An Introduction to ABAQUS®

M. Salviato

William E. Boeing Department of Aeronautics and Astronautics
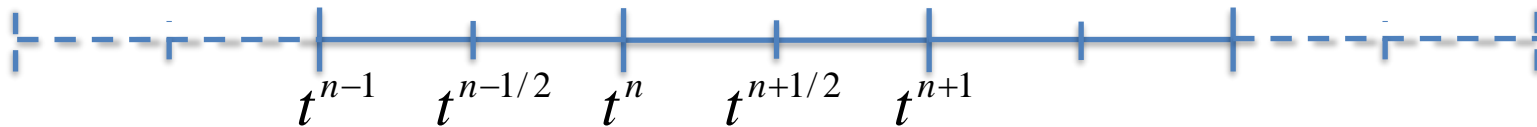
Seattle, Spring 2016
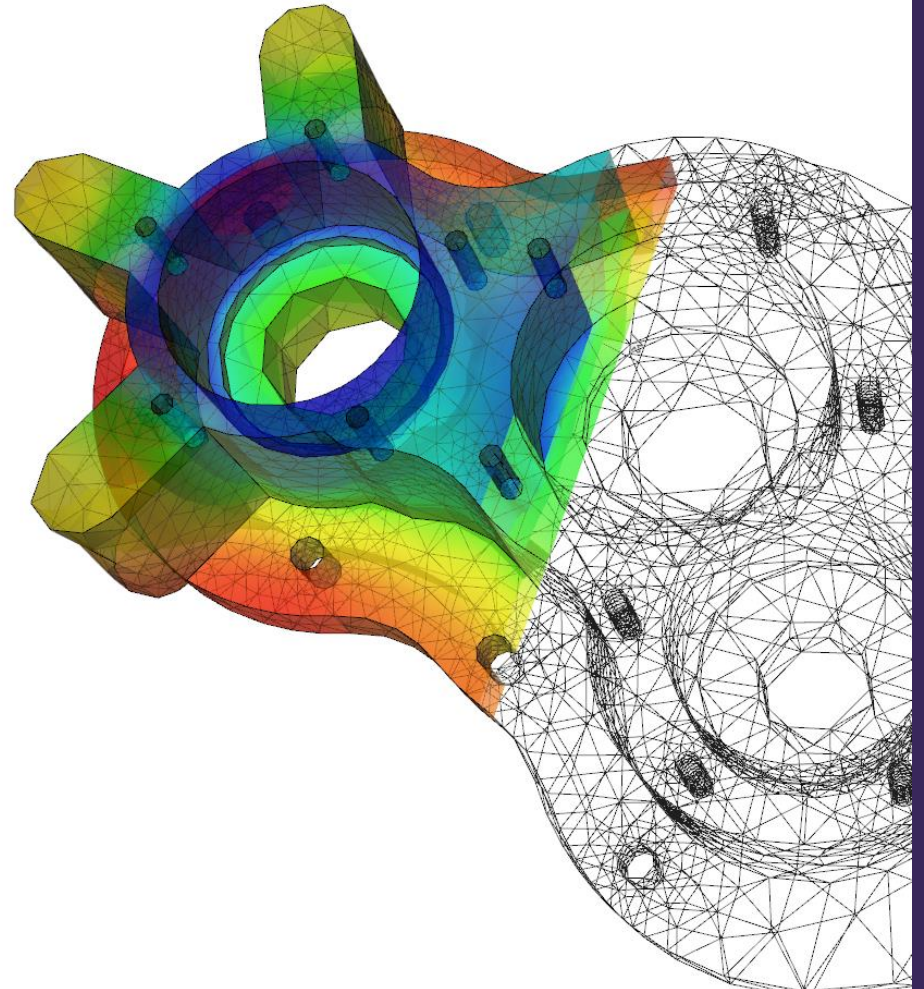
In crashing simulations an *explicit algorithm of solution* is usually preferred. It generally guarantees more robustness than implicit methods. ***ABAQUS uses a central difference scheme in time***.

Let's introduce the following convention:

$$\Delta t^{n+1/2} = t^{n+1} - t^n \qquad t^{n+1/2} = \frac{t^{n+1} + t^n}{2} \qquad \Delta t^n = t^{n+1/2} - t^{n-1/2}$$

$t^n =$ Time at n-th step of analysis

The problem to solve is dynamic. Accordingly, the following system of equations need to be solved:

$$\mathbf{Ma}^n = \mathbf{f}^n = \mathbf{f}^{ext}\left(\mathbf{u}^n, t^n\right) - \mathbf{f}^{int}\left(\mathbf{u}^n, t^n\right) \tag{1}$$

$\mathbf{M} =$ global mass matrix (usually diagonal)

$\mathbf{a}^n =$ global nodal acceleration vector

$\mathbf{f}^n =$ global nodal force vector

$\mathbf{f}^{ext}\left(\mathbf{u}^n, t^n\right) =$ global vector of nodal, external forces (calculated as done in FEA I class)

$\mathbf{f}^{int}\left(\mathbf{u}^n, t^n\right) =$ global vector of nodal, internal forces

Note that, if the material is linear elastic, the following expression can be used:

$$\mathbf{f}^{\text{int}}\left(\mathbf{u}^n, t^n\right) = \sum_p \iint_{\Omega_p} \mathbf{B}^T \mathbf{C} \mathbf{B} d\Omega_p \mathbf{u}^n = \sum_p \mathbf{K}_p \mathbf{u}^n$$

$\mathbf{K}_p =$ Element stiffness matrix of p-th element

$\mathbf{C} =$ Elastic stiffness matrix

Now, from the displacements of previous and current steps we can approximate the velocity at time n+1/2:

$$\mathbf{v}^{n+1/2} = \frac{\mathbf{u}^{n+1} - \mathbf{u}^n}{t^{n+1} - t^n} = \frac{\mathbf{u}^{n+1} - \mathbf{u}^n}{\Delta t^{n+1/2}}$$

This gives the following integration formula:

$$\mathbf{u}^{n+1} = \mathbf{u}^n + \Delta t^{n+1/2} \mathbf{v}^{n+1/2} \tag{2}$$

Now, the acceleration can be approximated as:

$$\mathbf{a}^n = \frac{\mathbf{v}^{n+1/2} - \mathbf{v}^{n-1/2}}{t^{n+1/2} - t^{n-1/2}}$$

Giving the following integration formula:

$$\mathbf{v}^{n+1/2} = \mathbf{v}^{n-1/2} + \Delta t^n \mathbf{a}^n \tag{3}$$

Now, re-arranging (1) and substituting it into (3), one gets:

$$\mathbf{v}^{n+1/2} = \mathbf{v}^{n-1/2} + \Delta t^n \mathbf{M}^{-1} \mathbf{f}^n$$

In the previous equation, the velocity at time n-1/2 is known, the mass matrix is known, the time increment in known and the vector of nodal forces at the current time n is known. Accordingly, one can easily calculate the vector of nodal velocity for step n+1/2. Once the velocity is known, the new nodal displacement vector can be easily calculated using the integration formula (2):

New displacement vector for time n+1

$$\mathbf{u}^{n+1} = \mathbf{u}^n + \Delta t^{n+1/2} \mathbf{v}^{n+1/2}$$

The updated displacement field can then be used to calculate nodal forces etc at the new step n+1

Advantages of explicit method:

- Easy to implement
- Very robust (if some conditions are met, it converges very often)
- No extra matrix inversions per step needed

Disadvantages of explicit method:

- Conditionally stable (there are some conditions on the size of the time step to converge)
- Equilibrium is not imposed explicitly

It can be shown that a necessary condition for stability applies:

Stable time step $\quad \Delta t = \alpha \Delta t_{crit},$ $\qquad \Delta t_{crit} = \dfrac{2}{\omega_{max}} \leq \min_{e,I} \dfrac{2}{\omega_{max}} = \min_{e} \dfrac{l_e}{c_e}$

$\omega_{max} =$ Maximum frequency of the linearized system

$l_e =$ Element characteristic length (e.g. diagonal length)

$c_e = \sqrt{\dfrac{E}{\rho}} =$ Sound wave propagation velocity in the material

$E =$ Young's modulus

$\rho =$ Material density

$\alpha =$ Courant number. Good practice values: [0.8,0.98]

Even 1 very small element can reduce the stable time step significantly!

The stiffer and the less dense the material, the smaller the time step!

The stable step can be very small! This means that if the total time of the simulation is not sufficiently small, the number of iterations is huge!

Avoid small elements when possible!

Use explicit solver only for short simulations (unless mass scaling is possible)

can

mass

V_z=15 m/s

u_x=u_y=u_z=0

100

60

60

160
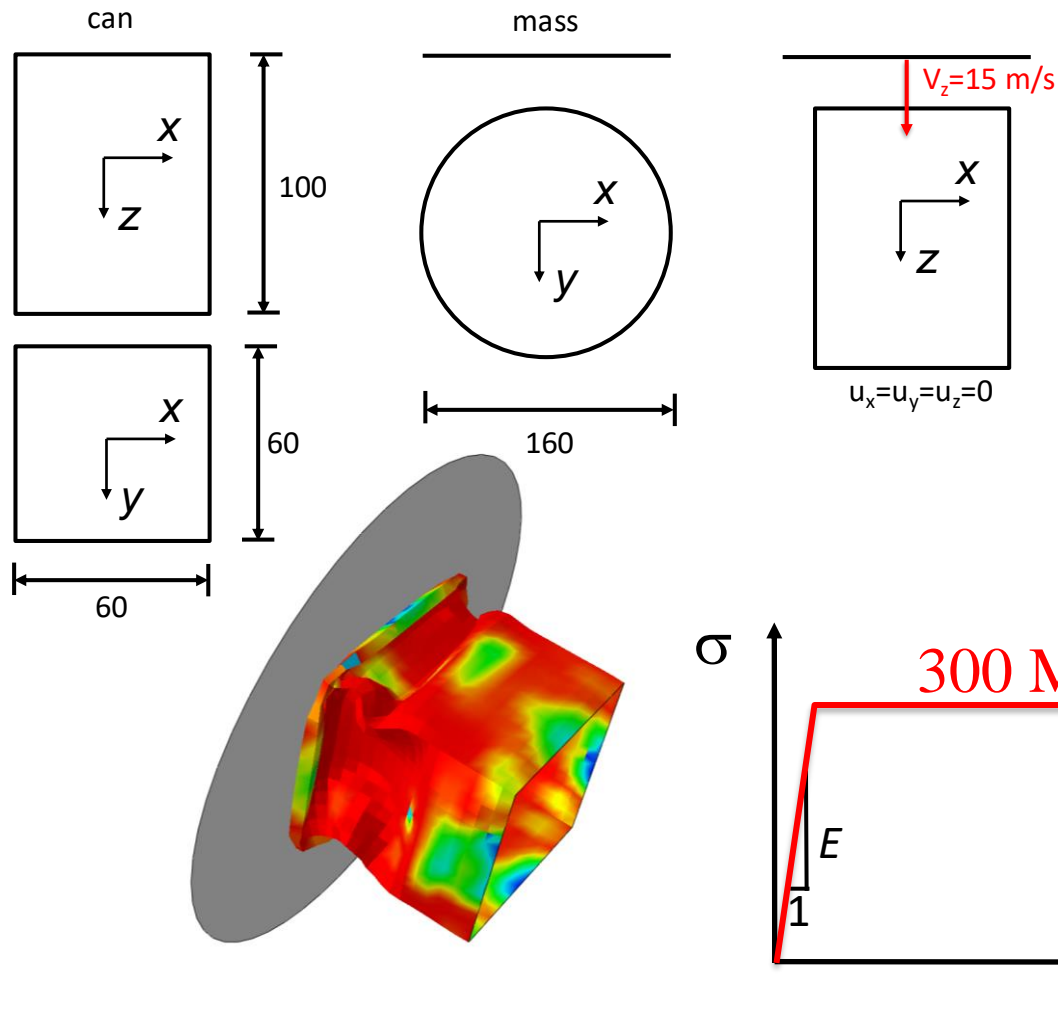
Consider the aluminum can in the figure impacted by a flat mass of **10 kg** at an initial velocity of **15 m/s**;

The can is assumed to be **homogeneous**, **isotropic** and **elastic-perfectly plastic**. The density of *Al* is **2.7 g/cm³** and the thickness of the can is **t = 1 mm**. A coefficient of friction of **c_f = 0.3** is assumed between the mass and the can.

σ

300 MPa

E

1

ε

Elastic properties:

Young's modulus *E* = 70 GPa

Poisson's ratio *v* = 0.3

Use ABAQUS/EXPLICIT to find:

a) the force vs displacement plot exerted by the impacting mass
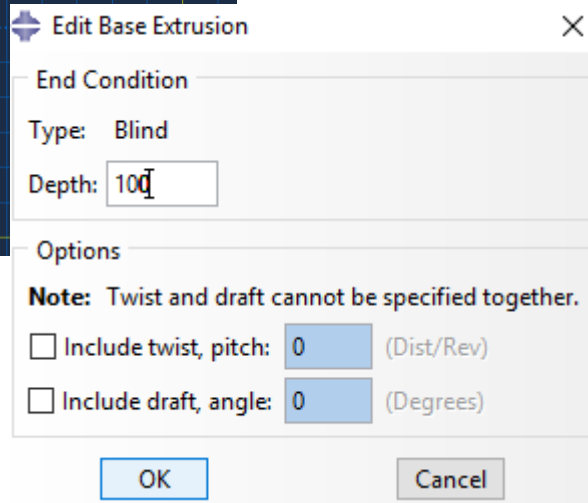
- This time we will create two parts: the can and the circular plate. Let's start with the can. **Double click on parts from the model tree as done in problems 1 and 2.**

- Specify the Part name, e.g. "Can", in the name field

- This time, we need to model the crashing of a structure in space ***let's select 3D in the modeling space card***;

- We want to study the deformation of the can and the energy dissipated during the crash. **Assign "deformable" in Type to allow the deformation of the can.**

- Different from the previous problems, now we have an additional option in "Shape", i.e. Solid. The solid option allows the construction of geometries which will be meshed by 3D elements (e.g. brick or hexahedral elements). This type of elements generally leads to a very high number of degrees of freedom of the structures which means large systems to be solved (i.e. need for significant computational resources). You should use this option if you expect the stress to be completely 3D and all the stress components need to be accounted for in the analysis. In the present case, the structure is very thin so modeling it using 2D elements in space is a reasonable assumption. **select shell**.

- We need to specify how we will create the structure. The plan is drawing the cross section and then construct the can by extrusion along the z-axis. **Select Extrusion then click Continue…**
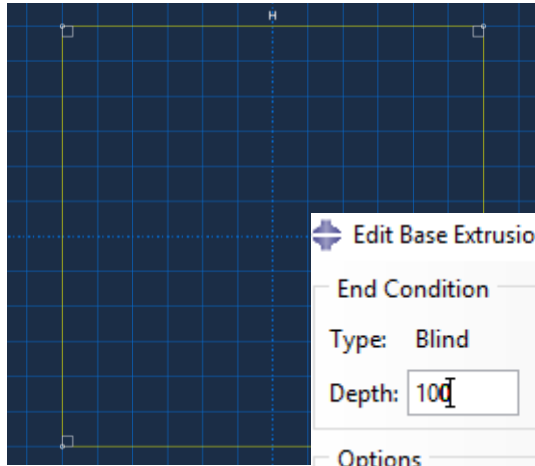
- **Sketch a 60 x 60 mm square centered at the origin using the same approach we used in problems 1 and 2**.

