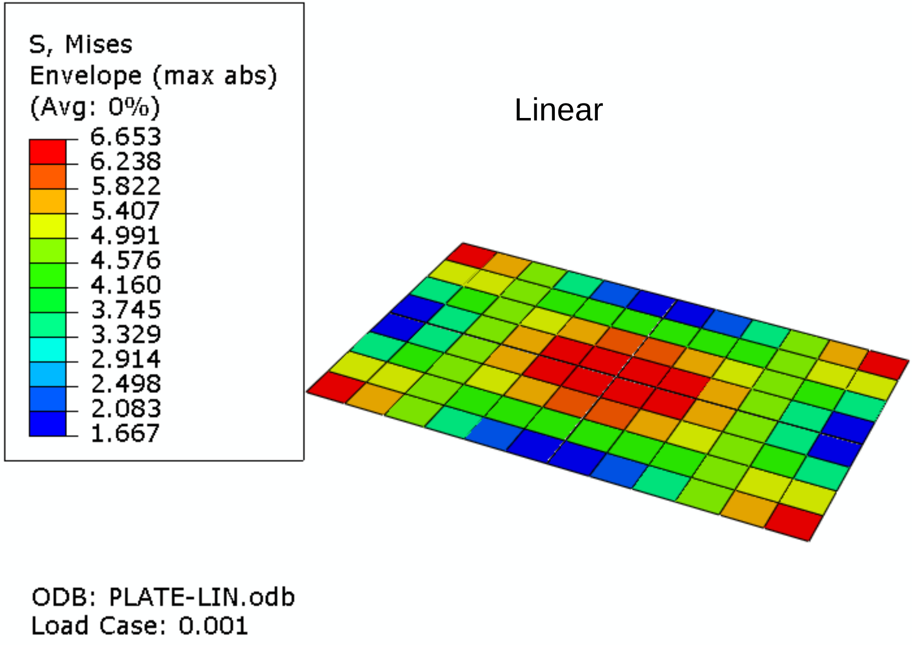


Geometric nonlinearity: Example

Loading 100 Pa

Stress [MPa]

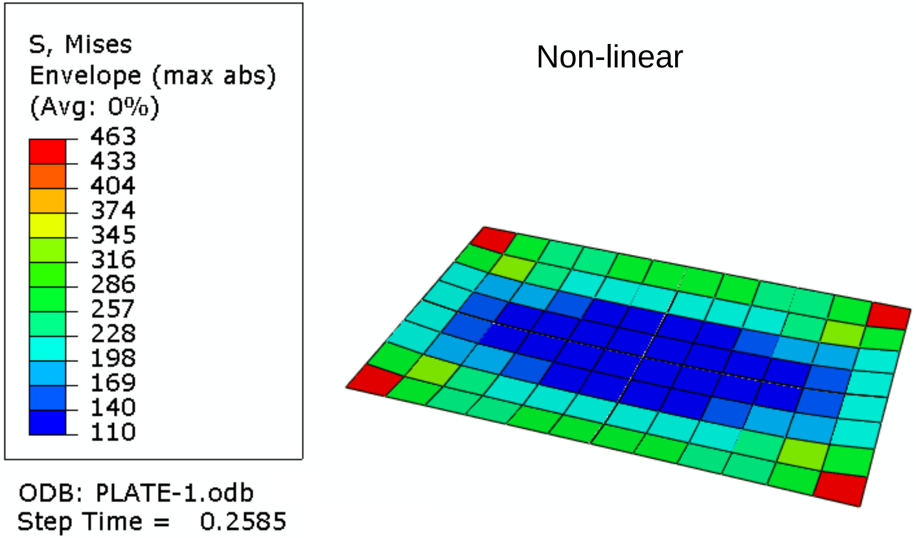


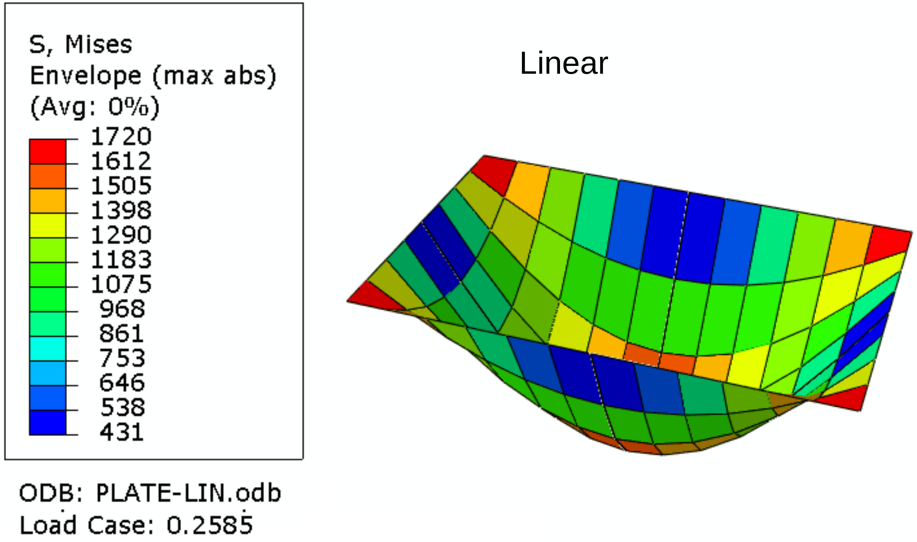


Geometric nonlinearity: Example

Loading 25850 Pa = 0.2585 bar

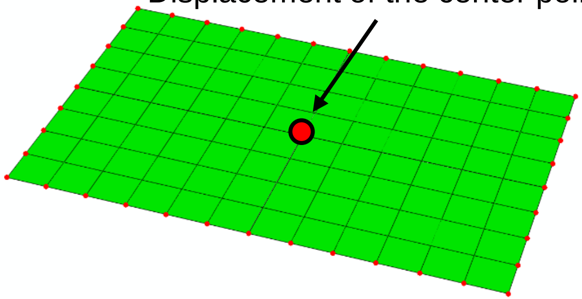
Stress[MPa]

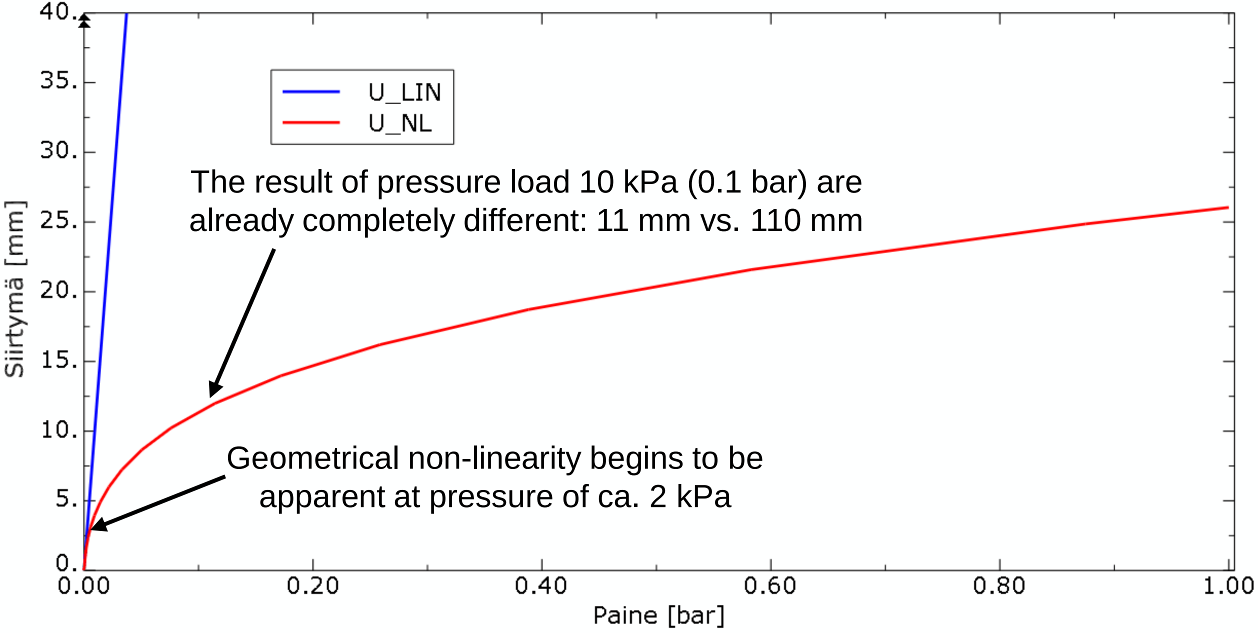




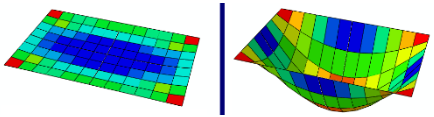
Geometric nonlinearity: Example

Displacement of the center point





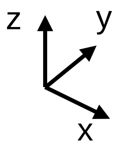
Geometric nonlinearity: Example

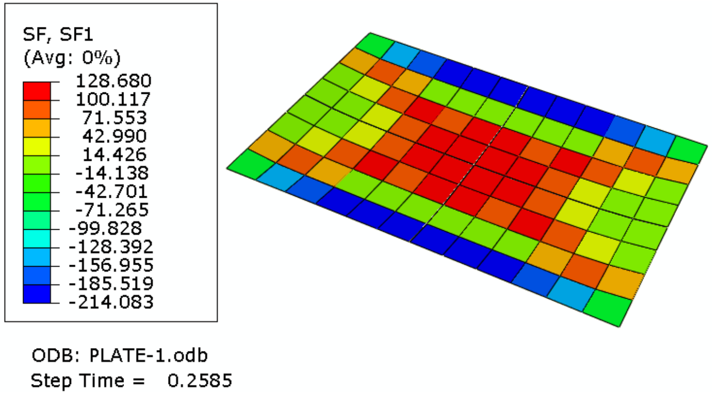


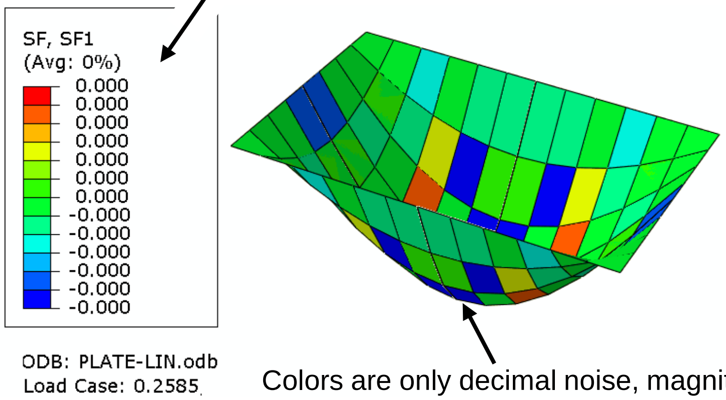
- Question: What causes this big difference?
- Answer: As the displacement increases, the plate begins to carry in the membrane mode.

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 $\sim$ 2 - 100 times the linear analysis
  - If there are other nonlinearities, 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:

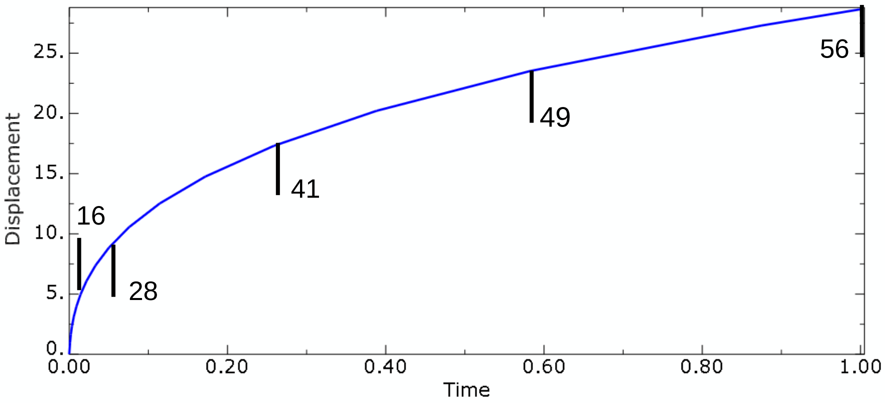
STEP	INC	ATT	SEVERE DISCON ITERS	EQUIL ITERS	TOTAL ITERS	TOTAL TIME/ FREQ	STEP TIME/LPF	INC OF TIME/LPF	DOF MONITOR	IF RIKS
1	1	1	0	4	4	0.00100	0.00100	0.001000		
1	2	1	0	3	3	0.00200	0.00200	0.001000		
1	3	1	0	3	3	0.00350	0.00350	0.001500		
1	4	1	0	3	3	0.00575	0.00575	0.002250		
1	5	1	0	3	3	0.00913	0.00913	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

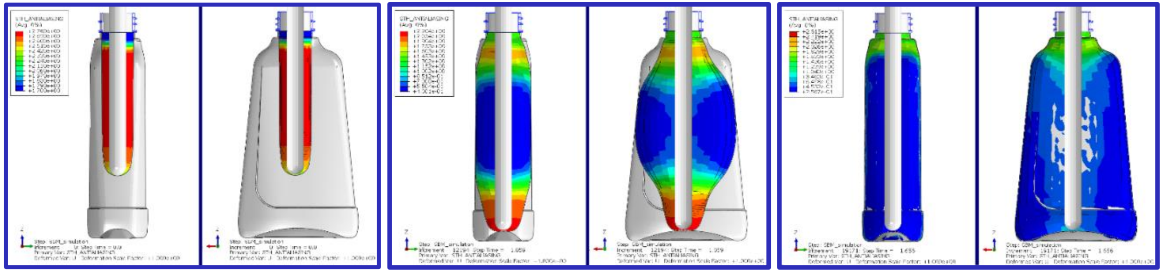
Time instant of the analysis

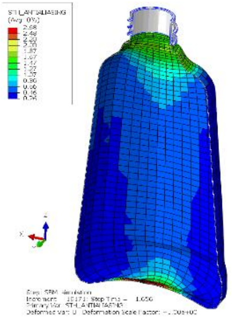
Displacement of the plate center point and required iterations



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%





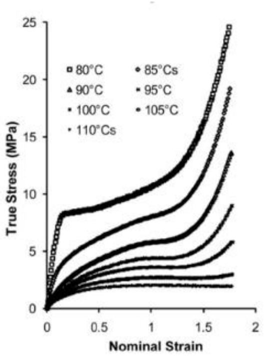
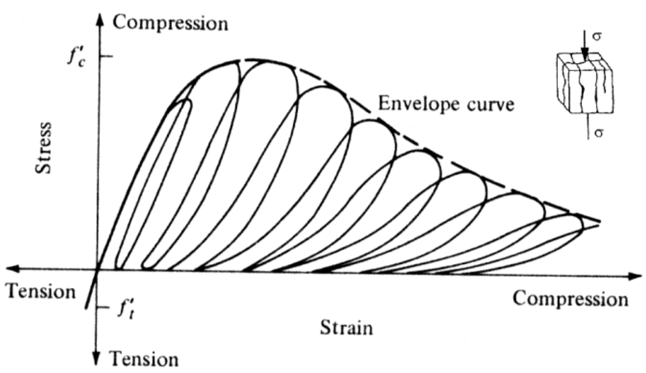
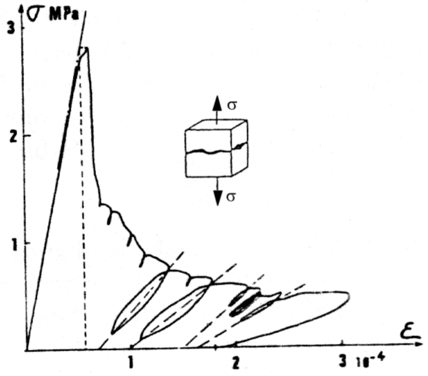


Figure 1. Influence of temperature on the behavior of poly(ethylene terephthalate) (PET) under equal biaxial (EB) deformation at a nominal strain rate of 1/s.

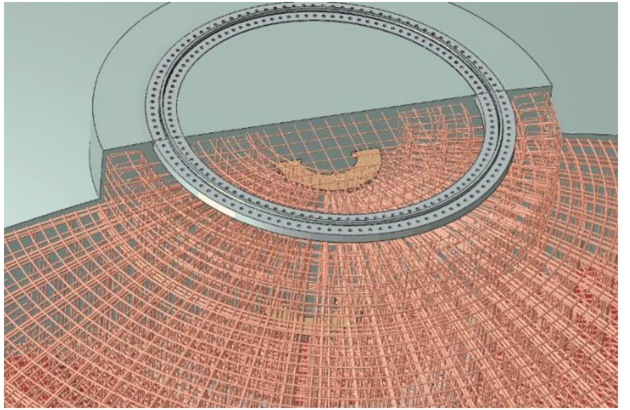
Material nonlinearity

- 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

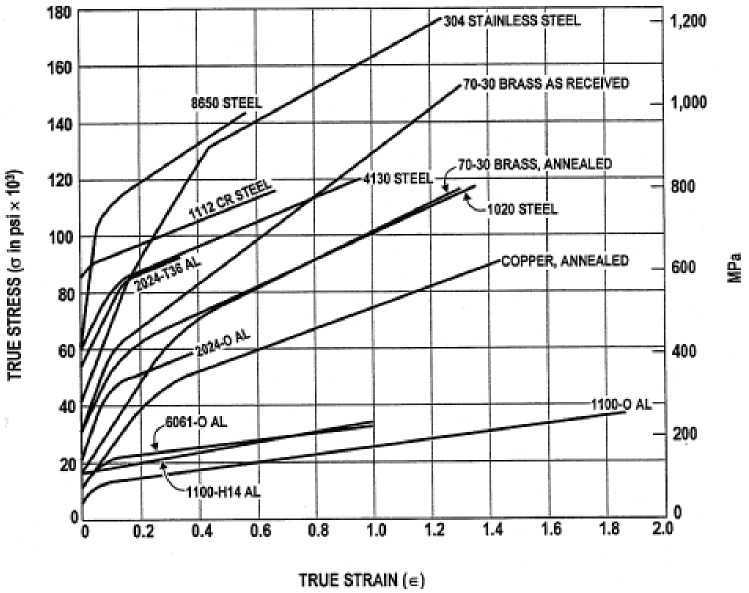


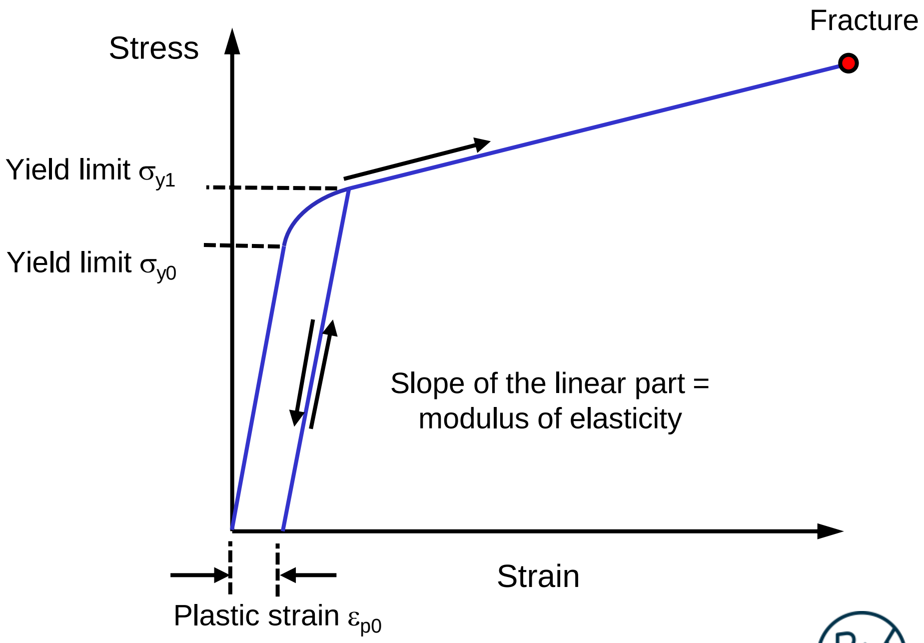
Plasticity

- 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

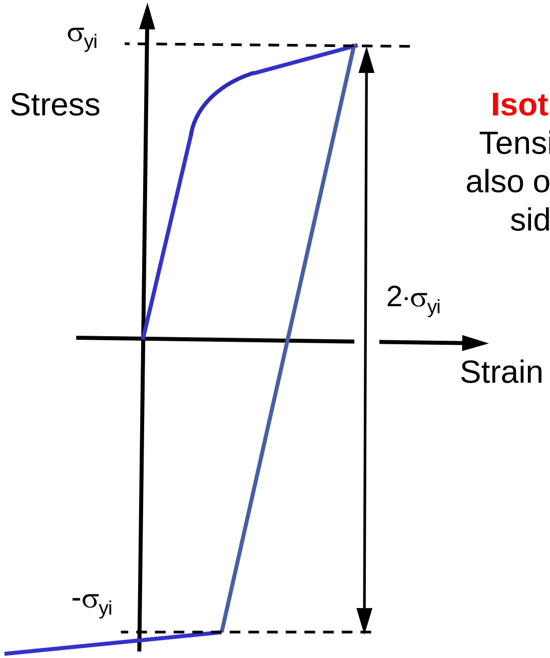
Plasticity, hardening

- Hardening: Yield limit grows with plastic deformation

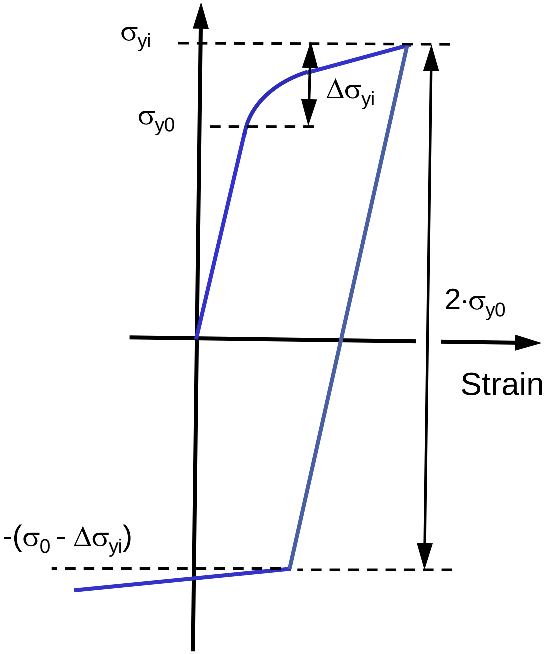




Plasticity, hardening



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



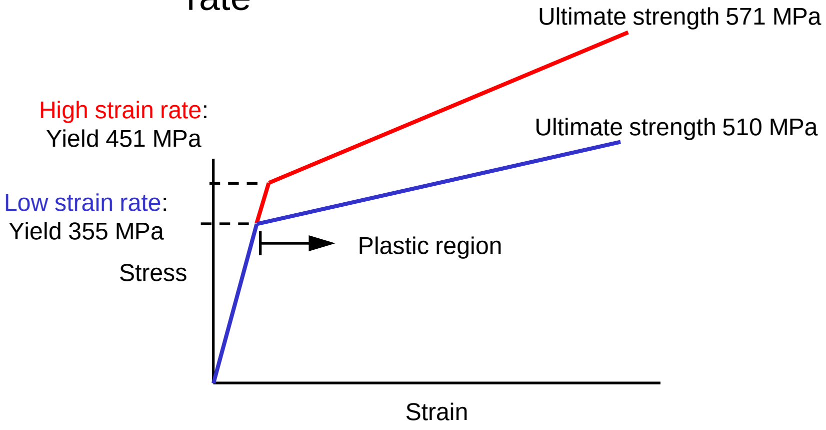
Kinematic hardening:
If the yield limit increases on the tension side, it decreases on the compression side.