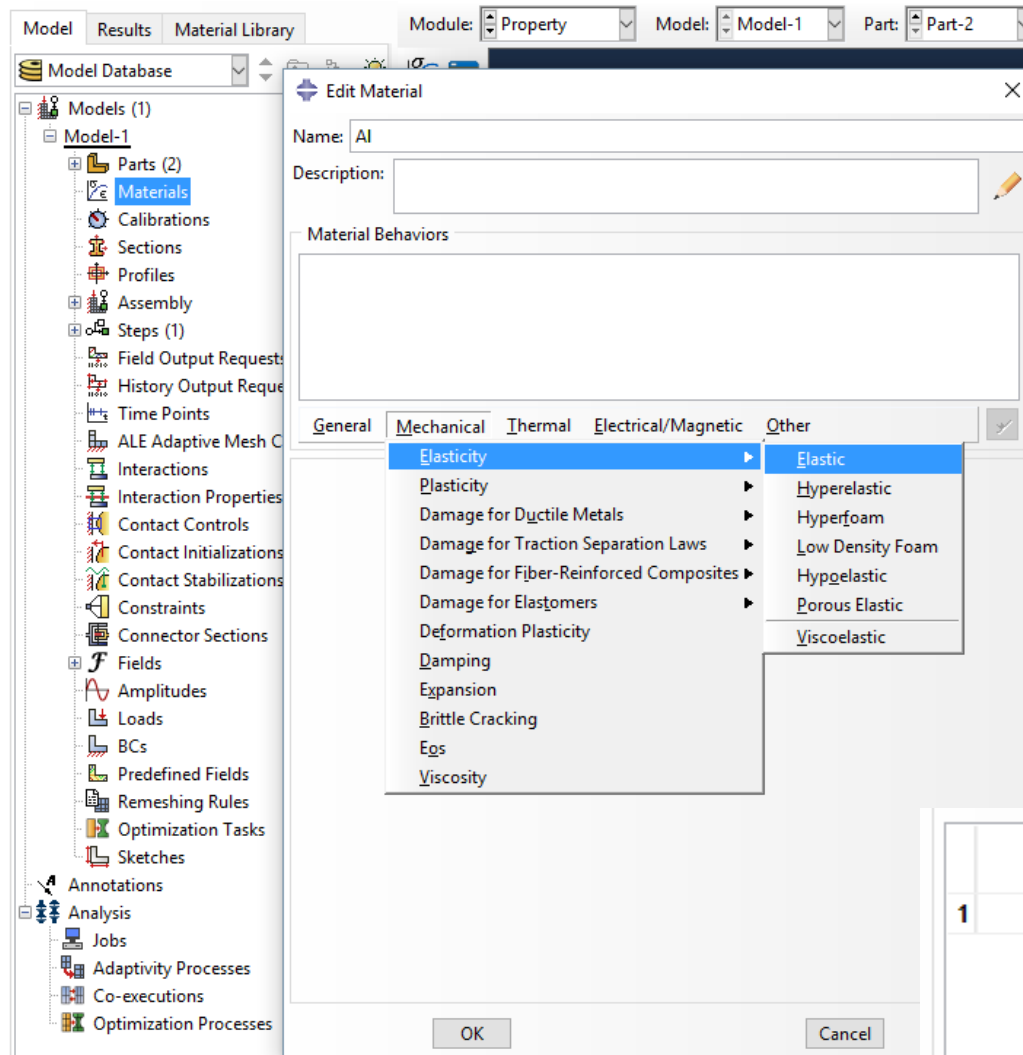- **Select Done at the bottom left** ← X Sketch the section for the shell extrusion Done

Edit Base Extrusion ✕

End Condition

Type: Blind

Depth: 100

Options

Note: Twist and draft cannot be specified together.

☐ Include twist, pitch: 0 (Dist/Rev)

☐ Include draft, angle: 0 (Degrees)

OK   Cancel

- We now need to indicate the depth of the extrusion. **Select 100.** Note that ABAQUS includes also the option of having e.g. a draft angle which can be useful to create tapered cans…

- **Press OK**. The part should now look as in the figure. Note that when using shell elements in 3D we generally model the middle plane of the structure.
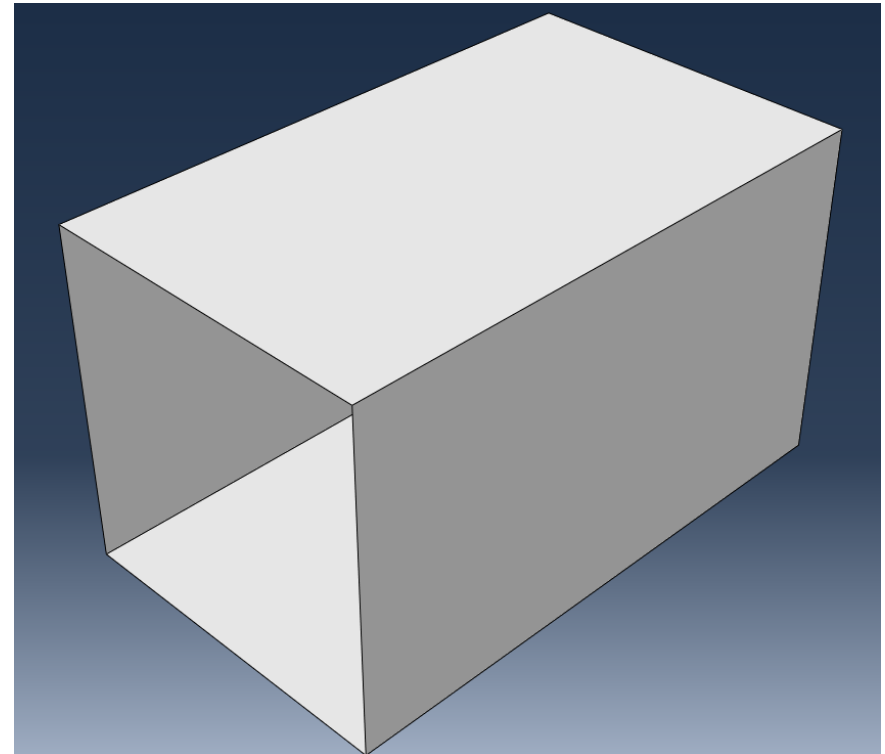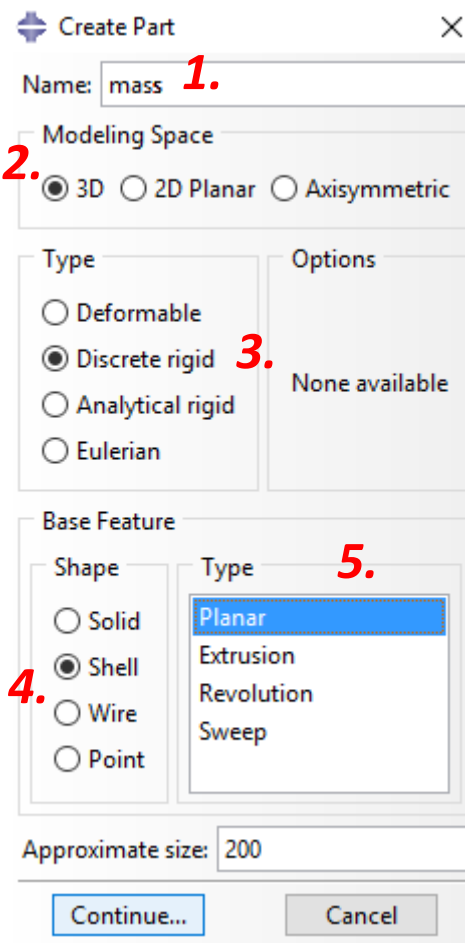
- Let's create the mass. **Double click on parts from the model tree as done in problems 1 and 2.**

- Specify the Part name, e.g. "mass", in the name field

- This time, we need to model the crashing of a structure in space **_let's select 3D in the modeling space card_**;

- We assume the mass is much stiffer than the can so we can neglect its deformation (this is often the case in this type of analysis. However, it is not always true. In the case of e.g. a projectile impacting a wall, the plastic deformation of the impactor is very important as well). **Assign "discrete rigid" in Type to create an rigid part.**

- **Select shell** as shape since we want to have a flat mass

- This time the part belongs to one plane only. **Select Planar then click Continue…**
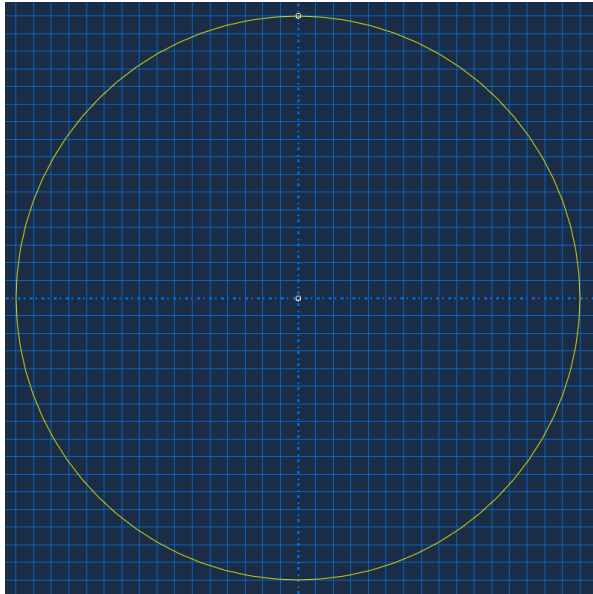
- **Sketch a circle centered at the origin with radius 80 mm using the same approach we used in problems 1 and 2**.

- **Select Done at the bottom left**

- The part should now look as in the figure.

- Now, we want to create a material for the can. ***Let's click twice on "Materials" in the model tree as we did in problems 1 and 2***.

- First, we need to specify the elastic properties of the material (this will characterize the first part of the elastic-perfectly plastic curve). **Click Mechanical/Elastic and assign a Young's modulus of 70000 and a Poisson's ratio of 0.3**

Edit Material

Name: Al

Description:

Material Behaviors

Elastic
Plastic

General   Mechanical   Thermal   Electrical/Magnetic   Other

Plastic

Hardening: Isotropic   ▼ Suboptions

☐ Use strain-rate-dependent data
☐ Use temperature-dependent data

Number of field variables: 0

Data

	Yield Stress	Plastic Strain
1	300	0
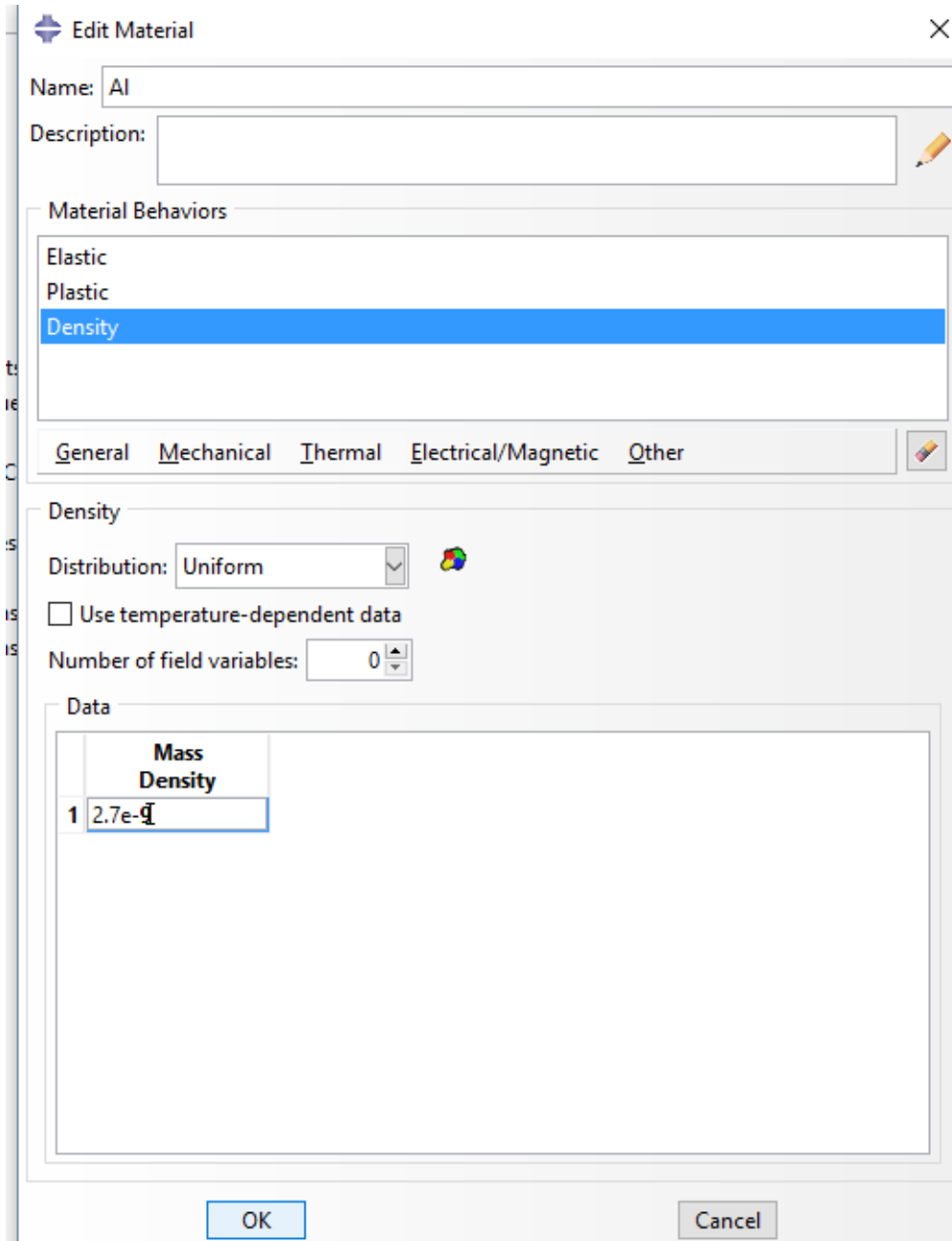
OK   Cancel

- Then, we need to specify the plastic behavior. In this case the stress remains equal to the yielding stress during plastic deformation. **Click Mechanical/Plastic and assign a Yield stress of 300 MPa.** Since we assume a perfectly elastic behavior prior to yielding, the plastic strain at yielding is zero. **Input 0 as Plastic strain**

At yielding, the material unloads to zero strain (i.e. no permanent deformation)

σ

$\varepsilon_{\text{plastic}} = 0$

ε
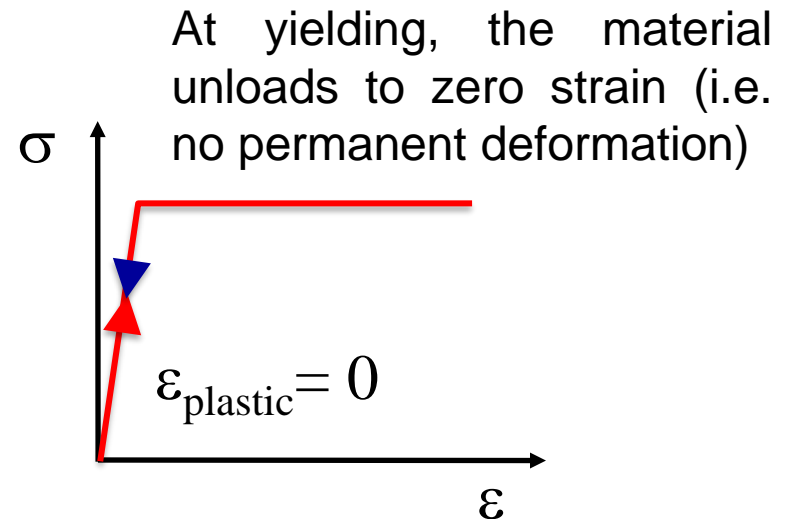
- Note that ABAQUS can handle very complicated plastic behaviors. One can input the whole stress – strain curve measured in the lab directly in the data fields. ABAQUS will interpolate the stress/strain curve accordingly.

Problem 3: Material creation (3)


- Finally, let's not forget that we are going to run a dynamic simulation. Accordingly we need to specify the density of the material to calculate the mass matrix. **Click General/density and assign a density of 2.7 10^{-9} ton/mm³.** Note that when working in N, mm and s as units for forces, length and time, ABAQUS uses tons to specify the mass.