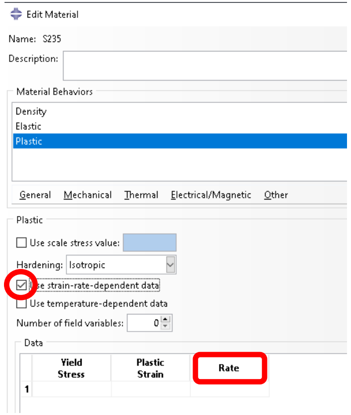
Bauschinger efect

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





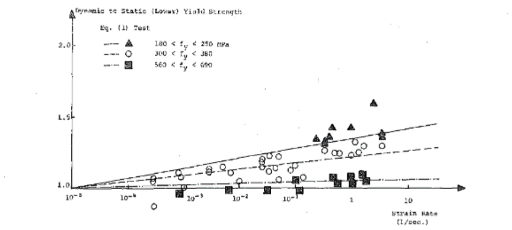


FIG. 2.—Strain Rate Effects on Lower Yield Strength of Steels with Different Yield Strengths

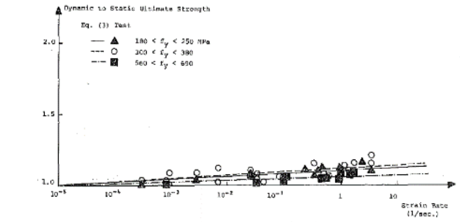


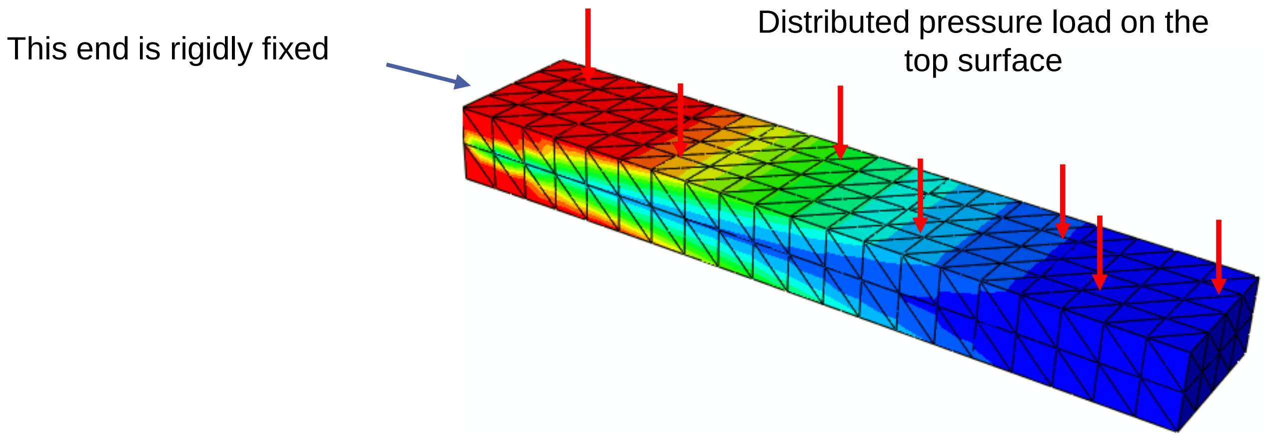
FIG. 4.—Strain Rate Effects on Ultimate Strength of Steels with Different Yield Strengths

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_{y0}
  - 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

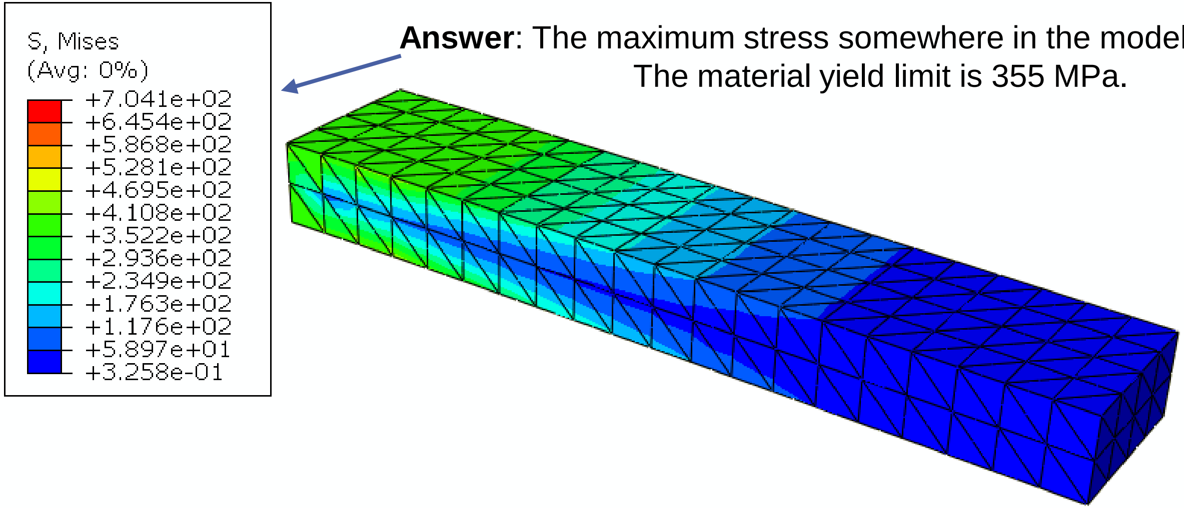
Plasticity, example

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



Plasticity, example

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

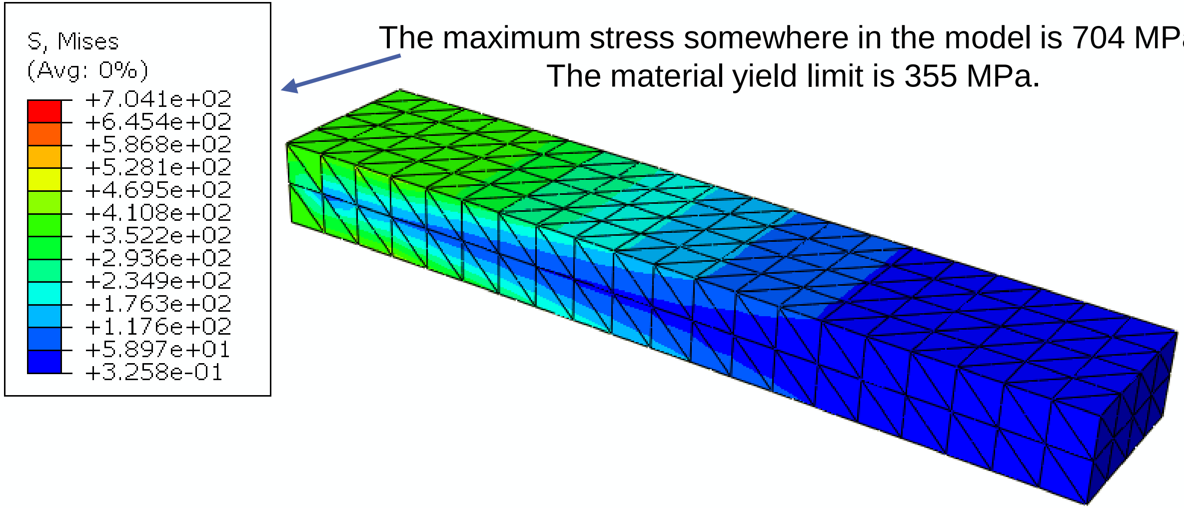


Answer: The maximum stress somewhere in the model is 704 MPa. The material yield limit is 355 MPa.

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.

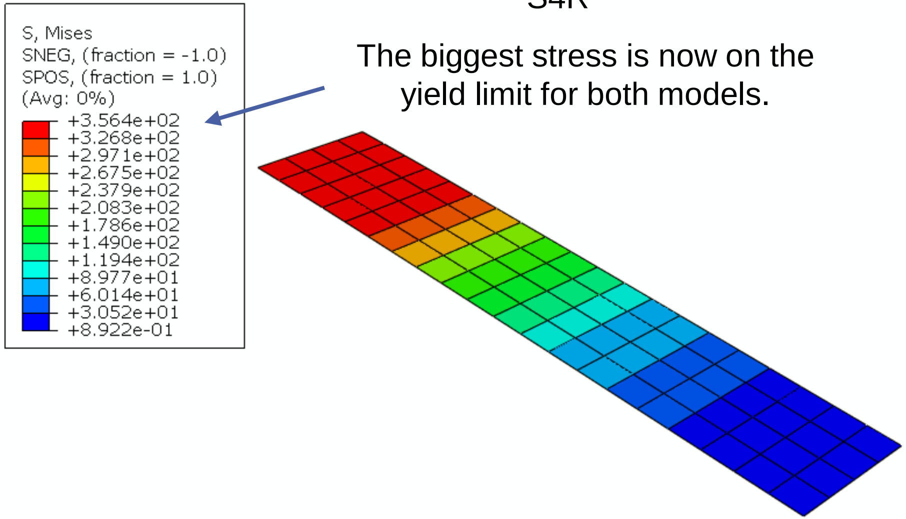


Plasticity, example

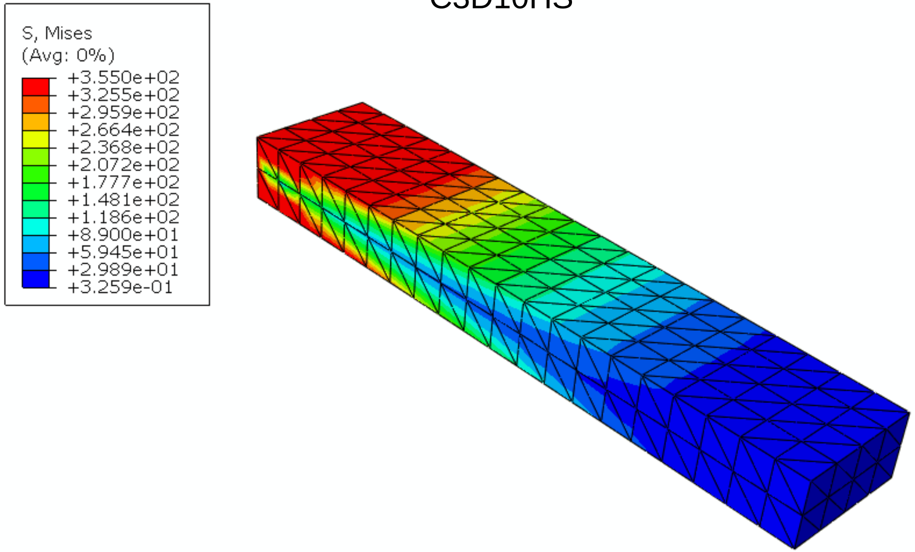
- When calculated with linear shell elements S4R and solid tetrahedrons C3D10HS (integration points on the surface)