- **Press OK.**

- We do not need to create a material for the impacting mass since we are going to model it as a rigid body and there are no deformations. However, we still need to assign some inertia properties. Since the body is rigid, its dynamics can be described by assigning a concentrated mass and three inertias. To do so, we need to assign a Reference point to the part.
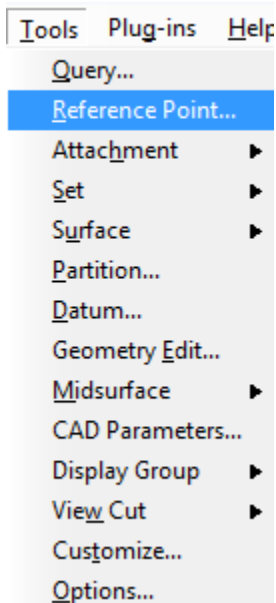
- **Go to Tools/Reference Point**
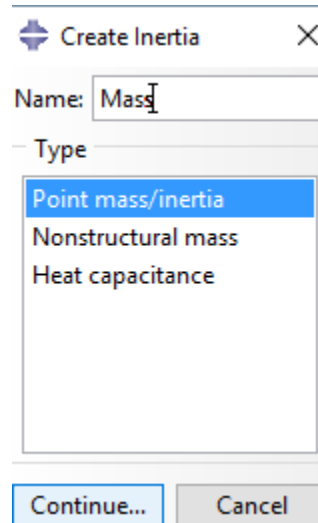
- **Select the center point of the circle then press OK**
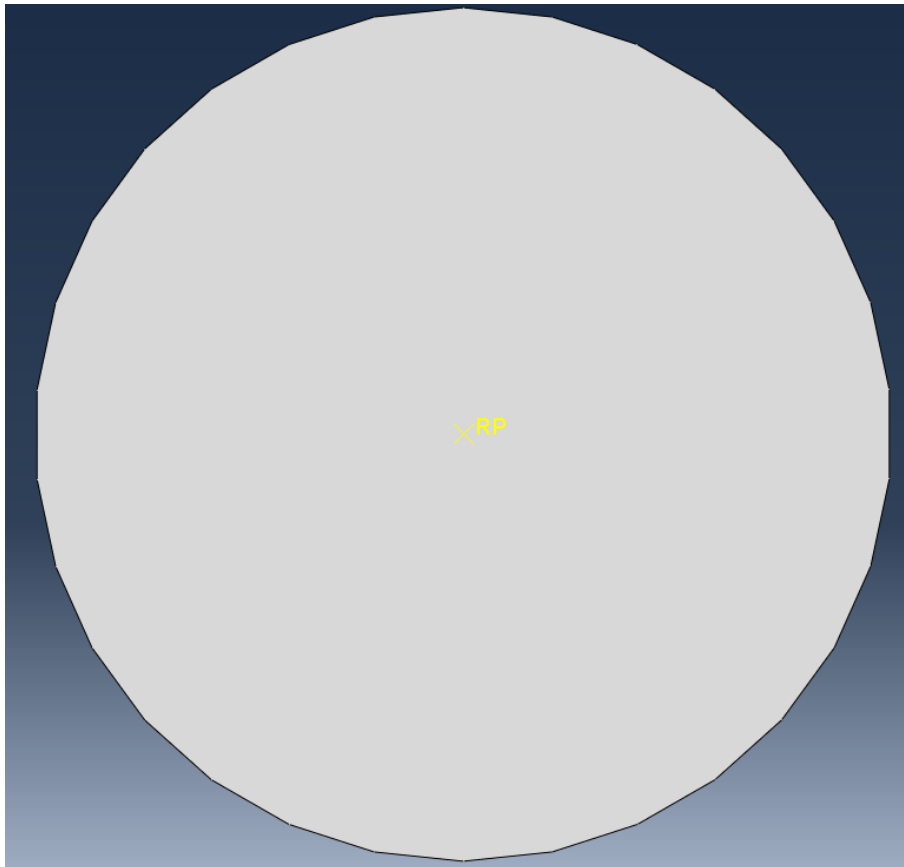
- **A cross with the label "RP" should appear at the center point indicating that the reference point has been created.** All the inertia properties of the mass will be ascribed to this point.
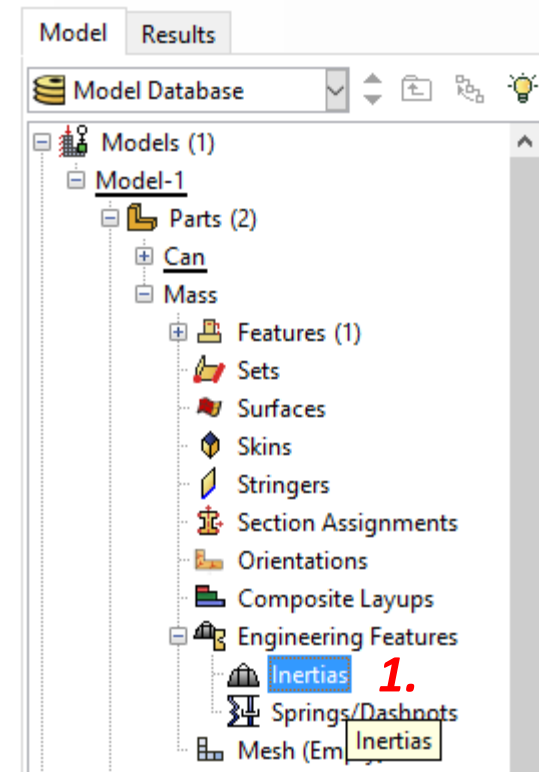
- Let's assign the mass. **Expand the part "Mass" in the model tree. Expand "Engineering Features" then click on Inertias**

- **Assign a name to the mass (e.g. Mass).**

- **Select Point mass/inertia** since we want to assign the inertia properties to a point. **Then, click on Continue…**

- **Select the Reference Point and click Done at the bottom left**
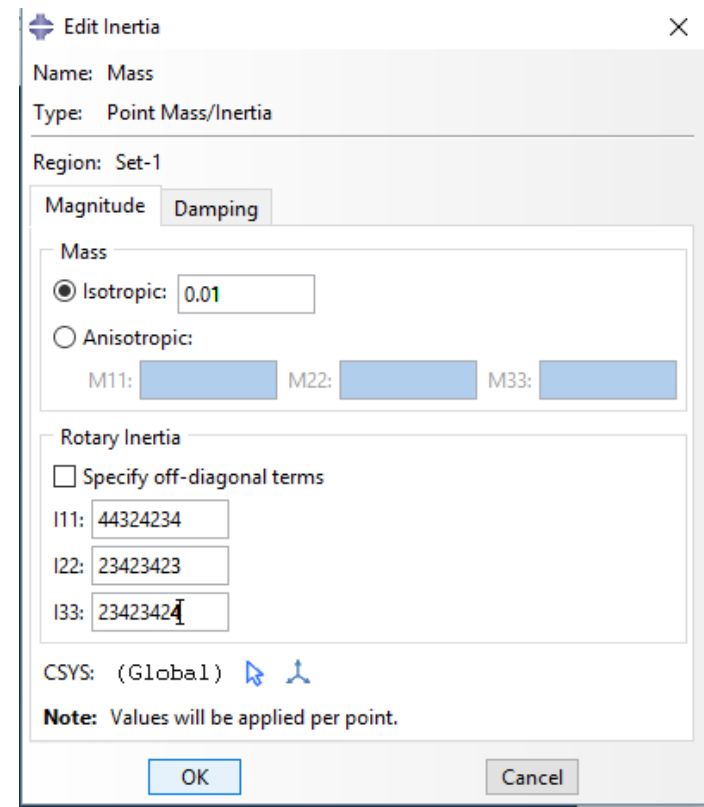
- **Assign a concentrated mass of 10 kg to the RP.** Note that the mass is assigned in tons

- **In our simulation we will fix all the rotations of the mass so the inertias are not significant. Just input some numbers.** Note that if the rotations are not constrained then it is necessary to input the correct inertias to obtain an accurate simulation.

- **Press OK.** A green dot will appear on the RP
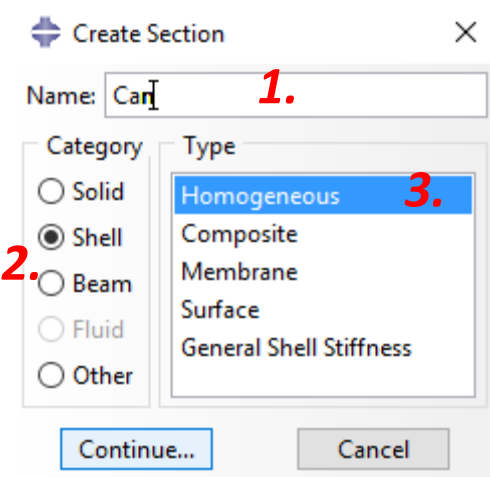
- **Following the usual procedure create a new section. Name it Can. Select shell as category** since we want to model the structure using 2D shell elements.

- **Select Homogeneous and press Continue...**

- **In the edit section card let's assign a thickness of 1 mm**

- **Select the material Al as the material related to the section**
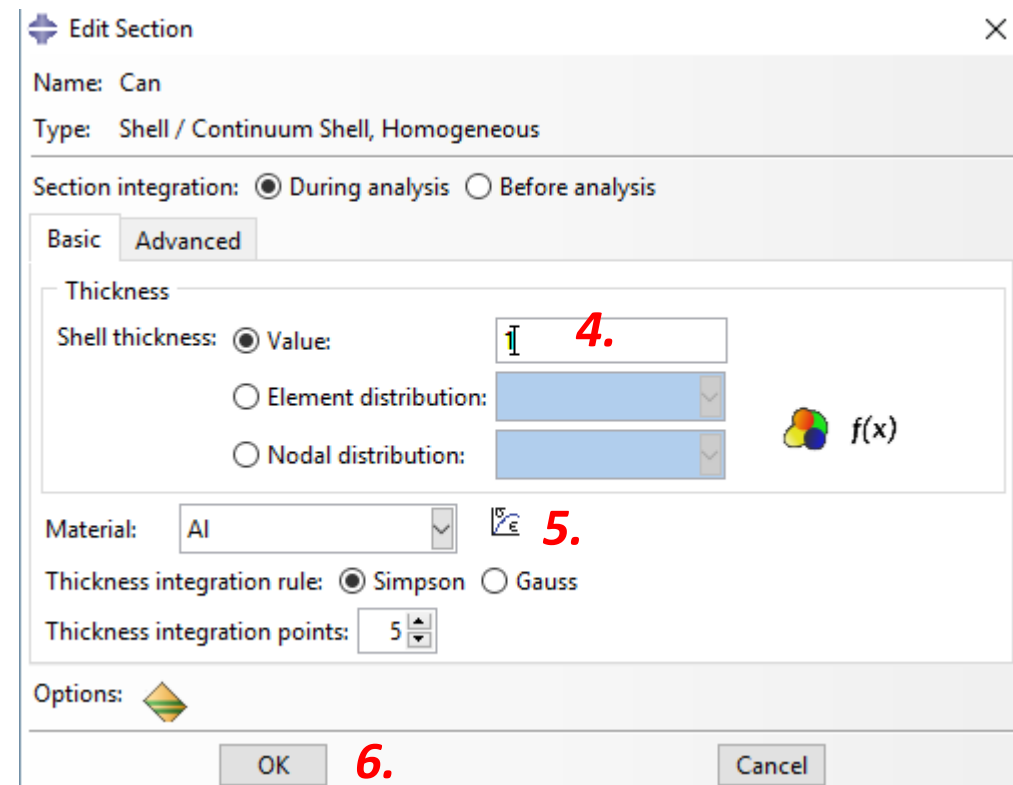
- **Keep the default values regarding the through-the-thickness integration.** By default ABAQUS uses 5 points and Simpson integration. (Always use at least 3 integration points)

- **Press OK**
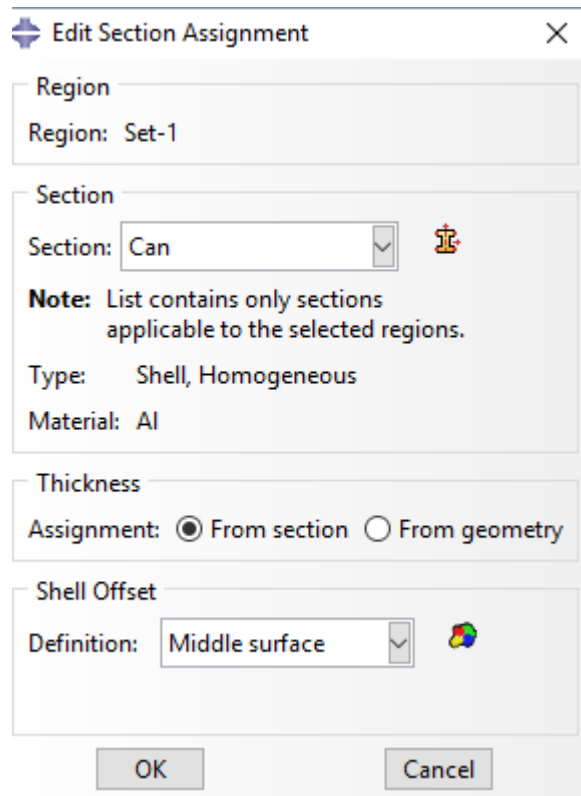
- **Click on the assign section icon**

- **Select "Can" as section**

- By default, the surface constructed to represent the part is the middle surface. This option can be changed to top or bottom surface if you prefer. Generally it is easier to use the middle surface but in some cases, e.g. to describe contact between surfaces, using top or bottom surfaces is more intuitive. In our case, **_we are using the middle surface so select this option in the Shell Offset field_**.
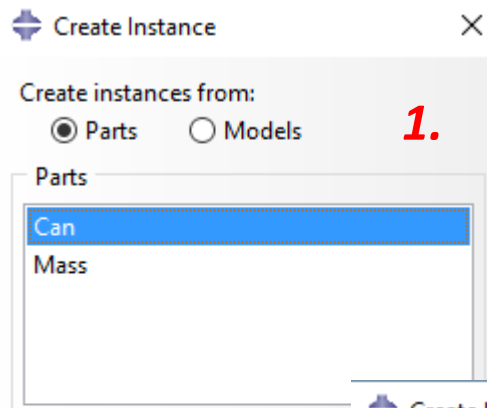
- **Press OK**

- **The part should now be green showing that the section has been assigned.** (Note that we need to apply the section to the can only. The mass does not need any section, being it a rigid body).
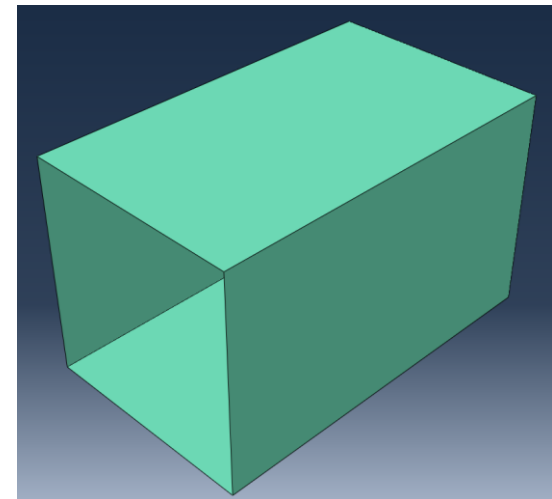
- After creating the parts, we need to create an assembly. This time we have 2 parts so we will need to include 2 instances and then define their interaction
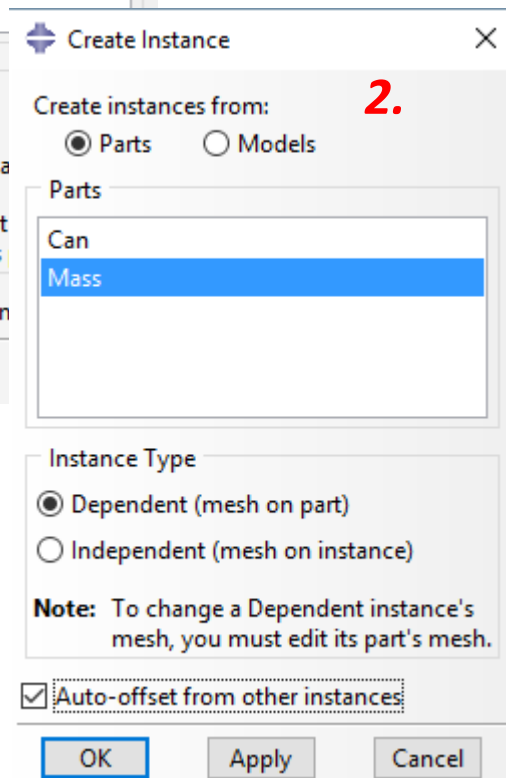

1.