S4R

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



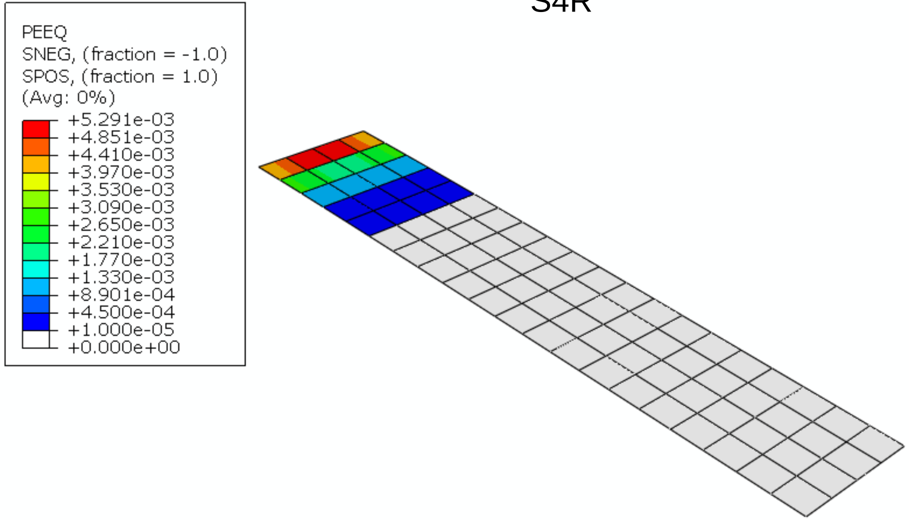
C3D10HS



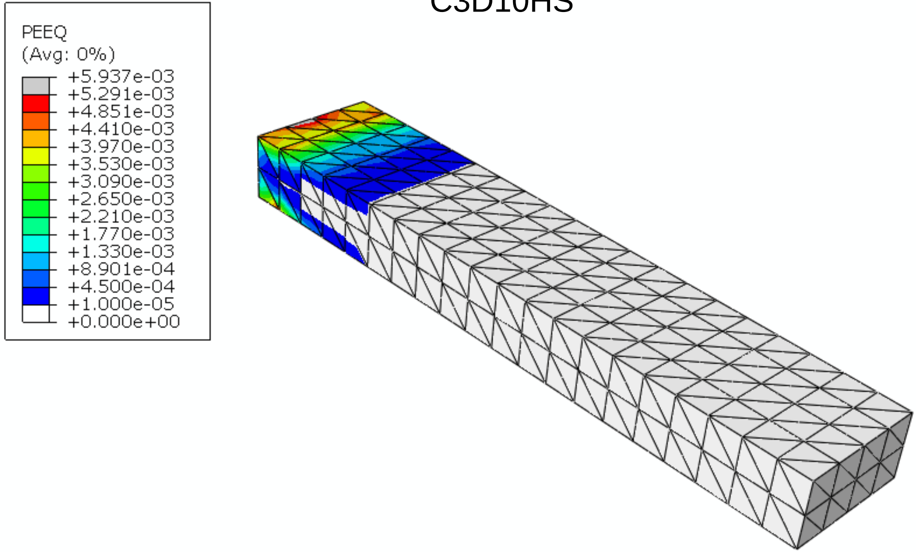
Plasticity, example

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

S4R



C3D10HS

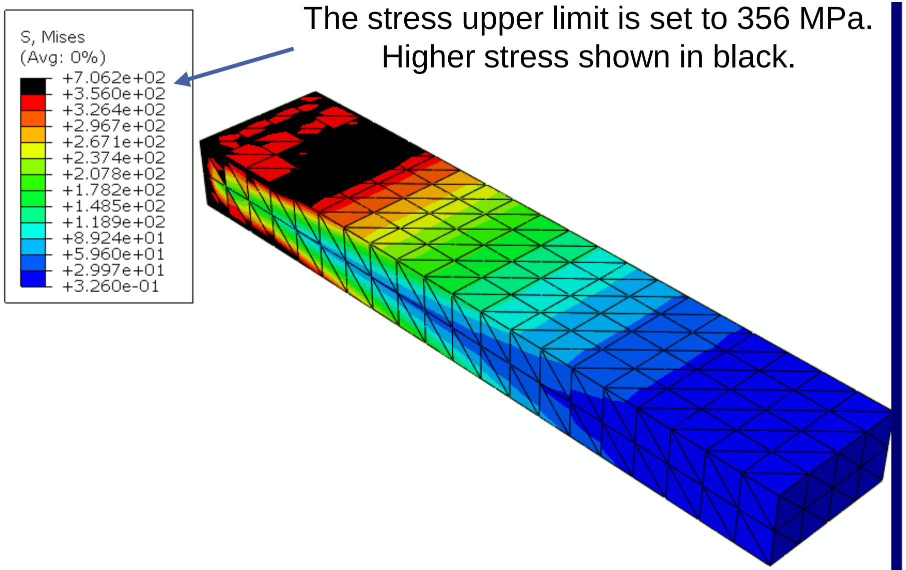


Plasticity, example

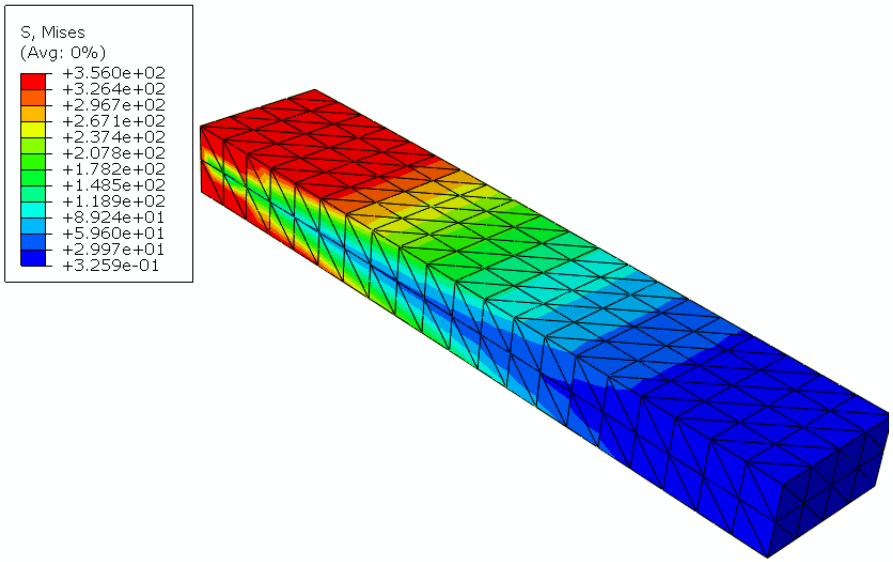
- However, even with ordinary solid tetras, the tension result is not as bad as it first appeared

C3D10

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

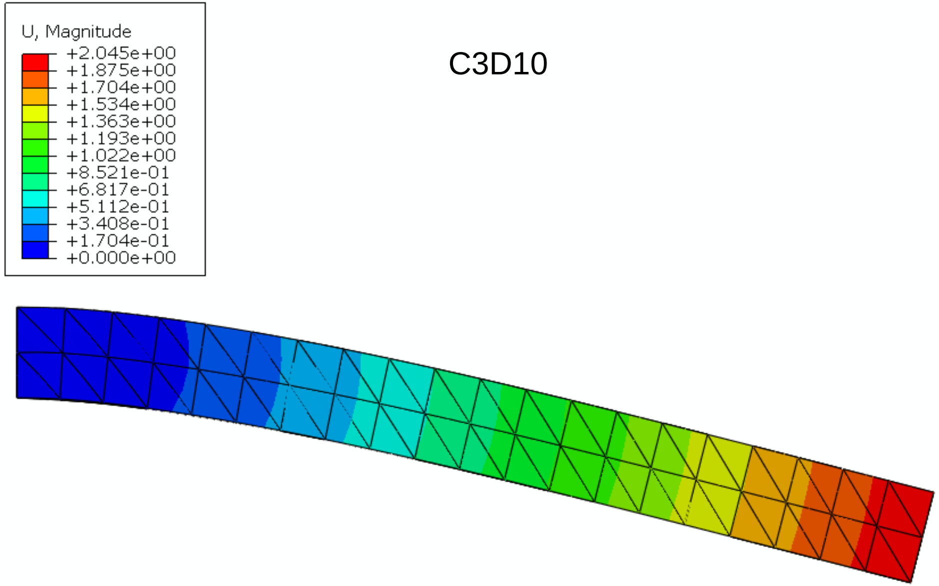


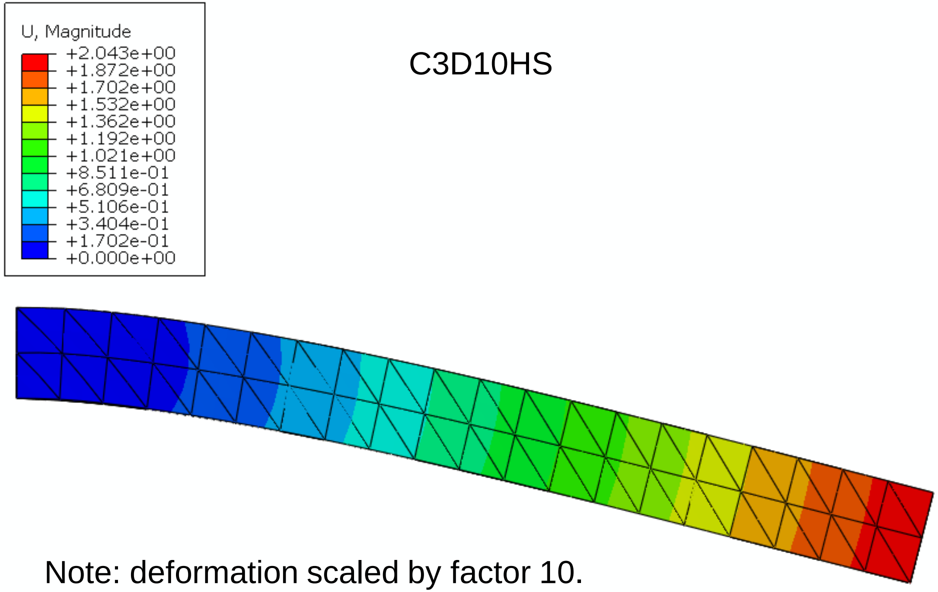
C3D10HS



Plasticity, example

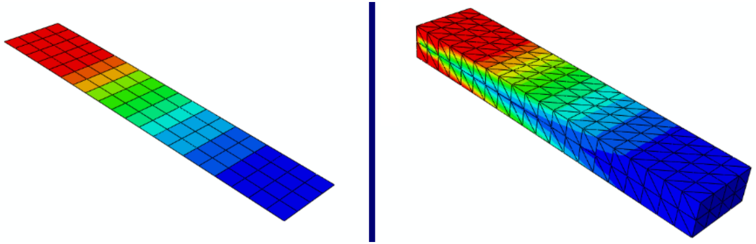
- The displacement result is practically the same in all models.





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

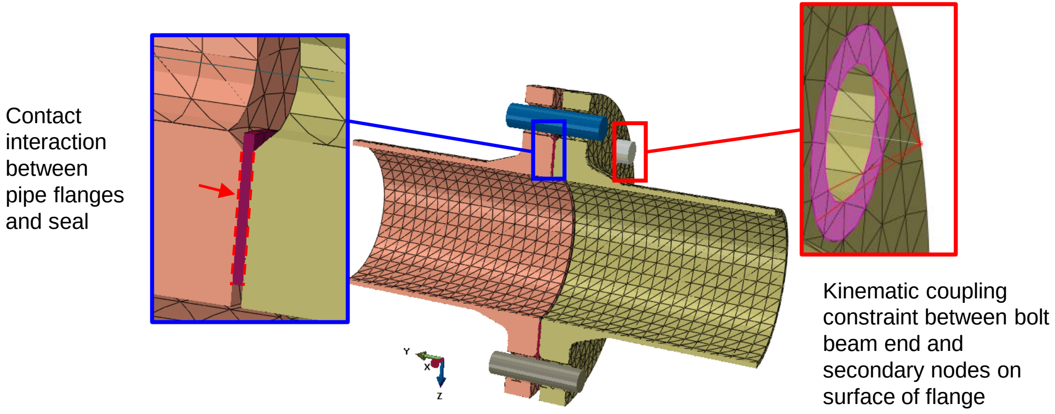
Answer 2: The C3D10HS elements incorporate the internal pressure DOF in the calculation. H = Hybrid element

Constraints and contact

WE MAKE
YOU INNOVATE

Constraints and connector elements

- By default, part instances in assembly do not interact with each other in any way
- The user must specify interactions between the parts using constraints, contacts, or elements



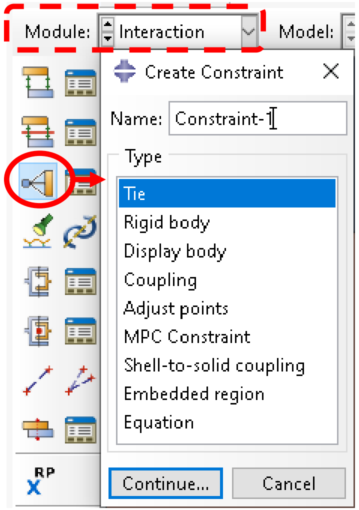
Example: contacts and constraints in bolted flange joints model

Constraints:
- remain active the full duration of the analysis
- status does not change

Contact:
- can be activated or deactivated in different steps
- status can change during the analysis step; open ⇔ close, slip ⇔ stick

Constraints

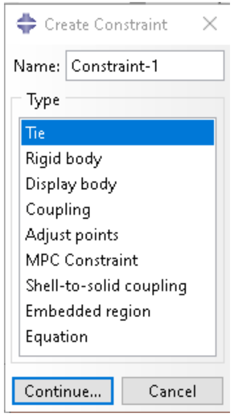
- Constraints are used to model kinematic relationships between points
- These relationships are defined between degrees of freedom in the model
- Constraints are created in the Interaction module

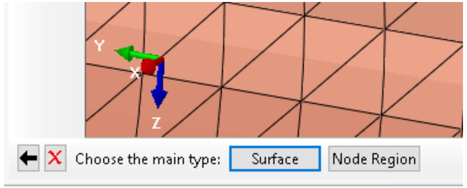


WE MAKE
YOU INNOVATE

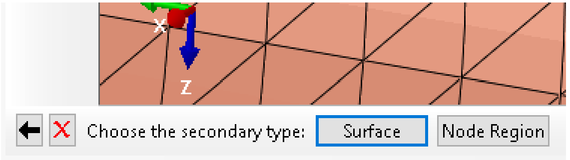
Tie constraints

- Describes a fully constrained contact
- A simple way to bond nodes/surfaces together permanently
- No slippage or separation of contacting nodes/surfaces regardless of loading magnitude





Main type should be *Surface*, unless node-to-node constraint is needed



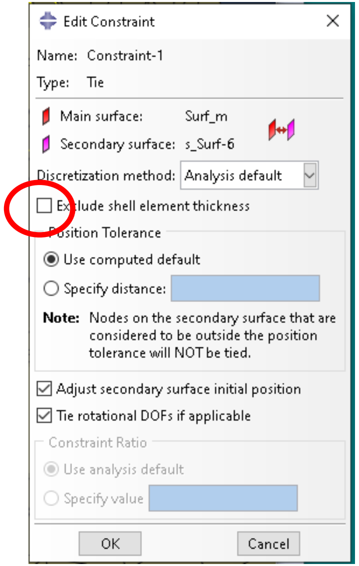
For *Secondary* type use

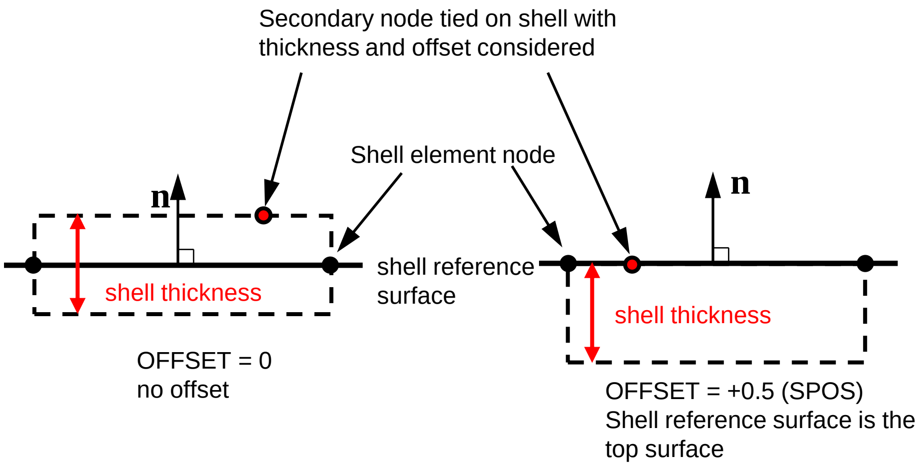
- *Surface* for more accurate stress output on tied surfaces. The constraint is applied in an average sense.
- *Node region* to exactly constrain specific nodes

WE MAKE
YOU INNOVATE

Tie constraints

Shell element thickness and offset are considered by default, but can be excluded if desired





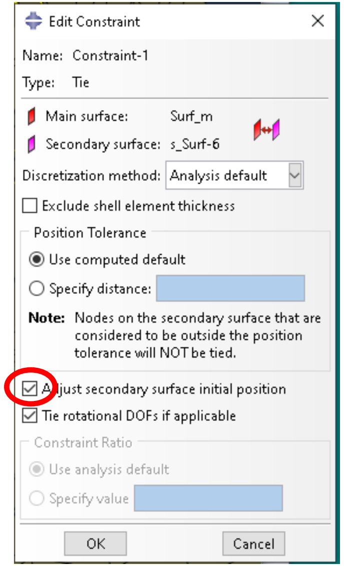
WE MAKE
YOU INNOVATE

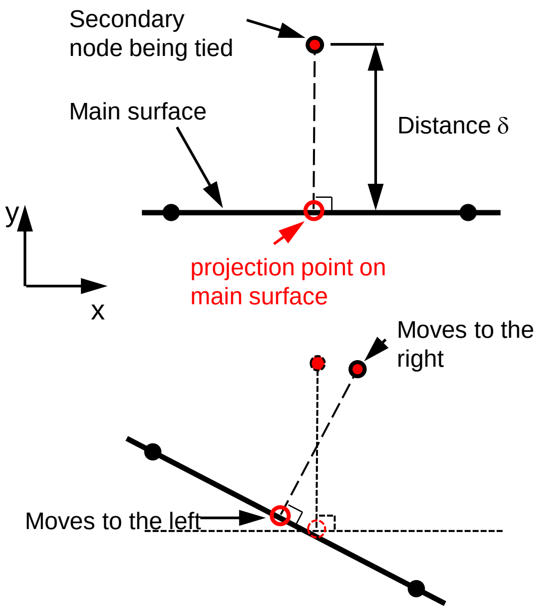
Tie constraints

Use *Adjust* to bring the secondary nodes exactly to the main surface

- This is especially important when tying solid element surfaces as shown below

The mesh is modified by moving the secondary nodes to the main surface in the beginning of the analysis without causing stress





Tie constraint:
Displacement of secondary node equals the displacement of projection point at the main surface:

$$u_{xS} = u_{xM}$$
$$u_{yS} = u_{yM}$$

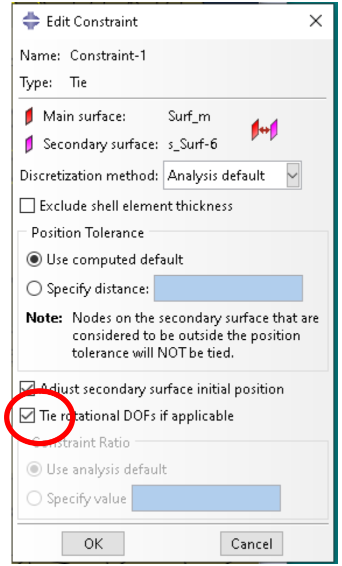
In case of rigid body rotation, the displacements of secondary node and the projection point are not equal. The tie constraint prevents the rigid body rotation and will cause unphysical stiffness and stresses *).

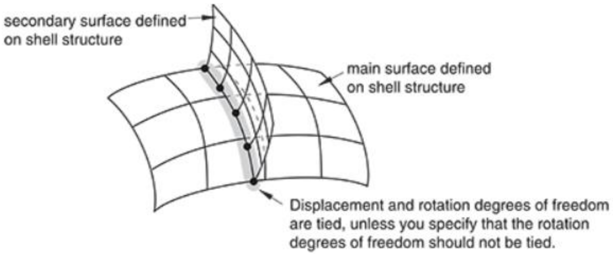
Rigid body rotation is NOT a problem when one surface is based on beam or shell elements with rotational degrees of freedom. In these cases, the rotation is correctly accounted for.

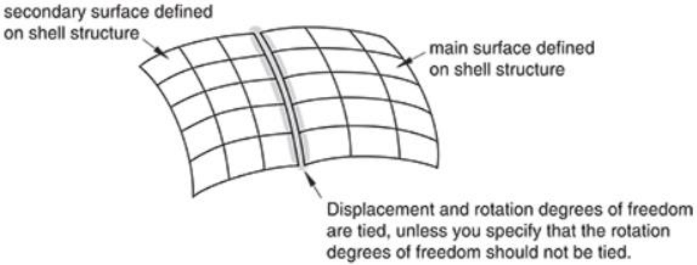
*) In Abaqus/Standard under certain conditions the rigid body rotation is correctly enforced for solid element meshes. Check this from the user documentation if needed.

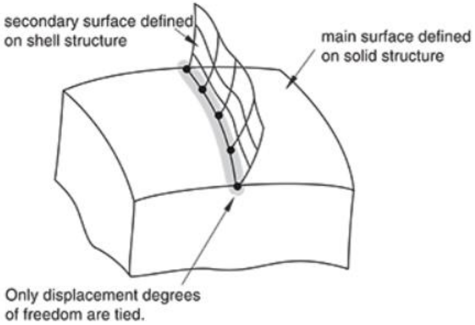
Tie constraints

By default, Abaqus will constrain the rotational degrees of freedom when they exist on both secondary and main surfaces









Note*:* Use *Shell-to-solid coupling* to constrain shell rotations to a solid element mesh

WE MAKE YOU INNOVATE

Rigid bodies

Rigid body is a collection of nodes and elements whose motion is governed by the motion of a single node called a reference node

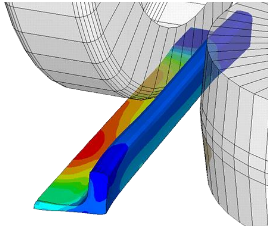
Any part instance or subregion of a part instance can be declared as a rigid body

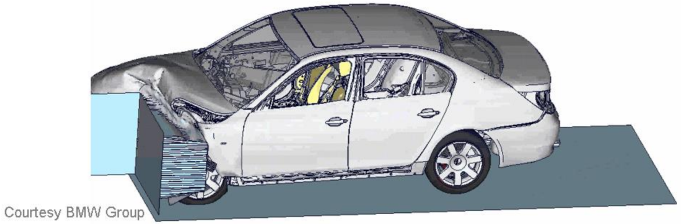
Rigid bodies are computationally efficient

- The 3D motion of the rigid body is completely defined by only 6 degrees of freedom
- Deformation and stress calculations are not done for a rigid body

Model a body as rigid if it is much stiffer that the other bodies it will come to contact with. Examples:

- Dies in metal forming simulations
- Massive bodies in impact simulations





Rigid bodies

Recommended approach

There are three approaches to geometry definition for rigid bodies

1. Define a rigid body using elements and assign a rigid body constraint; any meshed geometry and shape
2. Define an analytical rigid surface; surface geometry of limited shape
3. Write a subroutine RSUFRU; Abaqus/Standard only, not considered further in this course

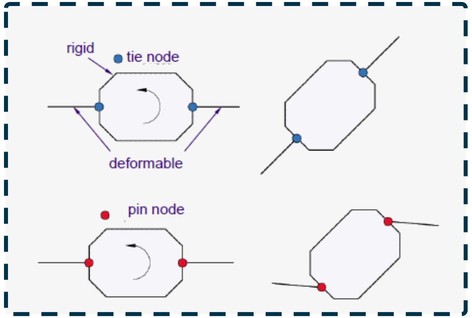
A rigid body definition consists of 1 reference node and at most

- 1 element set
- 1 tie node set
- 1 pin node set
- 1 analytical surface

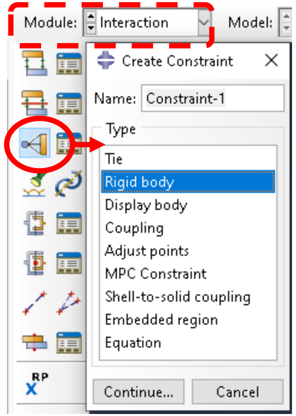
Each rigid body definition must be unique: Multiple rigid bodies cannot share nodes or elements.

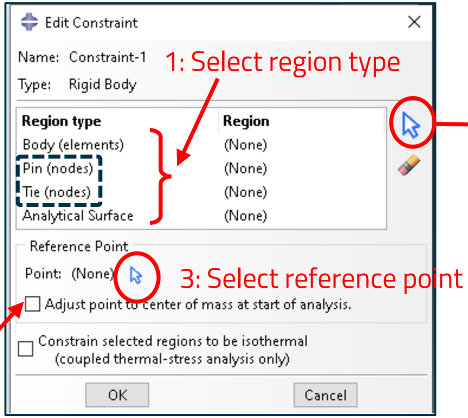
Rigid bodies

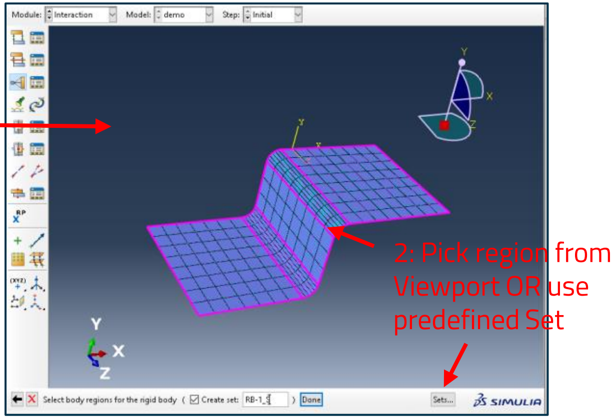
Approach 1:
- Create a part and mesh it
- Define a rigid body constraint as shown



Tie node: Rotation constrained
Pin node: Acts like a hinge







Same procedure when selecting the *Reference Point*

To model inertia correctly in a dynamic analysis, the rigid body reference point must be in the centre of mass of the rigid body. Using this option, Abaqus will automatically calculate the location of the rigid body based on the element geometry and properties.