- ***Click on the insert instance icon as done in problems 1 and 2***

- ***Select Can as the part to be inserted***

- ***Keep the option mesh on part*** (we do not expect to model cracks in this simulation so we do not need an independent mesh)

- ***Press Apply.*** This will insert the part in the assembly.

2.

- ***Now, select mass as part to be inserted***

- ***Keep the mesh-on-part option***

- ***Select the "auto-offset" option***. This will make sure the new instance will not overlap with the can

- ***Click OK***

- The assembly should look as in the figure:
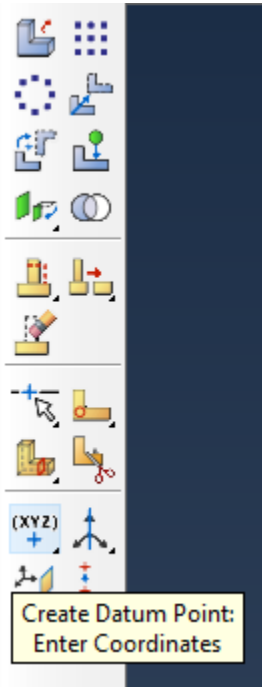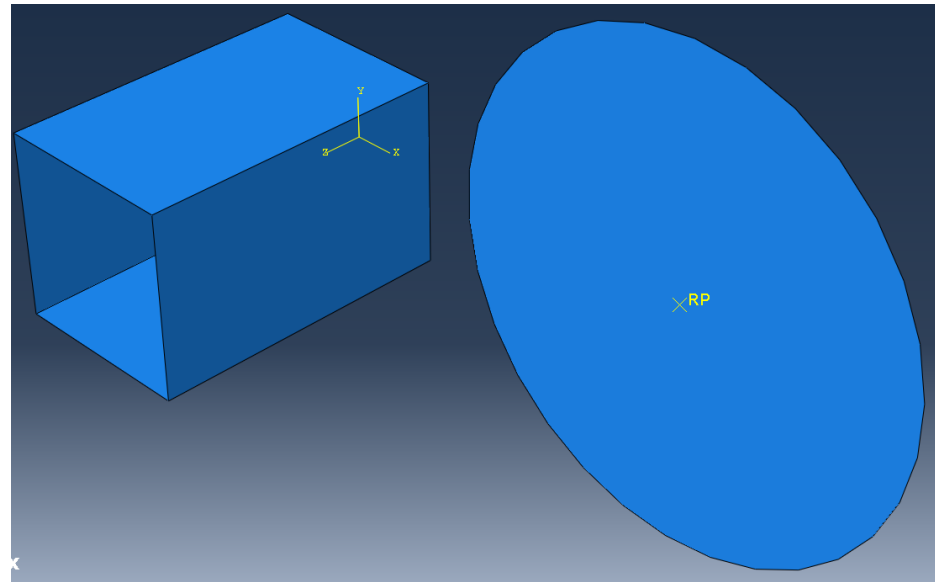
- The can axis is aligned with the z-axis. Let's keep the can where it is and let's move the impacting mass. The mass is already parallel to the x-y plane so first we just need to translate it such that its centroid belongs to the axis of the can. To help us with the translation, ***let's create a point at the center of the can cross section***. ***Click on the icon "Create Datum Point" from the module bar and hold the left button of the mouse***
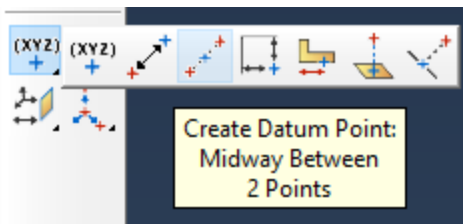
- A new window will show up with additional options, ***Select Create Datum Point: Midway between 2 points.*** This will allow the creation of a midpoint between two points selected in ABAQUS
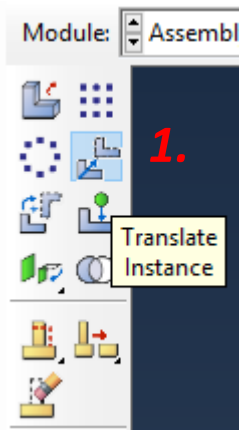
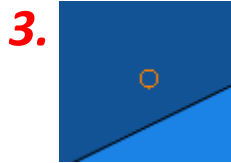- **<u>Select the nodes indicated in the figure</u>**:
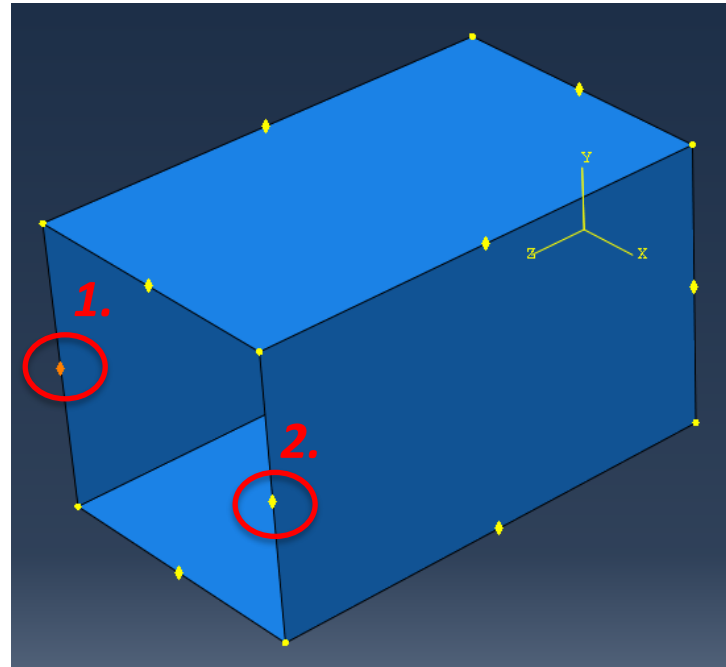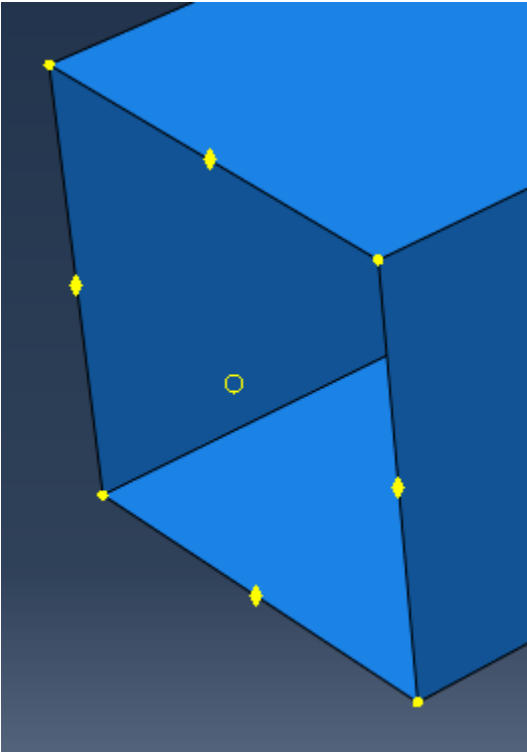
- A new point should appear at the center of the cross section. We will use it to facilitate the translation of the impacting mass.
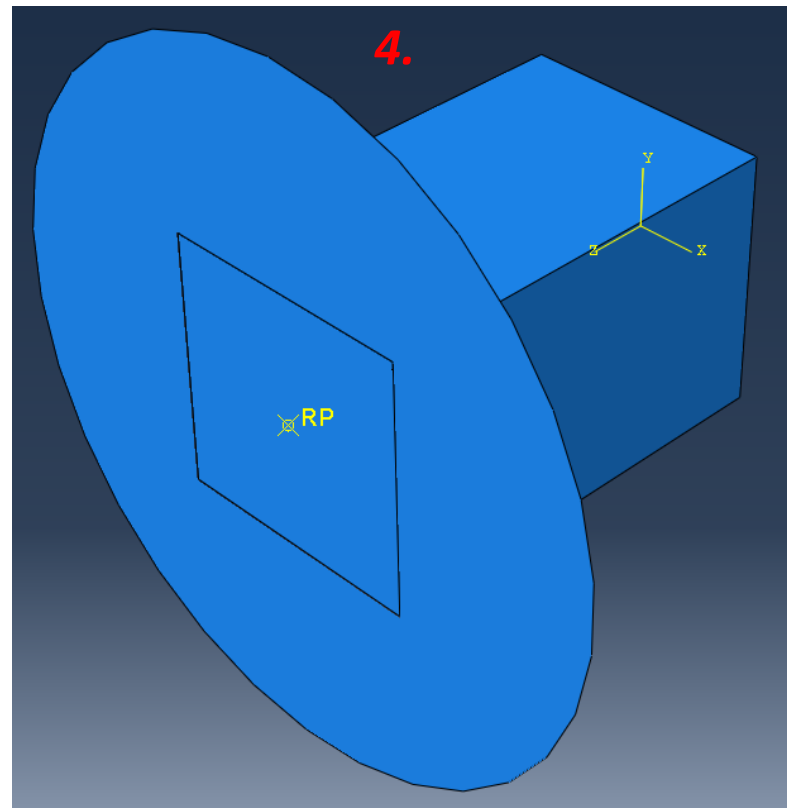
Problem 3: Assembly (4)

Module: Assembl

1.

Translate Instance

- ***Select the "Translate Instance" icon from the module bar***
- ***Select the impacting mass as the instance to be translated***

2.

RP

- ***Select the Reference Point as the starting point of the translation***

3.

- ***Select the point created in the previous step as end point of the translation then press OK at the bottom left***. The assembly should look as below

4.

- After the translation, the two parts are in contact. As a general rule, you should always avoid having parts in contact at the beginning of the crash simulation. This could lead to convergence problems in the first step. Accordingly, we need to slightly move the impacting mass in the positive z direction

- ***Select the "Translate Instance" icon and select the mass as done before. Then press Done at the bottom left.***

- This time we want to specify the displacement vector using Cartesian coordinates. This can be done in ABAQUS specifying the location of the tail of the vector (start point) and the location of the head of the vector (end point). ABAQUS will automatically calculate the magnitude, direction and sense of the displacement vector and it will apply it to the instance. ***Input 0,0,0 as start point at the bottom left and press Enter. Then, input 0,0,-0.5 as end point. This will create a vector directed along the z-axis, pointing in the positive direction and with a magnitude of 0.5 mm. Press Enter. Then, press OK at the bottom left.*** The assembly should look as below. Note that now the mass and the can are not in contact. (note that the distance between the mass and the can should be as limited as possible. The larger this distance the longer will take to have the first contact, i.e. higher number of calculations required)
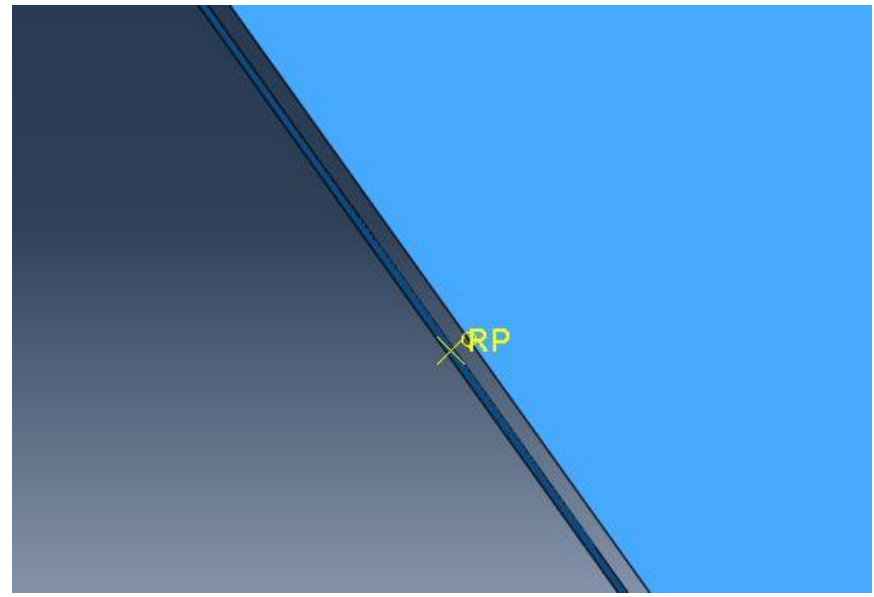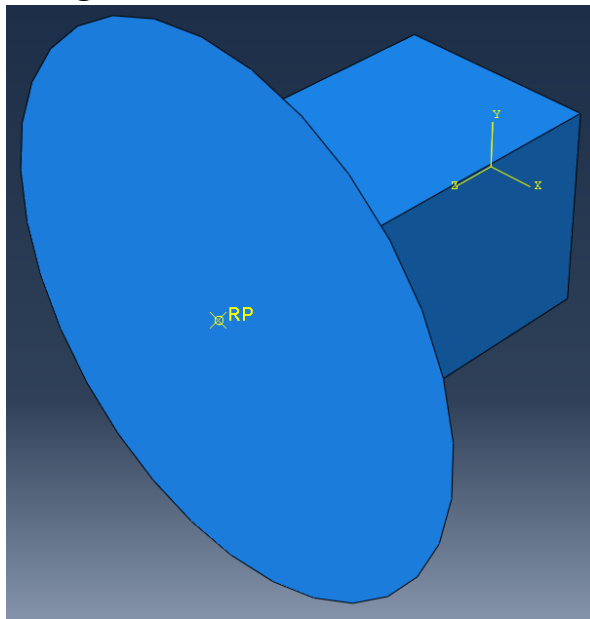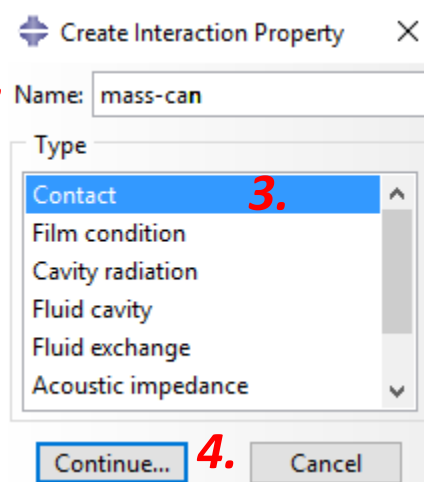
- We need to define a contact interaction between the can and the mass to simulate the crashing. Modeling contact accurately is still an open subject of research and the literature abounds of procedures for contact. Also ABAQUS has more than one contact algorithm but, in this example, we will only use the **GENERAL EXPLICIT CONTACT ALGORITHM WITH PENALTY STIFFNESS**.

$$F = k\Delta$$

k

Δ
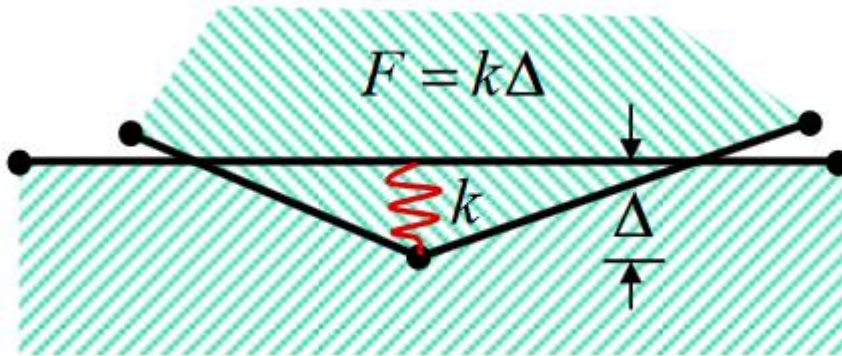
- The penalty stiffness algorithm hinders surface interpenetration by introducing springs between elements at the surface. The stiffness of the spring is sufficiently high to simulate contact but is not infinite since this would create problems for the stability of the solution.
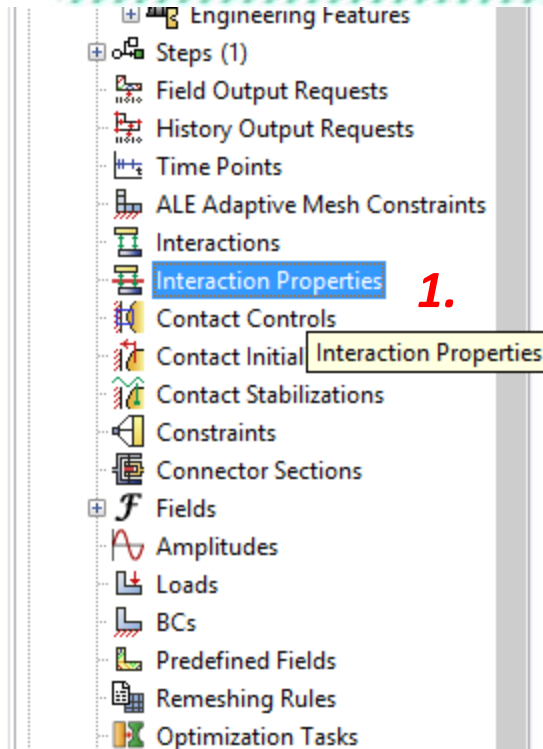
- First, let's assign some properties to the contact interaction. ***Click on "Interaction properties" in the model tree***

Model tree:
- Engineering Features
- Steps (1)
- Field Output Requests
- History Output Requests
- Time Points
- ALE Adaptive Mesh Constraints
- Interactions
- Interaction Properties **1.**
- Contact Controls
- Contact Initial | Interaction Properties
- Contact Stabilizations
- Constraints
- Connector Sections
- *F* Fields
- Amplitudes
- Loads
- BCs
- Predefined Fields
- Remeshing Rules
- Optimization Tasks

Create Interaction Property ✕

2. Name: mass-can

Type
- Contact **3.**
- Film condition
- Cavity radiation
- Fluid cavity
- Fluid exchange
- Acoustic impedance

Continue... **4.**   Cancel

- ***Name the interaction (e.g. mass-can)***

- ***As Interaction type, select Contact and press Continue…***

- We need to specify the main properties of the contact interaction. ***Select Mechanical/Tangential behavior***

- ***Select Penalty in the "friction formulation field***. This will assign the penalty stiffness algorithm for the contact.