Note: If the rigid body motion is constrained using boundary conditions, then the reference point can be anywhere.

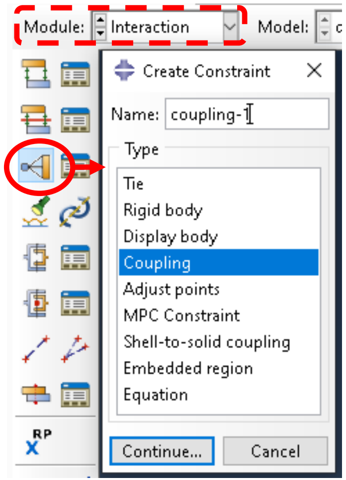
```
RIGID BODY MASS AND INERTIA
---------------------------

REFERENCE NODE IS:          1

POSITION OF THE REFERENCE NODE
      X                 Y                Z
  -7.749675         -14.52178         25.00000

TOTAL MASS OF RIGID BODY

  1.053615OE-05
```

Rigid body mass data is written into the .dat file

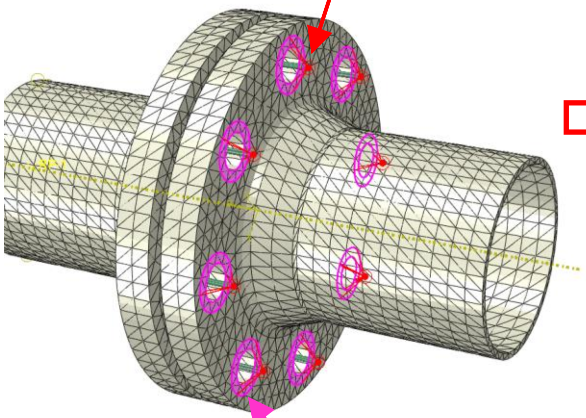
WE MAKE
YOU INNOVATE

Kinematic and Distributing Coupling constraints

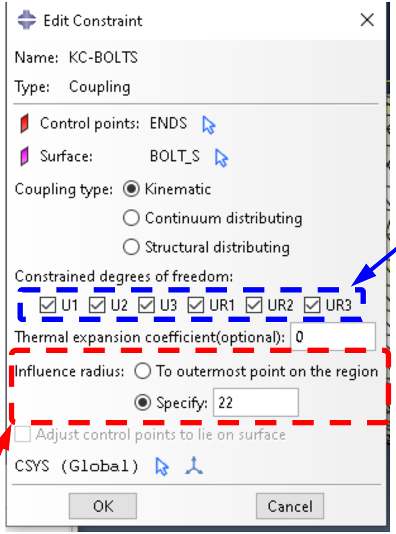
A group of coupling nodes is constrained to the motion of a reference node



Select the constraint control point/points



Select the constrained surface/node region



Select which DOFs from the nodes on the surface are constrained

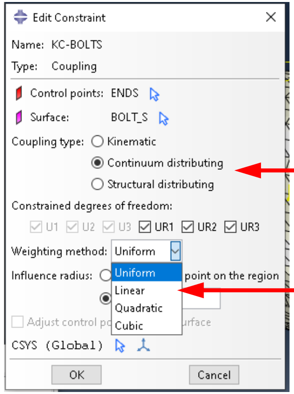
Only those nodes of the constrained surface that fall inside the influence radius from each control point are constrained.

WE MAKE YOU INNOVATE

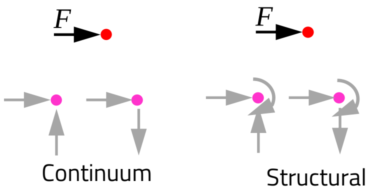
Kinematic and Distributing Coupling constraints

Two types are available

- Kinematic: Constraints the motion of the coupling nodes rigidly to the motion of the reference node
- Distributing: Applies the constraint in an average sense. The reference node displacement and rotation correspond to a weighted-average motion of the coupling nodes



Continuum distributing: Only forces are applied to coupling nodes
Structural distributing: Forces and moments are applied to coupling nodes

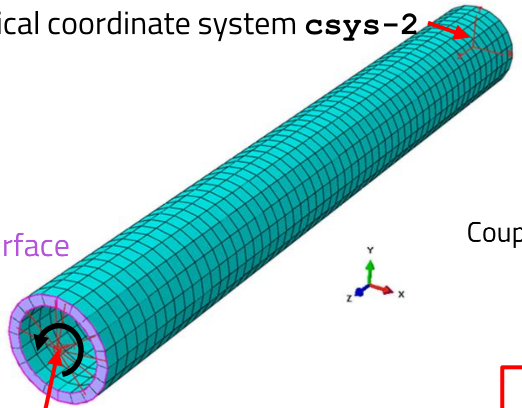


Weighing factor options. Constant and Linear/Quadratic/Cubic reduction as a function of distance from the reference node

WE MAKE YOU INNOVATE

Coupling constraint example

Cylindrical coordinate system **csys-2**

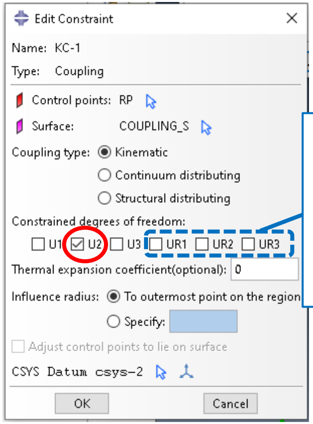


Coupled surface

Constraint control point

Torque applied here

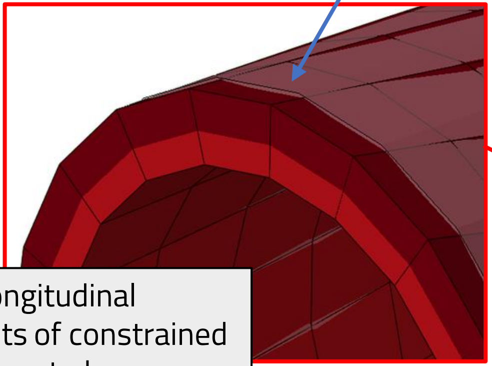
U1: Radial displacement
U2: Circumferential displacement
U3: Z displacement



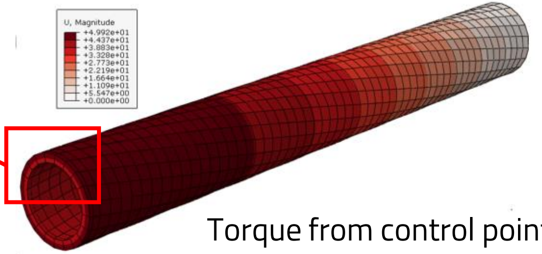
Note: As solid elements have no rotational displacements, selecting or unselecting UR1, UR2, UR3 has no effect when constrained surface is composed of solid elements.

Coupling refers to coordinate system **csys-2**

Undeformed shape shown



Radial and longitudinal displacements of constrained surface are free to happen

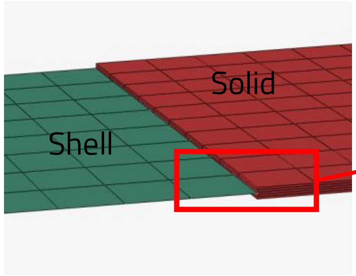


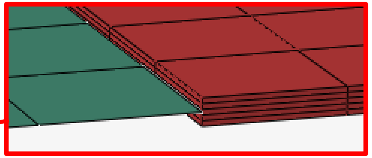
Torque from control point is transmitted to solid element tube

WE MAKE
YOU INNOVATE

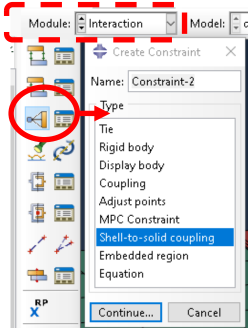
Shell to solid coupling

Shell to solid coupling is used to attach solid and shell element meshes so that the translation DOFs of the solid are coupled to the rotation DOFs of the shell

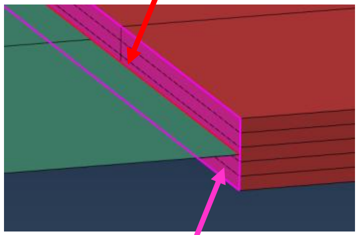




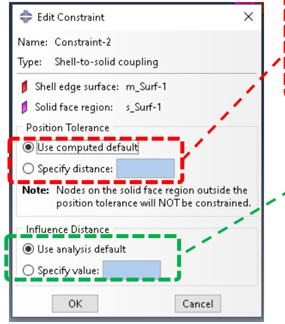
Bending stiffness correctly modelled



1: Select shell edge



2: Select surface or node region from solid



Position tolerance: absolute distance from the solid surface within which all shell nodes to be included in the coupling must lie. Shell nodes that lie outside this tolerance are not coupled to the solid surface.

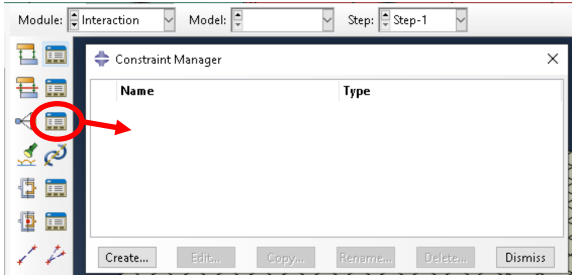
Influence distance: absolute distance from the shell edge within which the solid surface node must lie to be included in the coupling. Default is half of the shell thickness

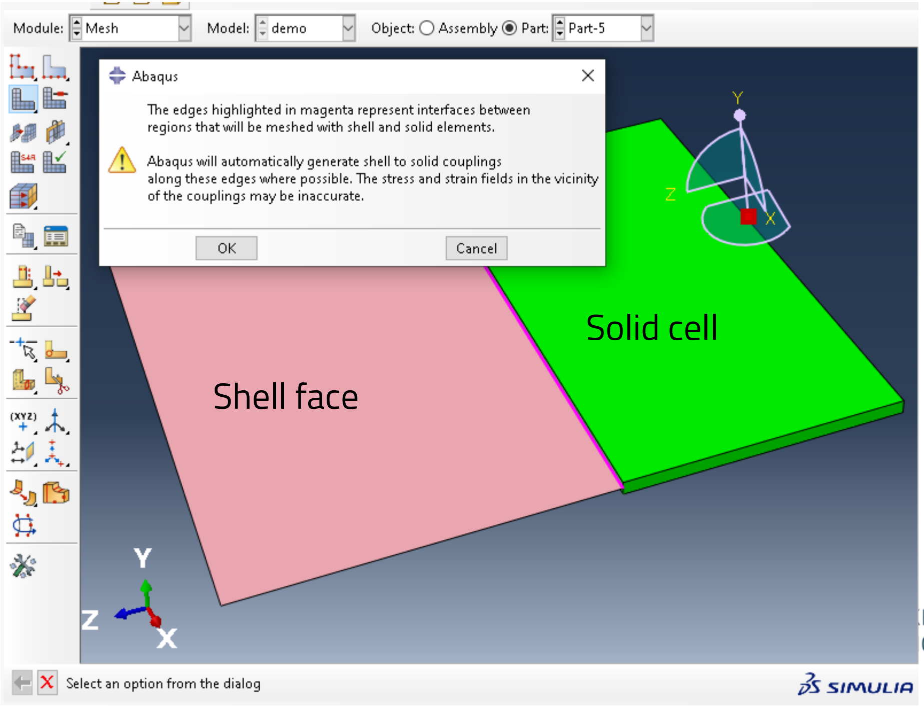
WE MAKE
YOU INNOVATE

Shell to solid coupling

If a part contains both shell and solid geometry, shell-to-solid coupling constraints are automatically created along the shell-solid interfaces during meshing

- Default settings are used for *Position Tolerance* and *Influence Distance*
- The constraint is internally generated and does not show in the *Constraint manager*