- Let's assign the coefficient of friction to the surface. ***Input 0.3***

- ***Press OK***

- After creating the contact properties, let's now assign the contact interaction to the desired properties. In crash simulations is very likely that several surfaces will get into contact. A safe assumption is that all the surfaces of the model may experience contact. We want to use this assumption in the simulation.
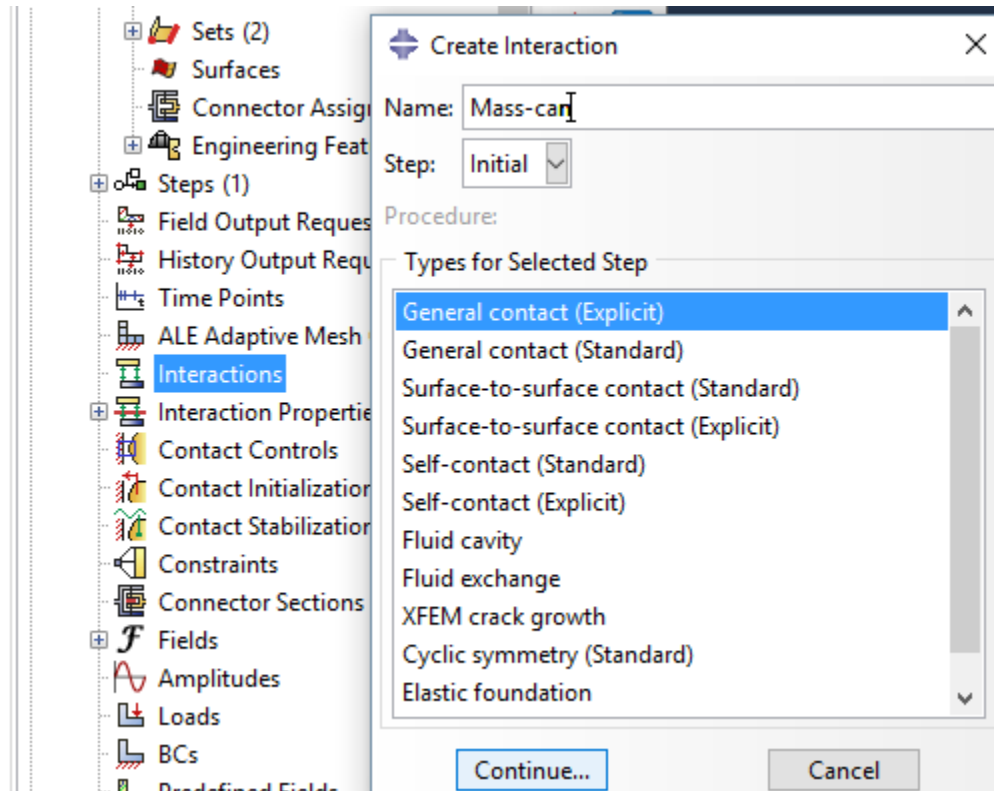
- ***Select "Interaction" from the model tree.***
- ***Name the interaction (e.g. Mass-Can)***
- ***Select "General Contact (Explicit)"*** to select the general contact formulation provided by ABAQUS for explicit simulations

- We want to assign contact to all the surfaces. ***Select "ALL with self in the Contact domain.***

- We need to specify a property for the contact analysis. ***Select the previously created interaction property (e.g. mass-can) in the "Global property assignement" field.***

- ***Press OK.***

- Let's model the can using 2D shell elements with full integration. **_Click on the icon_** [icon] **_then select the can_**

- **_Select "Explicit" element library_** since we are going to run an explicit simulation

- **_Select Shell as family of elements_**

- **_Untick the option "Reduce Integration"_**. This will guarantee the adoption the highest order shape functions

- **_Keep the default values in the element controls_** (we will cover the element controls in future analyses)
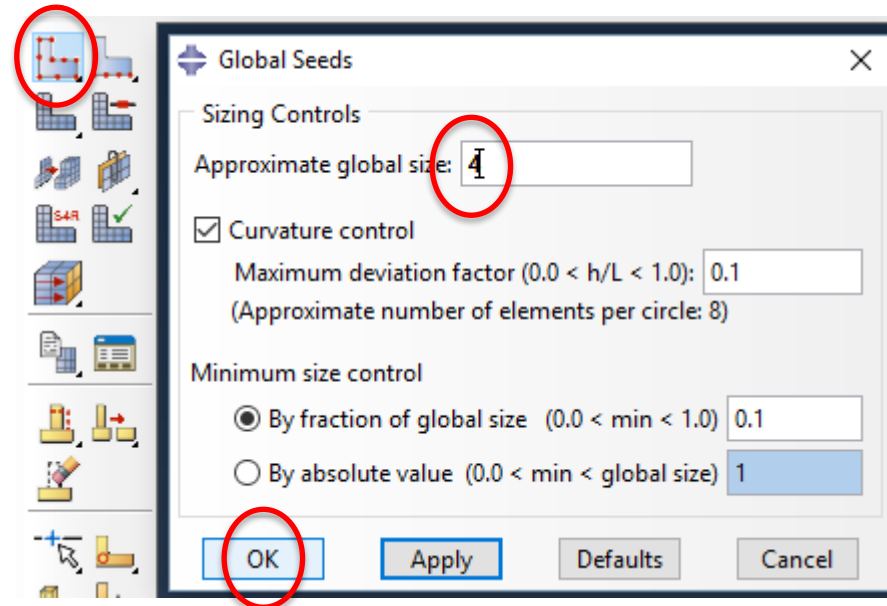
- **_Press OK._**

- ***Assign a structured mesh of quadrilateral elements*** only following the same procedure used for problems 1 and 2.
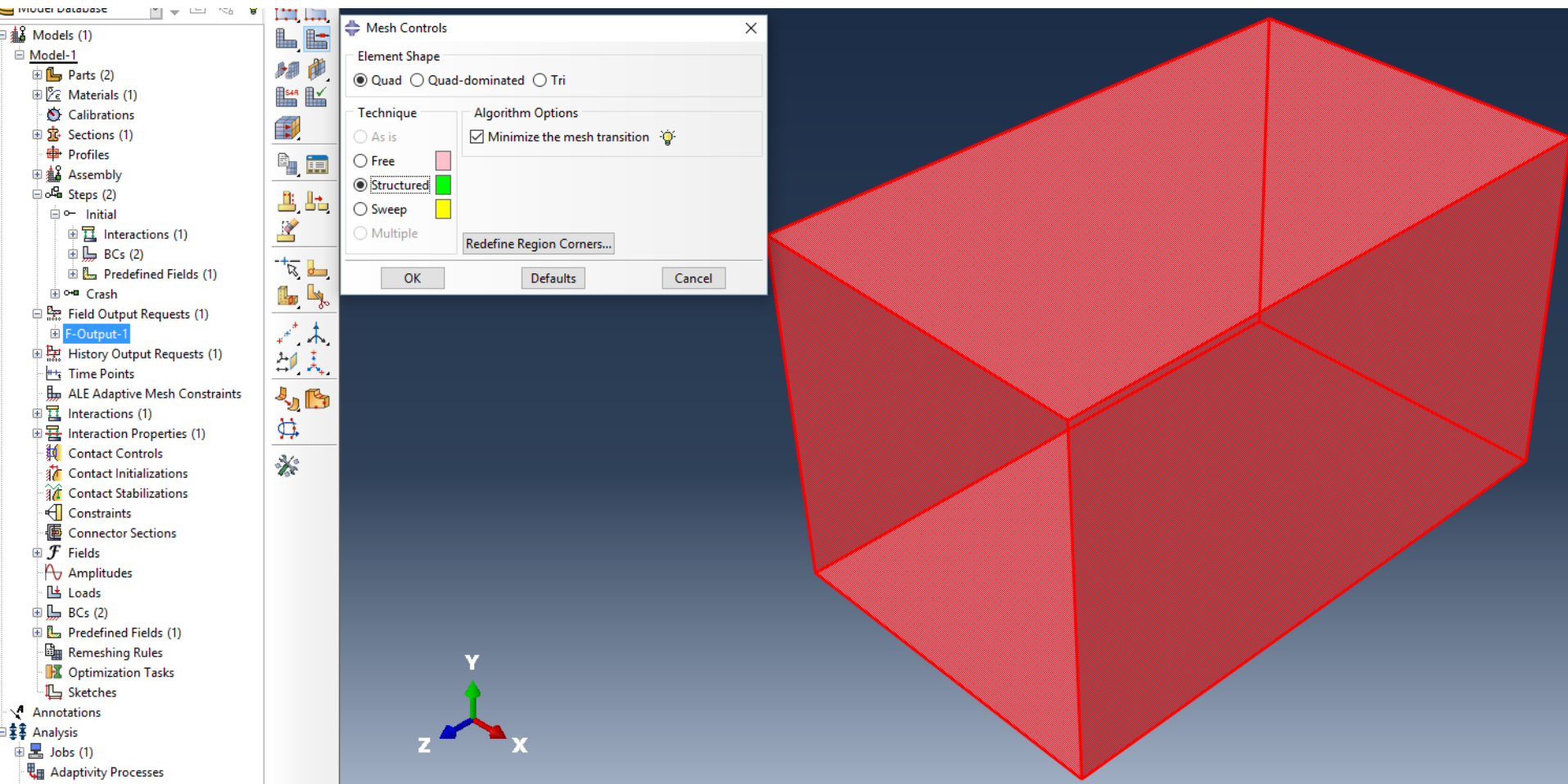
- We want to use a coarse mesh and we do not have particular stress concentrators in the structure. Accordingly, we can use a very uniform mesh. Instead of applying different seeding to each line as we did in problems 1 and 2, **_let's assign a uniform global seeding of 4 mm to all the lines_**.

- **_Click on the "Global seeding" icon from the module bar_**

- **_Select 4 mm as global element size_**

- **_Press OK. Then, click on the mesh icon_**

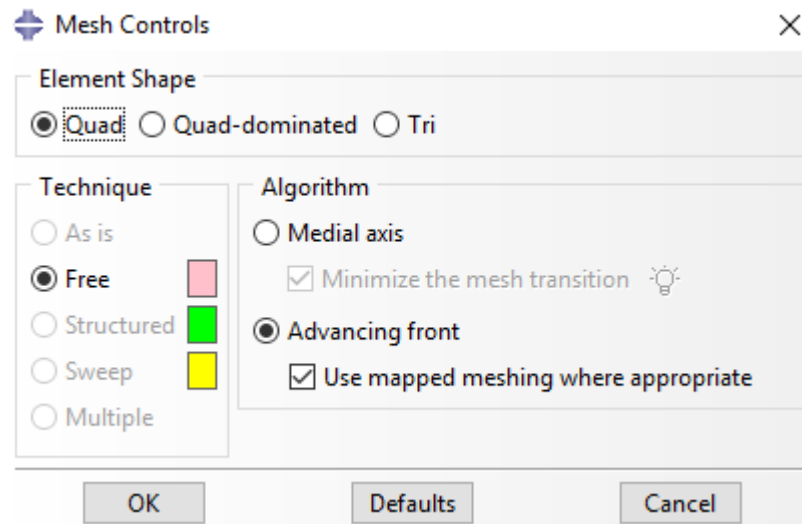- **_Press Yes on the bottom left._** Your can should look as below.
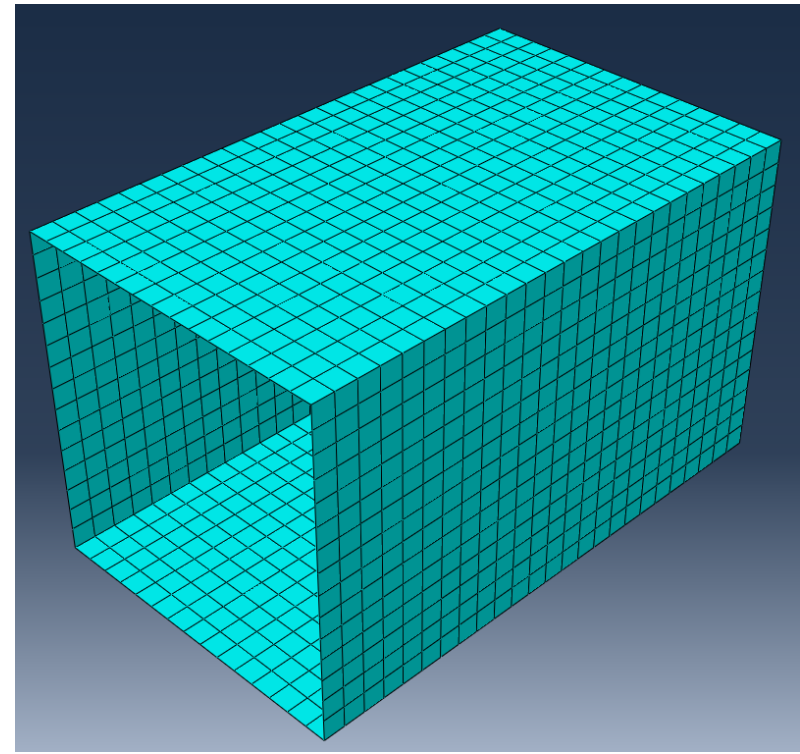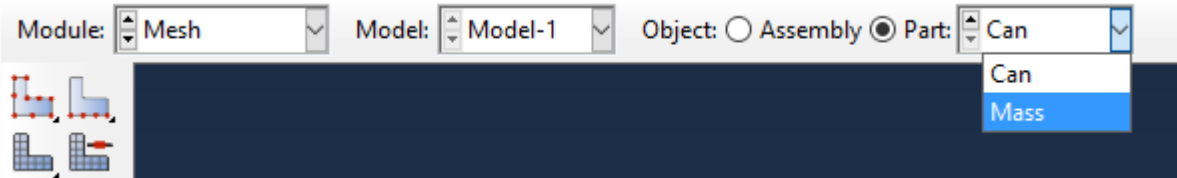
- Let's mesh the mass. ***Select the part "Mass" using the appropriate field as shown***

- Since in this case the elements are rigid, we do not need to take particular care about what type of shape functions will be used etc. ***Let's skip the element type assignment***
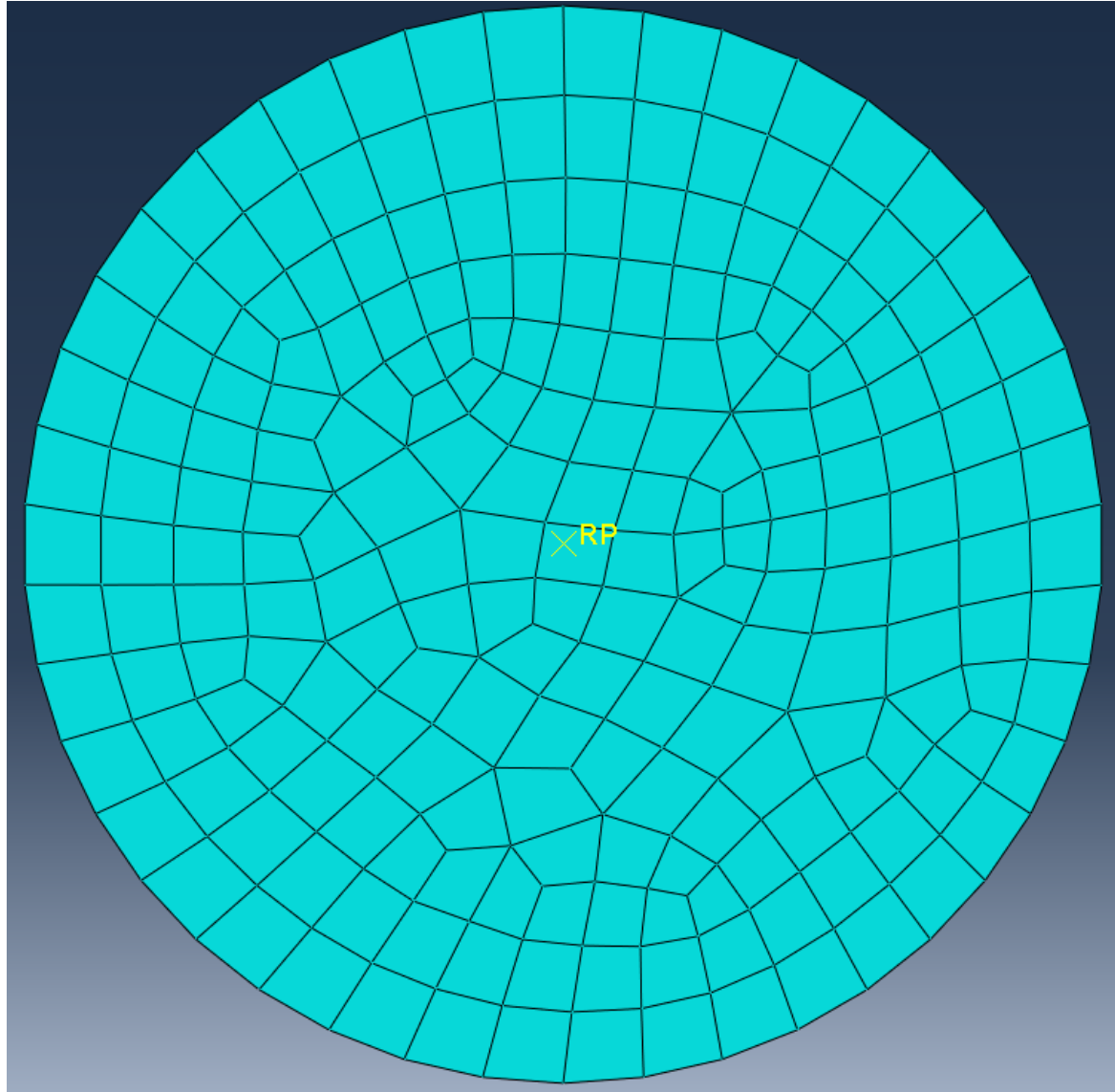

Assign a free mesh, quad dominated as done in previous problems
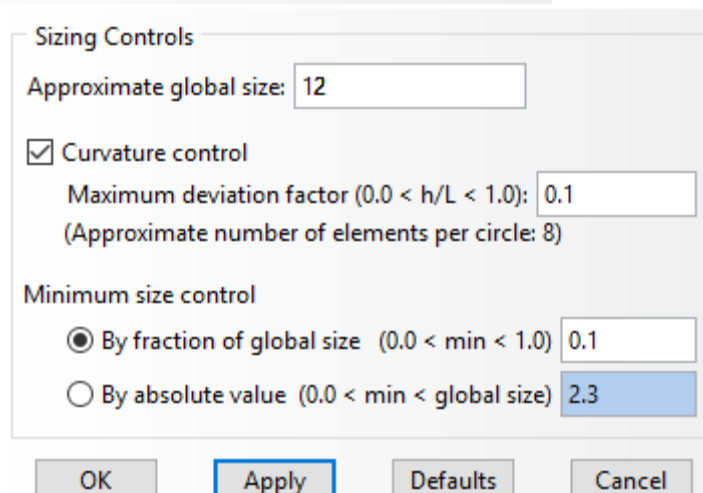
- ***Let's apply a global element size of 12 mm then press OK and mesh the part.*** Since the elements are rigid, we do not need a fine mesh. At the same time, the rigid elements should not be too much larger than the elements of the can to avoid convergence issues in the contact algorithm.

- The mass should look as shown

- We are ready to create the step. ***Click twice on Steps in the model tree*** (as done for problems 1 and 2)

1.

Create Step ✕

Name: Crash

Insert new step after

Initial

Procedure type: General

Dynamic, Implicit
Dynamic, Explicit
Dynamic, Temp-disp, Explicit
Geostatic
Heat transfer
Mass diffusion
Soils

Continue... | Cancel

- ***Name the step (e.g. Crash)***

- ***Select when you would like to add the step (in this case we only have the initial step so…just select initial).***

- ***Select Dynamic, Explicit*** to assign the dynamic, explicit solver to the model

- ***Click on Continue…***

2.

Edit Step ✕

Name: Crash

Type: Dynamic, Explicit

Basic | Incrementation | Mass scaling | Other

Description: |

Time period: 0.01

Nlgeom: ○ Off (This setting controls the inclusion of nonlinear effects
   ◉ On   of large displacements and affects subsequent steps.)

☐ Include adiabatic heating effects

- In the edit step card, we need to assign a total time for the simulation. For a given stable time step, the longer the total time the higher the number of computations so we want to keep the total time as low as possible.

Assuming a constant impact velocity of 15 m/s during the crash, the crash would take ~ 100/15000=0.007 s. Of course, this is a lower bound estimate since the velocity of the mass will decrease during the crash. ***Let's use 0.01 s as Time period. Then, press OK.***
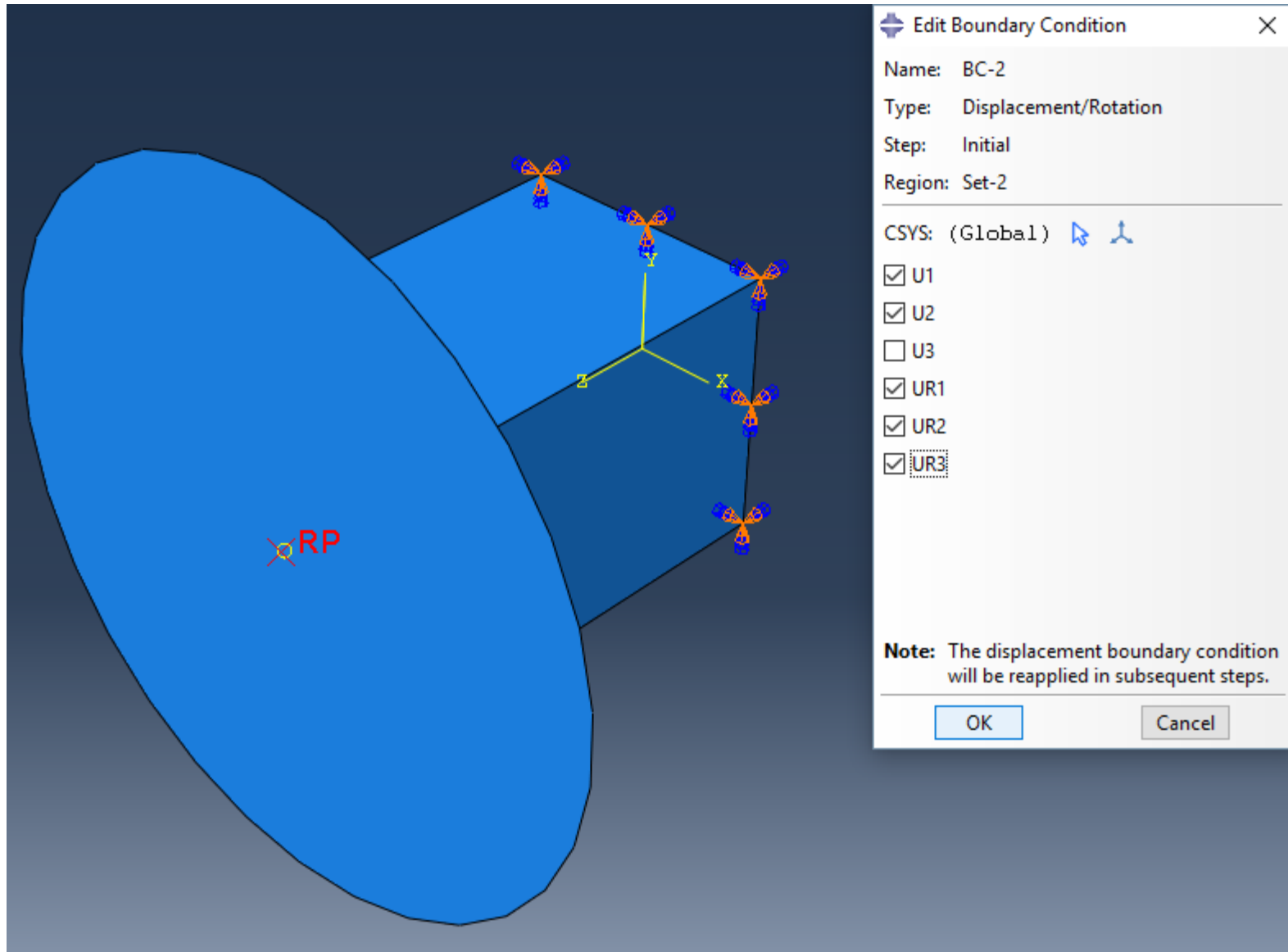
- Let's assign the initial BCs. Following the same procedure as problems 1 and 2, **_put all the dof of the right end of the can equal to zero ($u_1=u_2=u_3=u_{r1}=u_{r2}=u_{r3}=0$) (see figure)_**
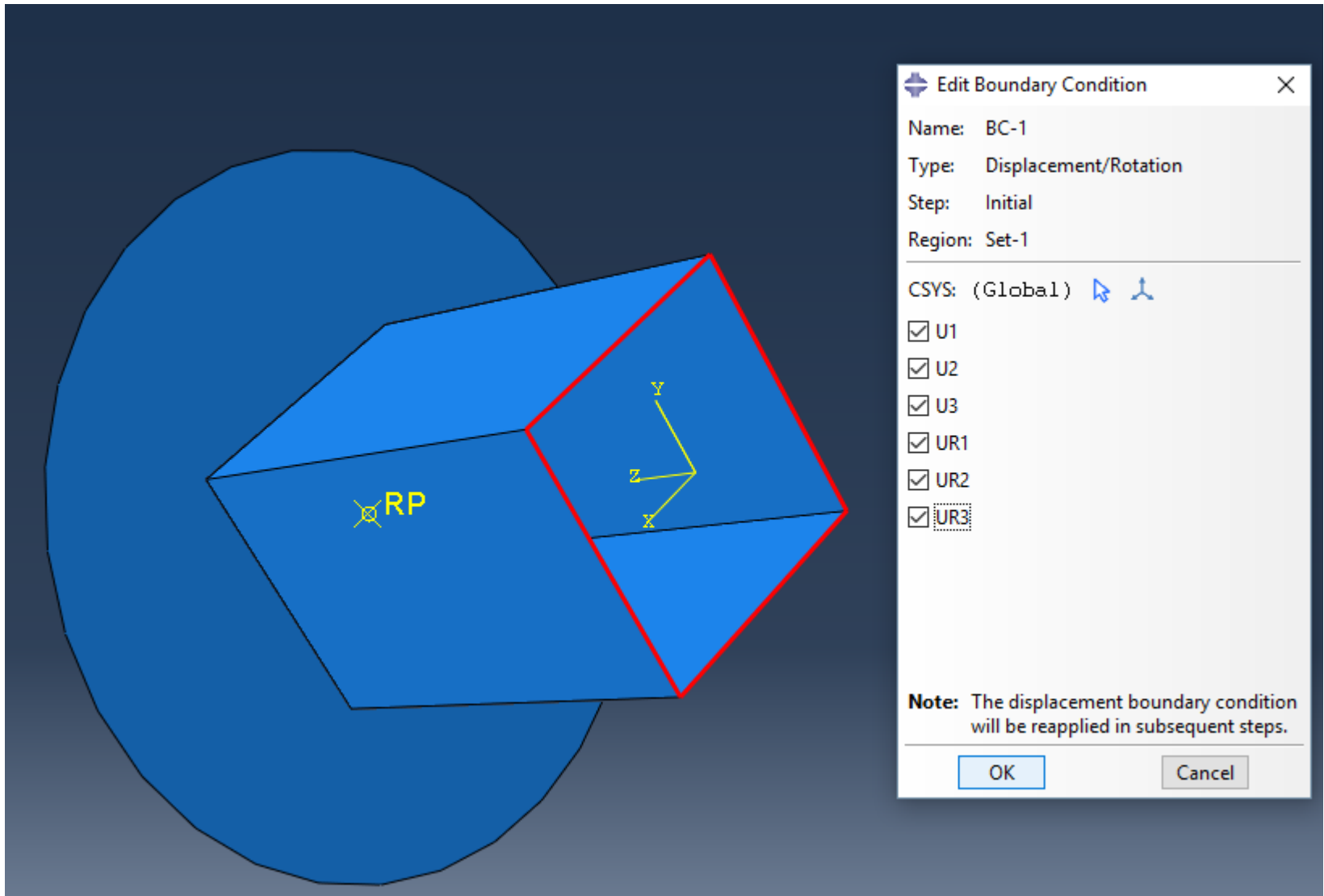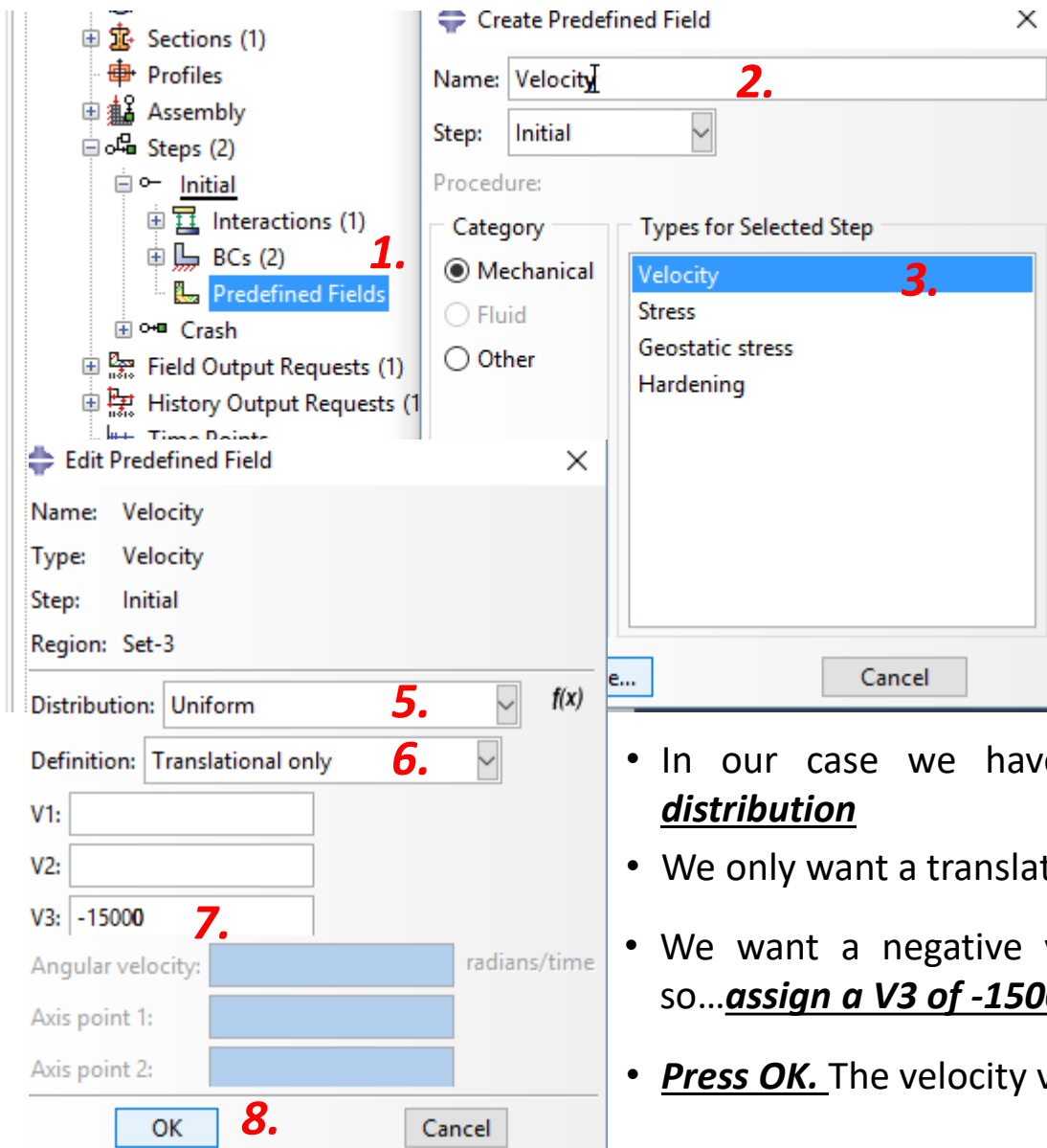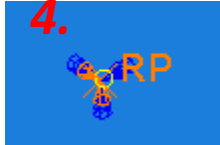
- We do not want any rotation of the mass or lateral translation so... *__assing $u_1=u_2=u_{r1}=u_{r2}=u_{r3}=0$ at the reference point (see figure)__*