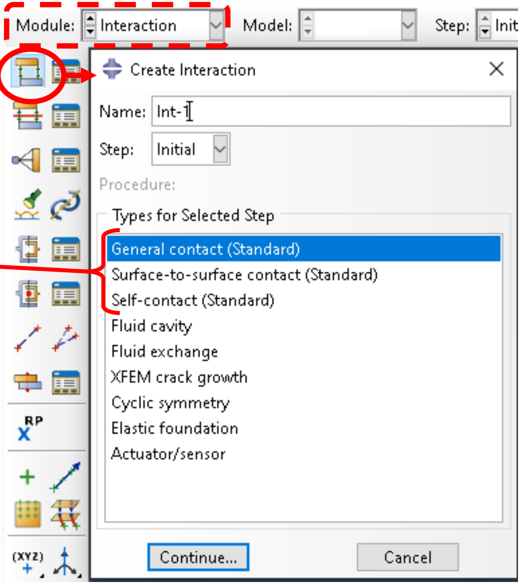


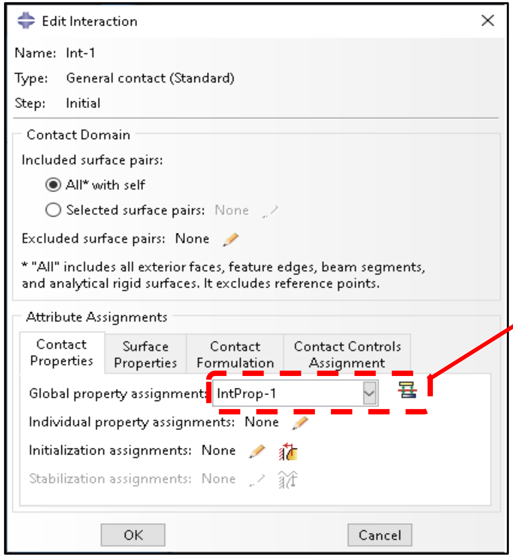


Contact interactions

- Contacts are used to model interactions between surfaces or nodes
- Contact status can change during the analysis depending on external loads
  - In the normal direction the contact can open or close
  - In the tangential direction the contact can slip or stick

Contact interactions





Contact interaction refers to an *Interaction property*. See next slide.

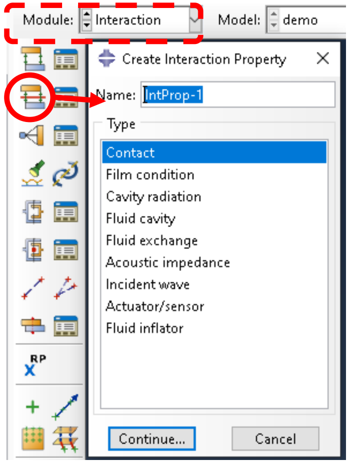
WE MAKE YOU INNOVATE

Contact interactions

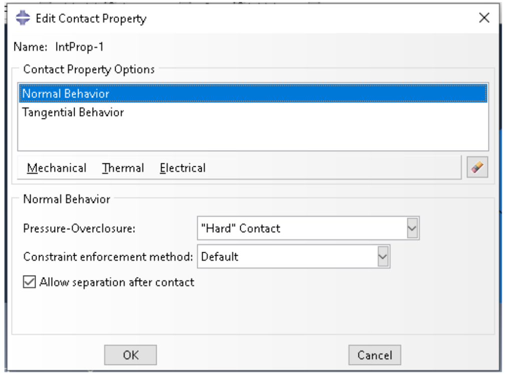
- Abaqus has two ways of defining contact interactions
  1. Contact pairs
     - Traditional way of defining contact
     - User selects and creates contact pairs of all surfaces that are likely to be in contact during the analysis
     - Can result in a more efficient analysis since contact interactions are limited in scope
  2. General contact
     - Automated creation of contacts inside Abaqus
     - Possibility for user to manually include or exclude surface pairs
     - Generally, the preferred method
- The choice between general contact and contact pairs is largely a trade-off between ease of contact definitions and analysis performance
  - Accuracy and robustness of both methods is similar
  - Both can be used together in the same analysis

Contact interaction properties

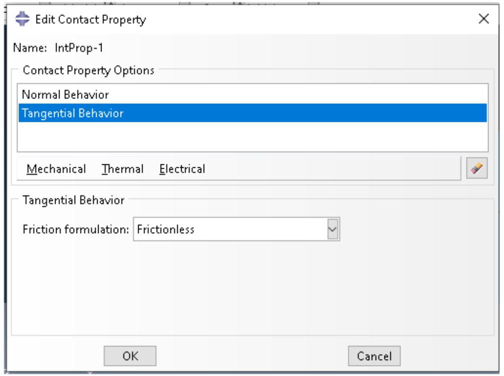
- Contact interaction definition requires contact properties



Properties in normal direction



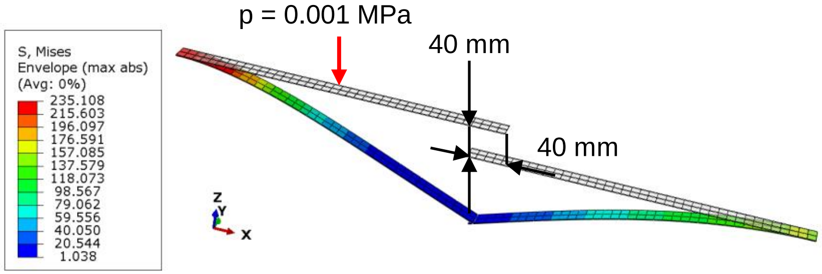
Properties in tangential direction

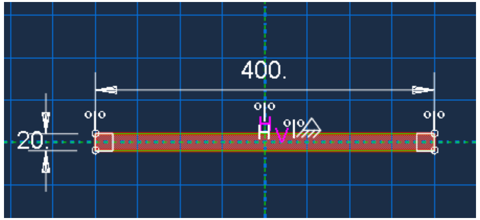


WE MAKE
YOU INNOVATE

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
- Frictionless contact

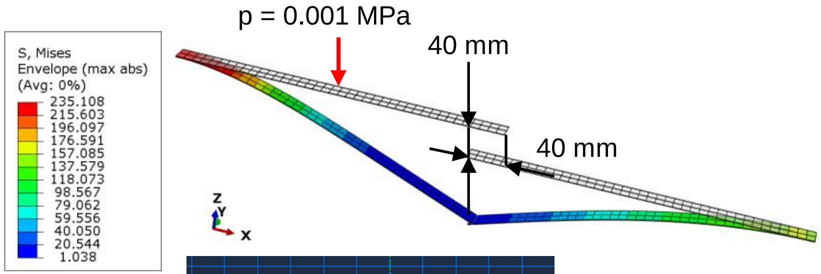


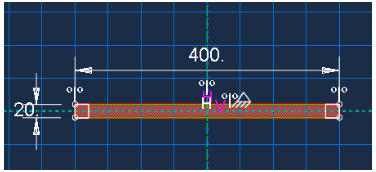


Sketch of planar shell

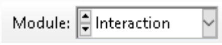
Workshop; Contacting beams

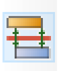
- Use a shell element model
- Dimensions [mm], L x W x t = 400 x 20 x 1
- E = 200 000 MPa, ν = 0.3
- Pressure load on top beam: 0.001 MPa
- Interaction properties for contact, see below.

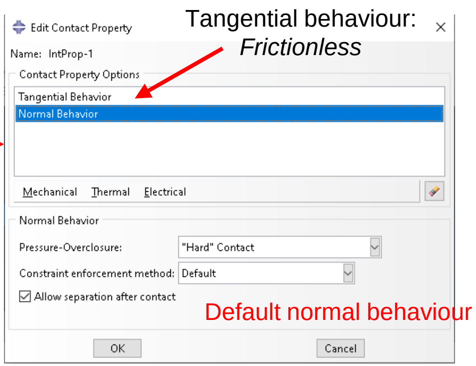


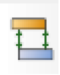


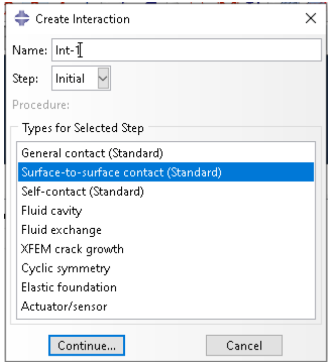
Sketch of planar shell

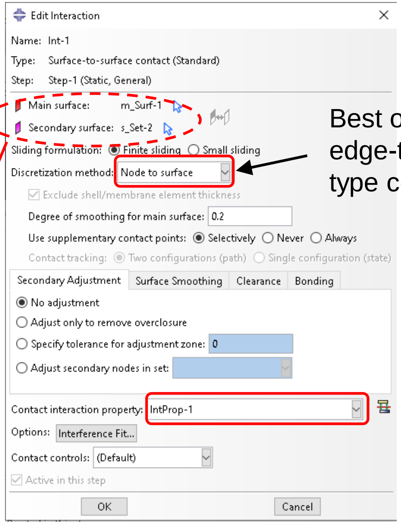




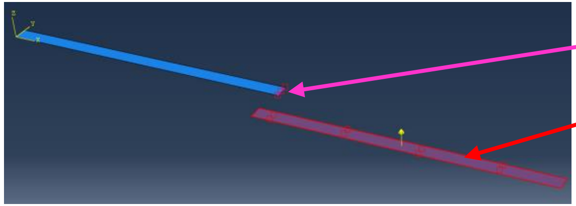








Best option for edge-to-surface type contact

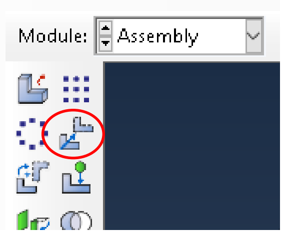


Secondary surface: End edge of topmost beam

Main surface: Top surface of lower beam

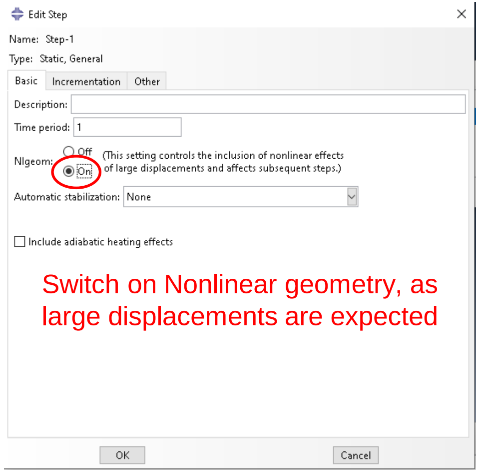
Steps

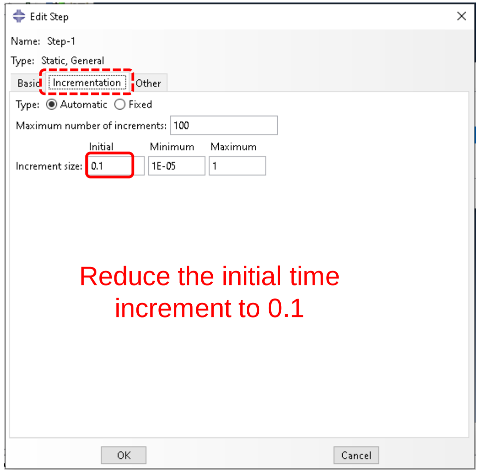
- Create a shell part. Two options for base feature
  - Planar shell
  - Extruded shell
- Create material and shell section
- Assign shell section
- Mesh part
- Create two instances of the same part in the Assembly
  - Both beams are identical, so no need to define two identical parts. Instead, two instances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

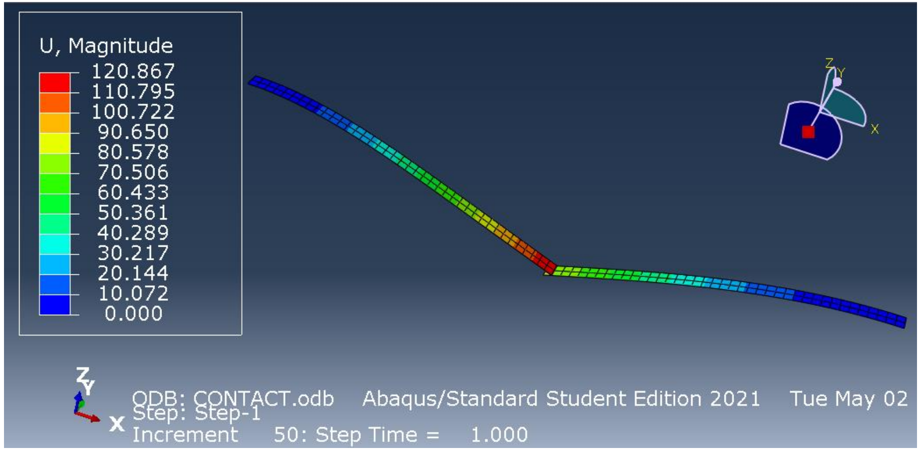
The incrementation of the load is automatically adjusted by Abaqus based on the convergence history.