- Finally, we need to assign an initial velocity to the mass. To do so, ***expand the initial step in the model tree and click twice on "predefined fields".***

- ***Assign a name to the field (e.g. Velocity)***

- ***Select mechanical as category and velocity as type.***

- ***Press Continue...***

  - ***Select the point on which we want to assign the velocity (the RP in our case)***

- ***Click Done at the bottom left***

- In our case we have a point velocity so...***keep uniform as distribution***

- We only want a translational velocity so...***keep translational only***

- We want a negative velocity of -15000 mm/s along the z-axis so...***assign a V3 of -15000***

- ***Press OK.*** The velocity vector should now appear on the RP
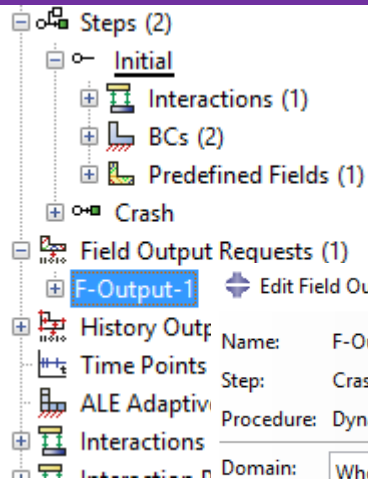
*...efore creating the job for the analysis we need to make sure ABAQUS will store all ...he results we need at the time interval we desire. **Expand "Field Output Requests" ...n the model tree and click twice on "F-Output-1".***

- ABAQUS wants to know how frequently it should save the results. **_Keep the option Evenly spaced time intervals in the frequency field and Insert 200 in the interval fields._**

- ABAQUS will subdivide the total time into 200 intervals and save the results at the end of each interval. *You can increase the number of intervals as desired but mind that the size of the ODB file will increase accordingly!*

- In the output variable field, ABAQUS shows the variables which will be saved. You are free to add more variables but remember that this will increase the size of the result file. **_Keep the default values for the current simulation and press OK._**

- As you learned in problems 1 and 2, ***Go into Analysis in the model tree and create a Job called "Crash"***

Edit Job ✕

Name: Crash

Model: Model-1

Analysis product: Abaqus/Explicit

Description: [   ]

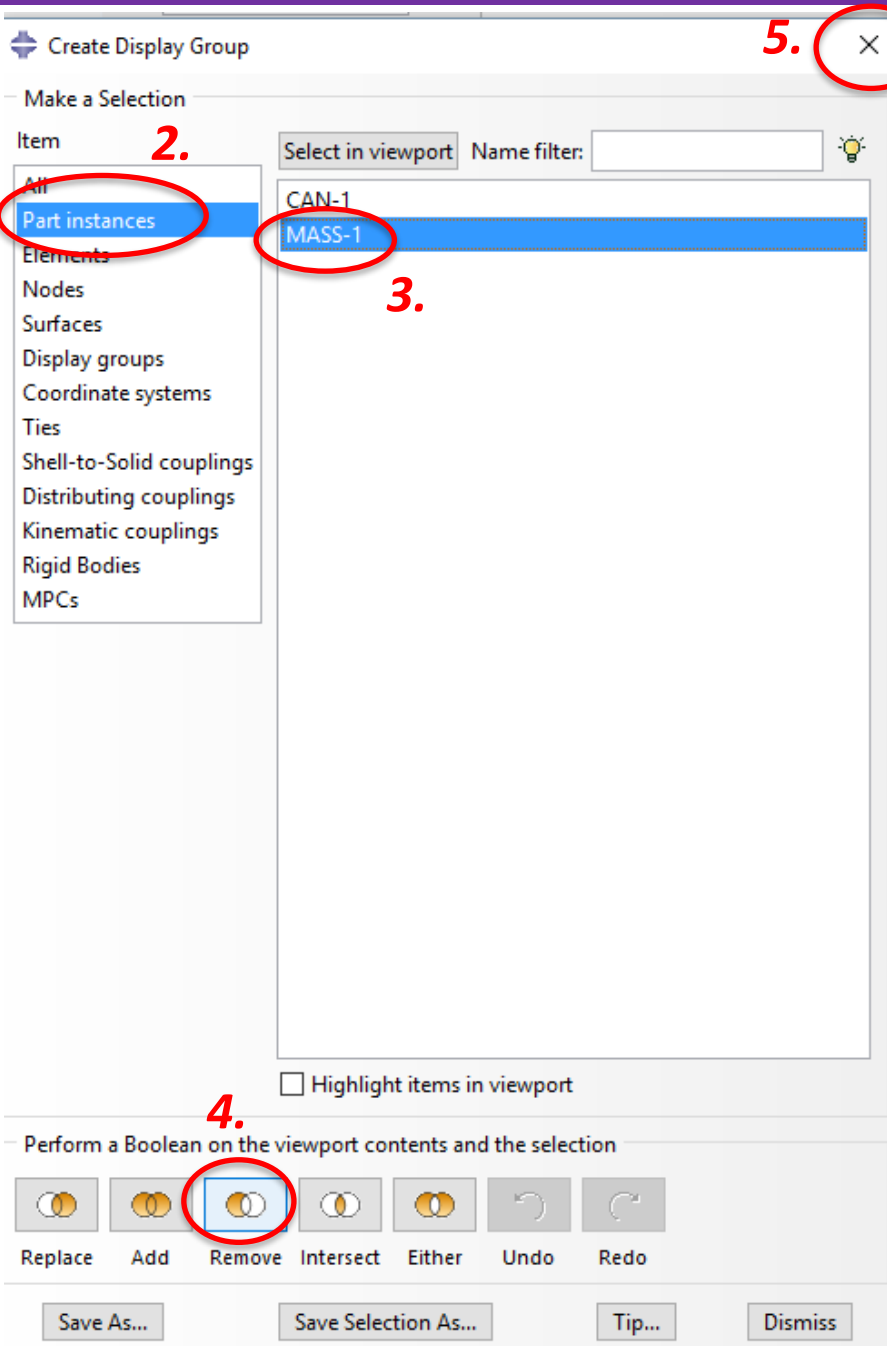| Submission | General | Memory | Parallelization | Precision |

Abaqus/Explicit precision: Double - analysis + packager ▾

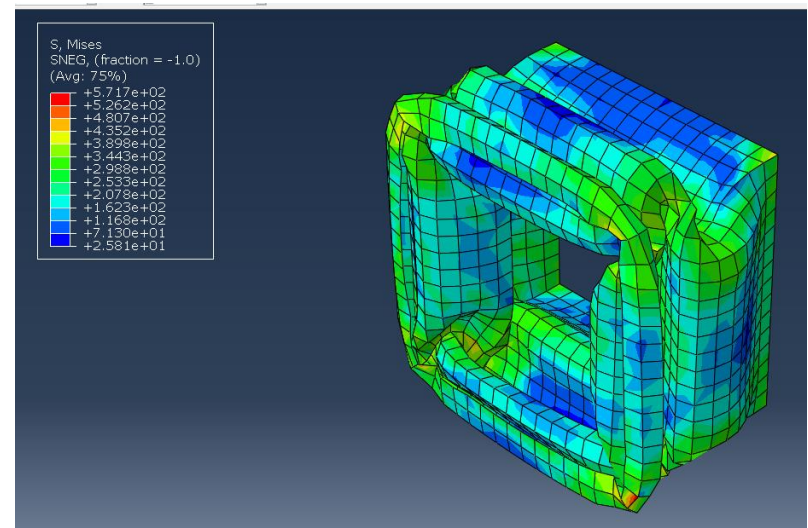Nodal output precision: Full ▾

[OK]   [Cancel]

- This time, ***Go the "Precision" tab within the edit job card.***

- ***Select Double+Analysis+Packager in the first field***

- ***Select Full in the second field***. This will make sure that ABAQUS will use double precision in all the computations required for the solution (This is always highly recommended, especially for explicit simulations which might require several thousands time increments for completion).

- ***Press OK and… Submit the analysis!***

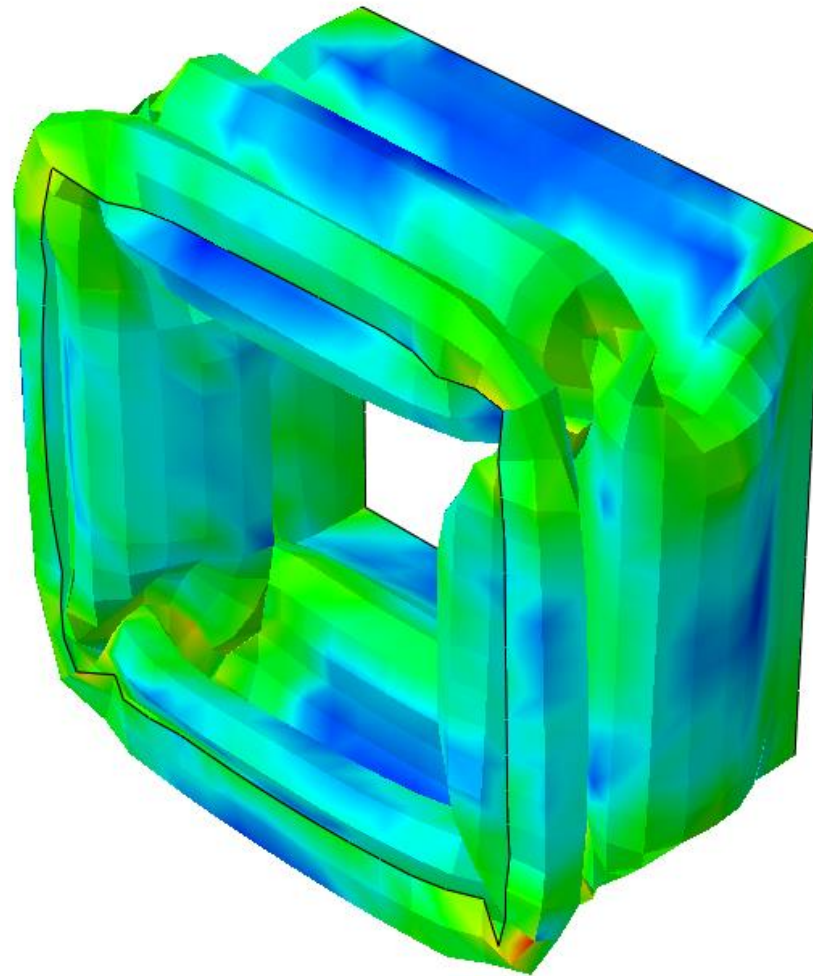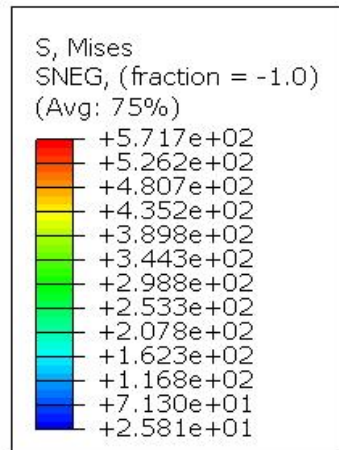- ***Once the analyses has completed, go to ABAQUS viewer as usual.***

Create Display Group

Make a Selection

Item | Select in viewport | Name filter: ☀
- All
- Part instances
- Elements
- Nodes
- Surfaces
- Display groups
- Coordinate systems
- Ties
- Shell-to-Solid couplings
- Distributing couplings
- Kinematic couplings
- Rigid Bodies
- MPCs

CAN-1
MASS-1

☐ Highlight items in viewport

Perform a Boolean on the viewport contents and the selection

Replace | Add | Remove | Intersect | Either | Undo | Redo

Save As... | Save Selection As... | Tip... | Dismiss

1. 2. 3. 4. 5.

- ***Click on the Display group icon in the viewer bar;***

- ***Select part instances*** as the type of item we want to remove from the view;

- ***Select Mass-1*** as the instance to be removed;

- Among the possible Boolean operations, ***choose the "Remove" option***;

- Among the possible Boolean operations, ***choose the "Remove" option***;

- ***Close the window*** the mass should have been removed from the view

S, Mises
SNEG, (fraction = -1.0)
(Avg: 75%)
+5.717e+02
+5.262e+02
+4.807e+02
+4.352e+02
+3.898e+02
+3.443e+02
+2.988e+02
+2.533e+02
+2.078e+02
+1.623e+02
+1.168e+02
+7.130e+01
+2.581e+01

S, Mises
SNEG, (fraction = -1.0)
(Avg: 75%)
+5.717e+02
+5.262e+02
+4.807e+02
+4.352e+02
+3.898e+02
+3.443e+02
+2.988e+02
+2.533e+02
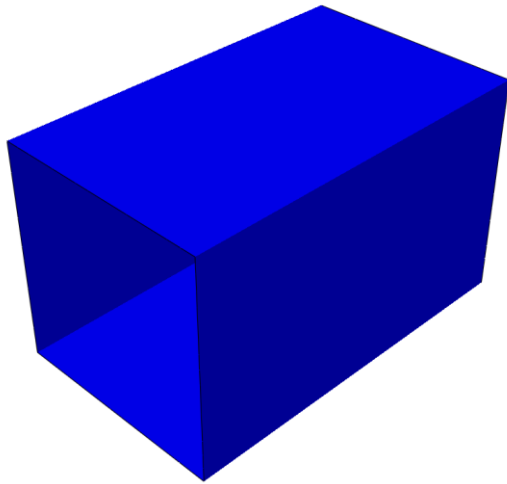+2.078e+02
+1.623e+02
+1.168e+02
+7.130e+01
+2.581e+01

- *__Note the extensive plastic buckling of the can. This extensive plastic deformation mode is the main energy absorbing mechanisms in aluminum and steel cans!__*
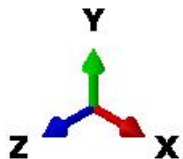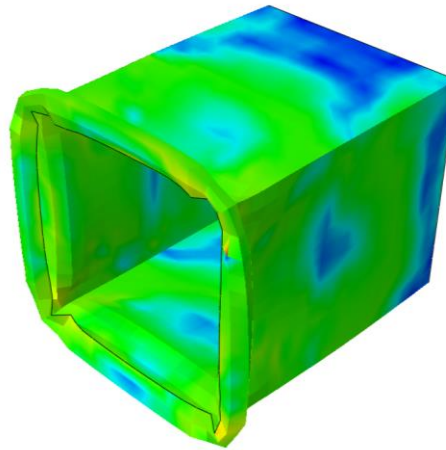
Step: Crash
Increment   29553: Step Time =   1.0000E-02
Primary Var: S, Mises
Deformed Var: U   Deformation Scale Factor: +1.000e+00

t = 0 ms

t = 1.5 ms

t = 3 ms

t = 4.5 ms

t = 6 ms

t = 7.5 ms

1.

Frame Selector

2.

Frame Selector

Crash

Crash: 0   Crash: 200

160
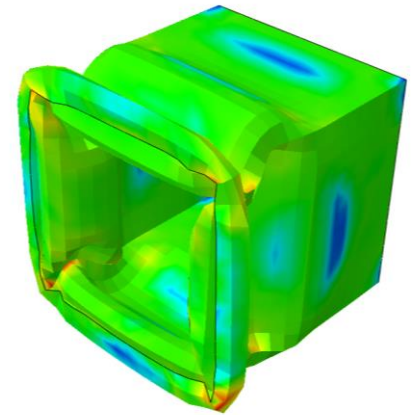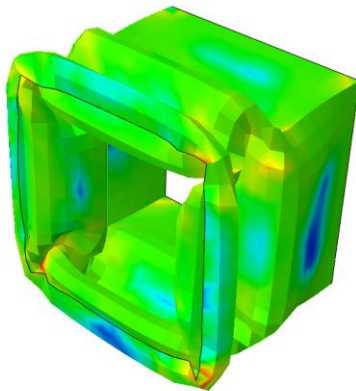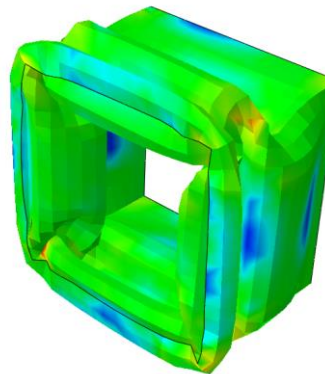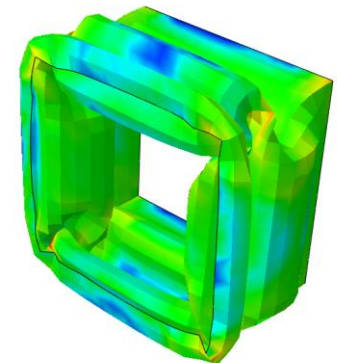
- By clicking the Frame selector icon at the top left of the screen it is possible to view the results at various time frames. Just play with the frame selector bar to navigate through the frames!
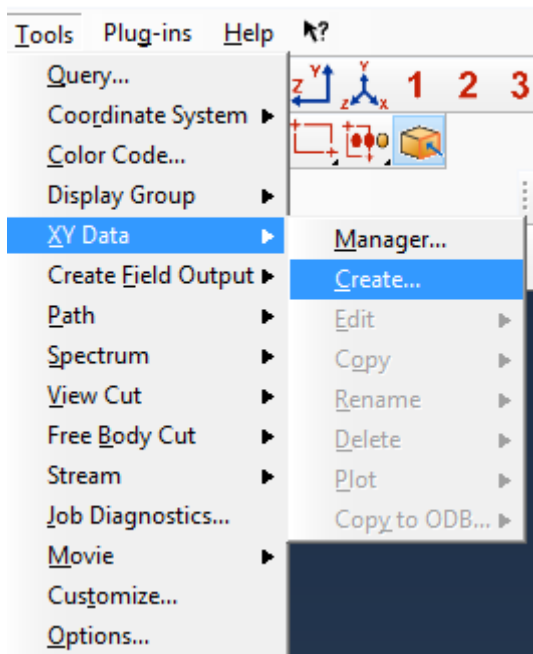
- In crash analyses, it is very important to know the evolution of the load exerted by the can to the mass during the impact. The acceleration of the impacting object is related to that force. Further, the evolution of the force with the axial displacement can be used to calculate the energy dissipated during the crash. ***Let's create a force-displacement plot***!

- We want to create a database containing the required information. As usual, ***Click Tools/XY data/Create in the menu bar***
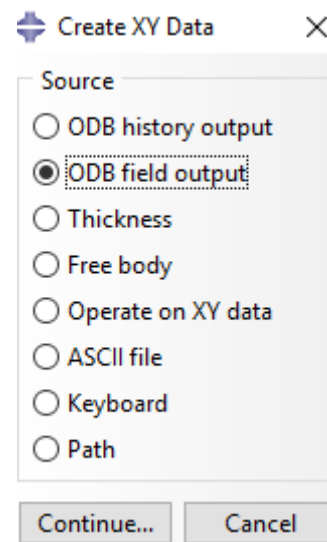
- ***Select ODB field output*** since accelerations, displacements, forces etc are field variables

- ***Press Continue…***

- The XY data card enables the selection of the required variables. While e.g. stresses and strains are available at integration points, forces, displacements, accelerations etc are calculated at nodes. To select the acceleration and the displacement, **_Select Unique Nodal as position;_**

- We will use the acceleration in the z direction to calculate the force acting on the mass so... **_Expand Spatial acceleration and Select A3 (acceleration in z);_**

- Scroll down, **_Expand Spatial displacement and Select U3 (displacement in z);_**

- We need to specify from which nodes we want to extract the required variables. **_Select the tab Elements/Nodes;_**

- **_Click on Edit Selection._** You will be able to select the nodes of interest. In our case, we want to know the acceleration and displacement of the mass so we need to select the node located at the Reference Point. This will be representative of the dynamic behavior of the mass.
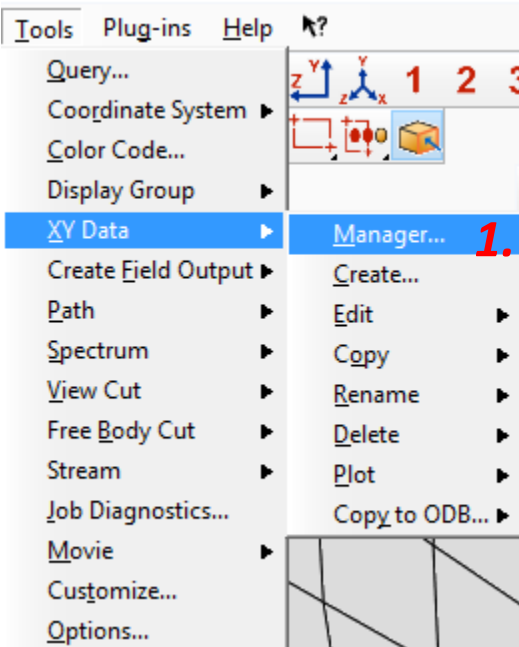
- **_Select the RP and the Click Done at the bottom left._**

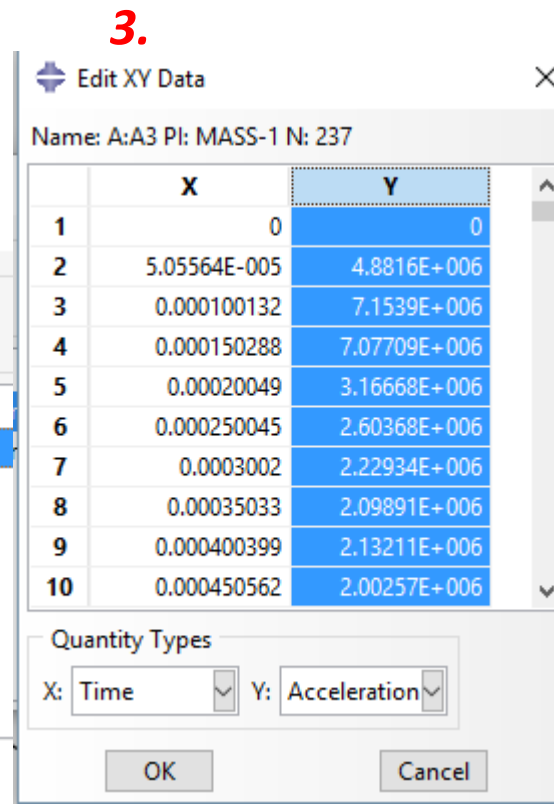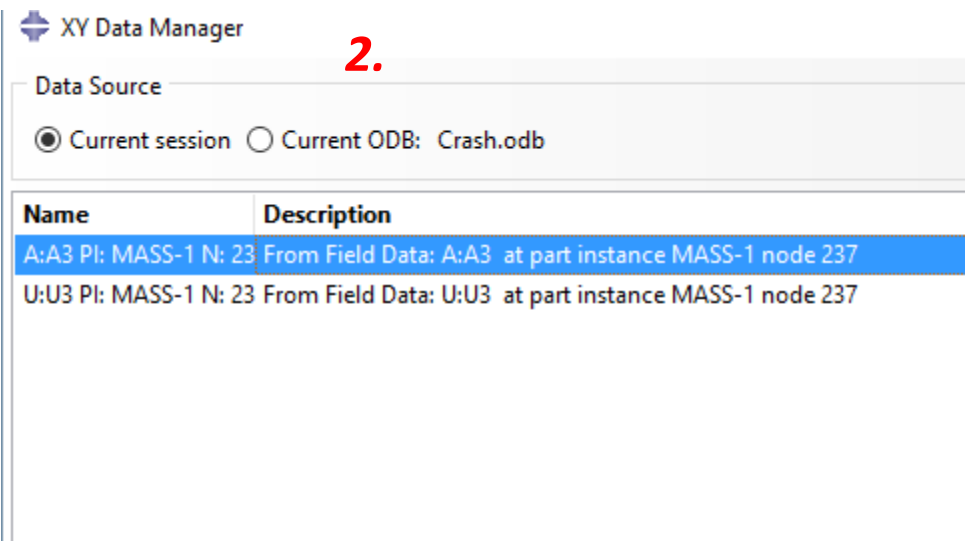- After selecting the RP the window should appear as below;

- **_Click on SAVE._** ABAQUS will save the acceleration and the displacement components in z of the RP for all the requested time intervals;

- A confirmation message will appear. **_Click OK then close the XY data window._**

- In this case we have 2 variables only. There is no need for creating a report. We can copy-past the variables directly from ABAQUS to the preferred software (e.g Matlab). **_Go to Tools/XY data/ Manager_**

- **_In the XY Data manager double click on the desired variable. This will open the database which you can copy-paste in the desired software!_**

Tools menu:
- Query...
- Coordinate System ▶
- Color Code...
- Display Group ▶
- XY Data ▶
- Create Field Output ▶
- Path ▶
- Spectrum ▶
- View Cut ▶
- Free Body Cut ▶
- Stream ▶
- Job Diagnostics...
- Movie ▶
- Customize...
- Options...

XY Data submenu:
- Manager... *1.*
- Create...
- Edit ▶
- Copy ▶
- Rename ▶
- Delete ▶
- Plot ▶
- Copy to ODB... ▶

XY Data Manager *2.*

Data Source

◉ Current session ○ Current ODB: Crash.odb

Name	Description
A:A3 PI: MASS-1 N: 23	From Field Data: A:A3 at part instance MASS-1 node 237
U:U3 PI: MASS-1 N: 23	From Field Data: U:U3 at part instance MASS-1 node 237

3.

Edit XY Data

Name: A:A3 PI: MASS-1 N: 237

	X	Y
1	0	0
2	5.05564E-005	4.8816E+006
3	0.000100132	7.1539E+006
4	0.000150288	7.07709E+006
5	0.00020049	3.16668E+006
6	0.000250045	2.60368E+006
7	0.0003002	2.22934E+006
8	0.00035033	2.09891E+006
9	0.000400399	2.13211E+006
10	0.000450562	2.00257E+006

Quantity Types

X: Time   Y: Acceleration

OK   Cancel

- To compute the force acting on the mass (the reaction force of the can) we just need to apply Newton's 2nd law. Multiply the acceleration with the mass to find the force. The plot of the force as a function of the displacement should look as below

Initial kin energy: 1.125 kJ

Initial velocity: 15 m/s

Mass: 10 kg
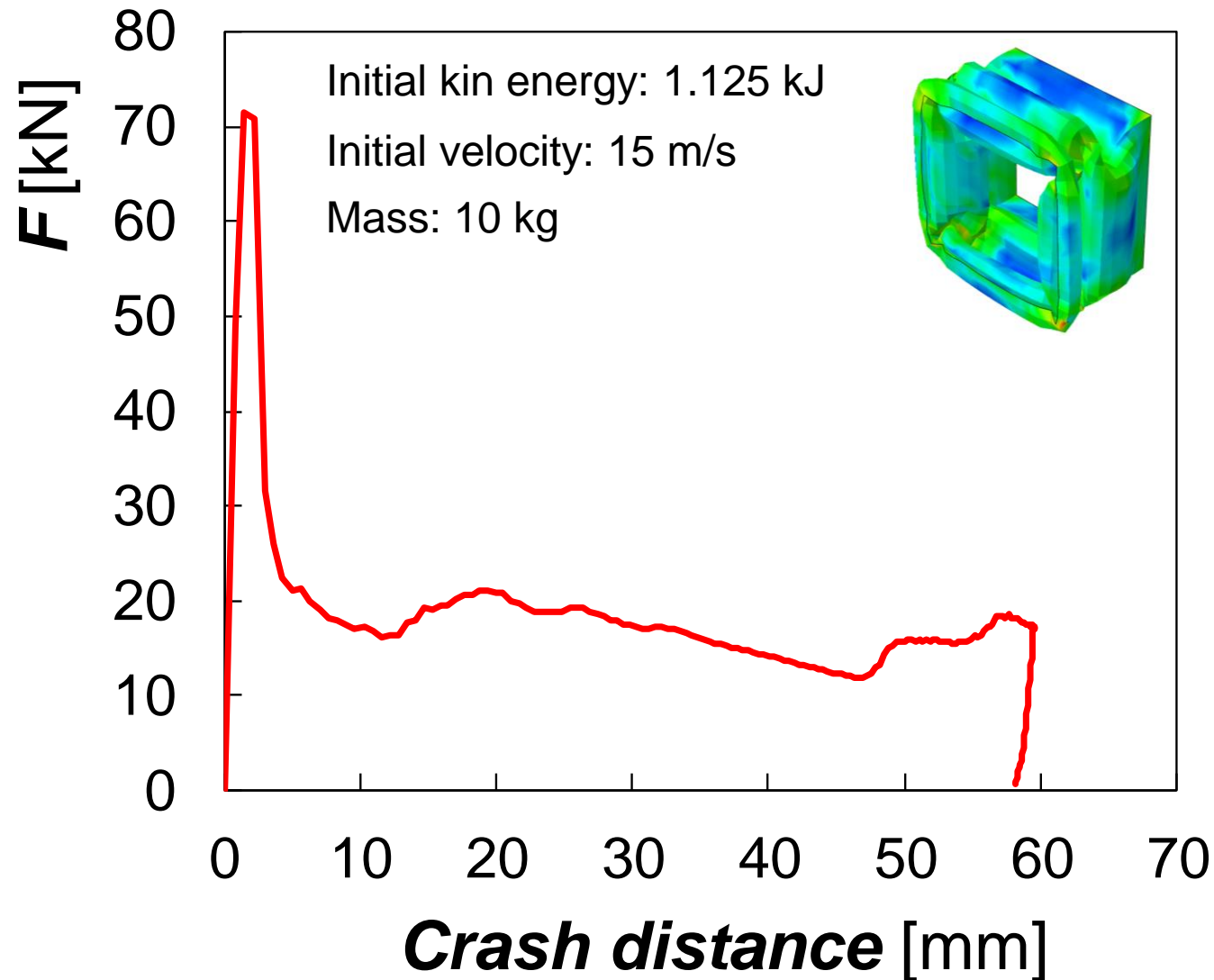
- The shape of the force – displacement curve is highly considered for the correct design of a crash absorber for civil/mechanical structures. The design should avoid the presence of spikes since they are related to dangerous decelerations and the average deceleration should not exceed prescribed limits.

Initial kin energy: 1.125 kJ

Initial velocity: 15 m/s

Mass: 10 kg

Not desirable!