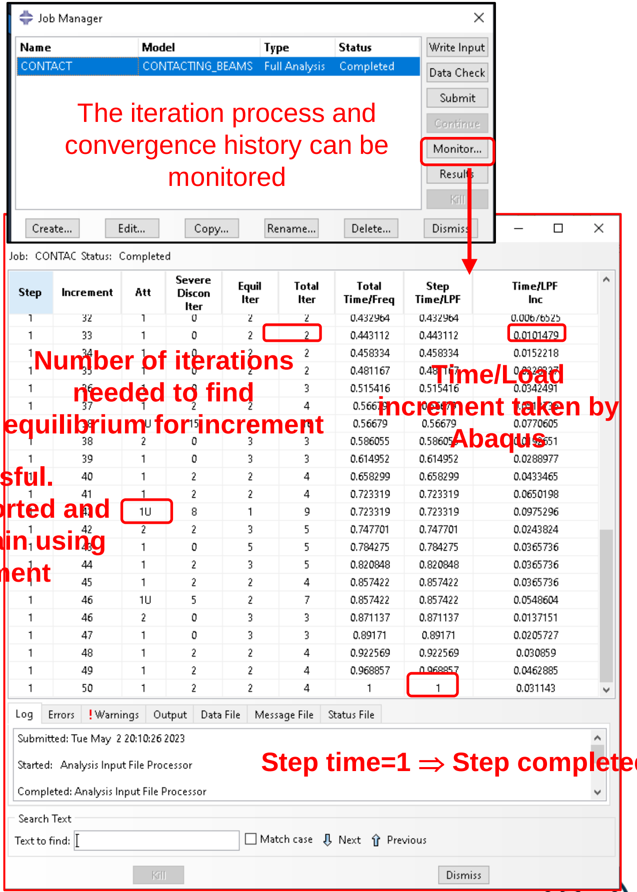
- In the *Interaction module* create a *Contact property* and *Contact interaction* as shown on the previous slide
  - By default, parts in the assembly do not interact with each other in any way
  - The analyst must define contacts or constraints between the parts

Steps, continued

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

Displacements of contacting beams





The iteration process and convergence history can be monitored

Number of iterations needed to find equilibrium for increment

Time/Load increment taken by Abaqus

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

Step time=1 ⇒ Step completed

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

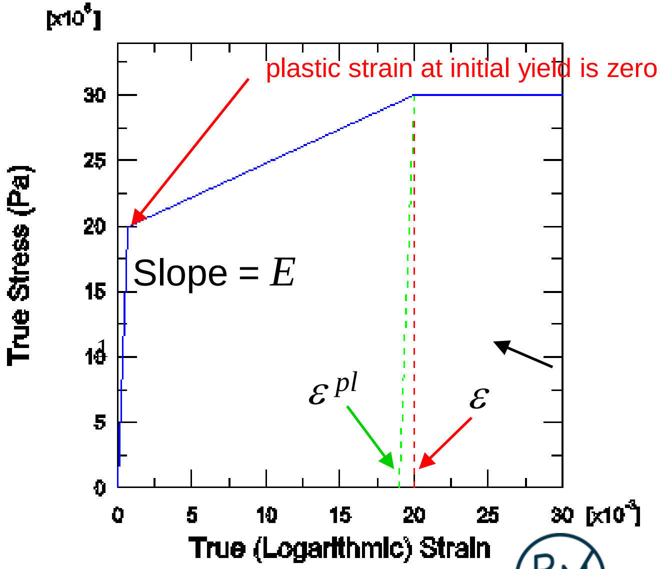
Nominal strain

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

True stress

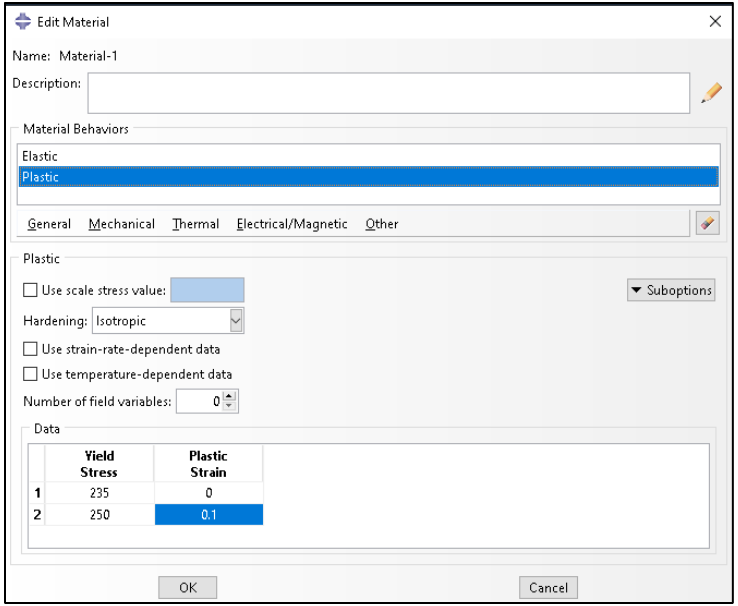
Nominal stress

Nominal strain

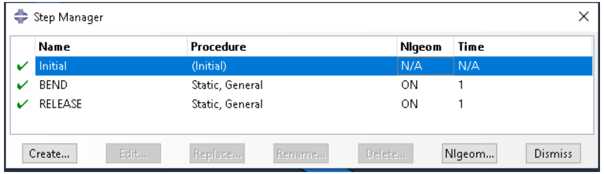


Material plasticity

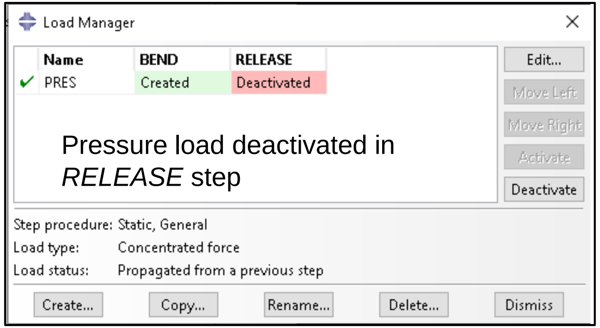
- Adding plasticity to Contacting beams example



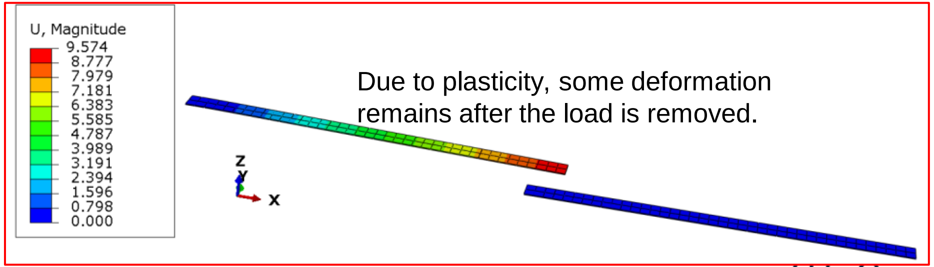
Plasticity added into material definition



RELEASE step added



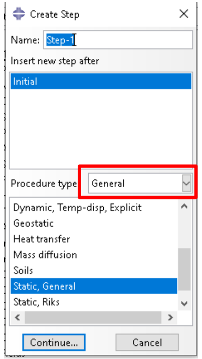
Pressure load deactivated in *RELEASE* step

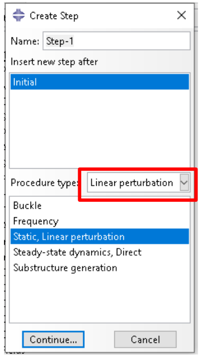


Due to plasticity, some deformation remains after the load is removed.

General and Linear perturbation steps

- Abaqus has General steps and Linear perturbation steps
- Linear perturbation = small displacements around a base state
  - Linear analysis ⇒
    - 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 affect 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

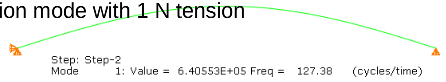
Step Name	Description
TENSION-1	
Step-2	
TENSION-2	
Step-4	

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

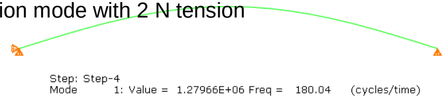
Index	Description
0	Increment 0: Base State
1	Mode 1: Value = 1.27966E+06 Freq = 180.04 (cycles/time)
2	Mode 2: Value = 5.11892E+06 Freq = 360.09 (cycles/time)
3	Mode 3: Value = 1.15187E+07 Freq = 540.16 (cycles/time)
4	Mode 4: Value = 2.04805E+07 Freq = 720.26 (cycles/time)

Applied force

Ux & Uy fixed

Uy fixed

1st vibration mode with 2 N tension



Analytical: 180.11

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.

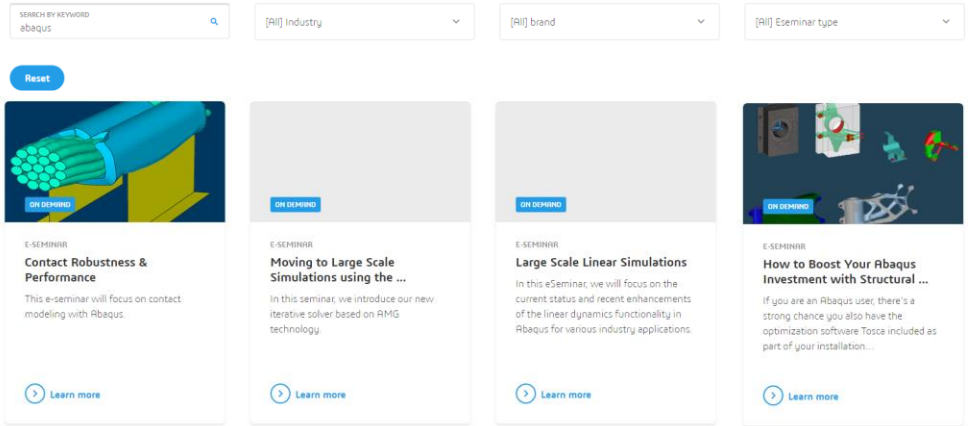
boltpipeflange_3d_solidgask.inp

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

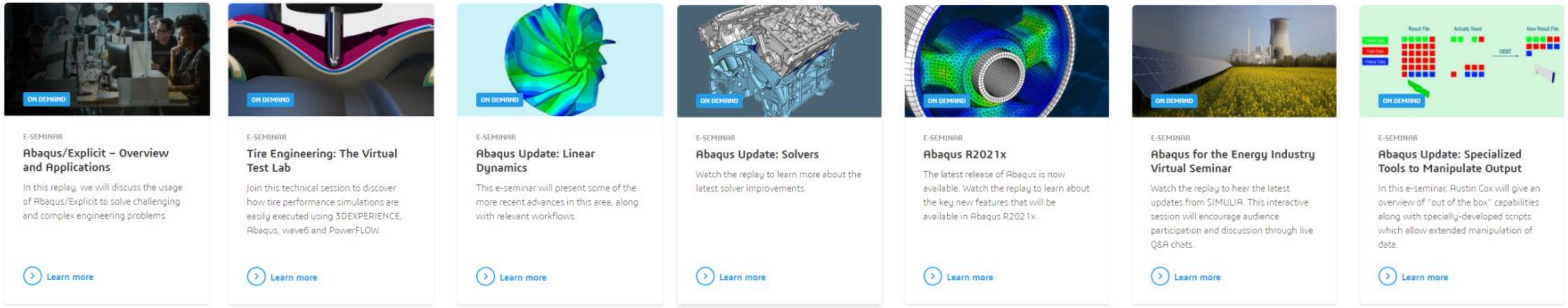
```
abaqus fetch job=boltpipeflange_axi_solidgask.inp
```

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/ $\Rightarrow$ E-Seminars



Thank you!