PennState
College of Engineering

ME 563: Nonlinear Finite Elements

Application of the Finite Element Method to Real World Problems

Application: Ball-Plate Impact with "Failure"

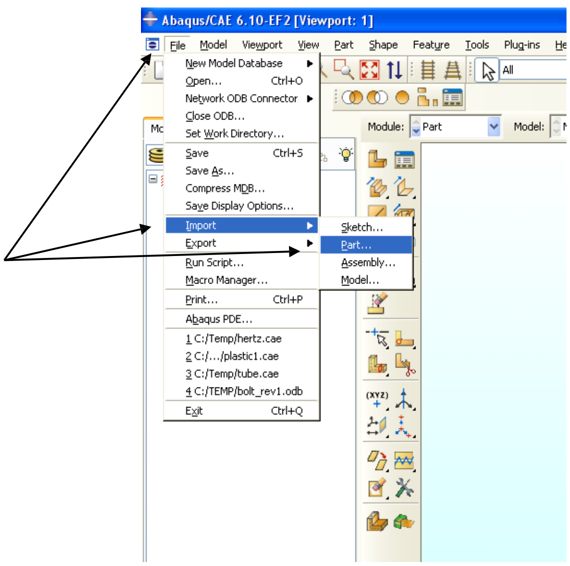
This tutorial accompanies covers a basic example of a ball being fired at an aluminum plate. An element deletion criterion is defined and therefore the plate ruptures and allows the ball to pass through

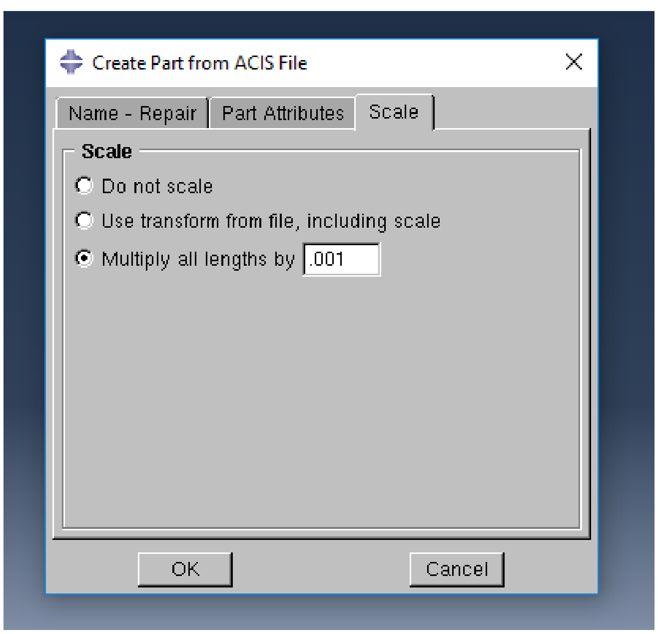
1. Geometry Import

Import the geometry in the form of a step file – **ball-plate.sat** located on Github.

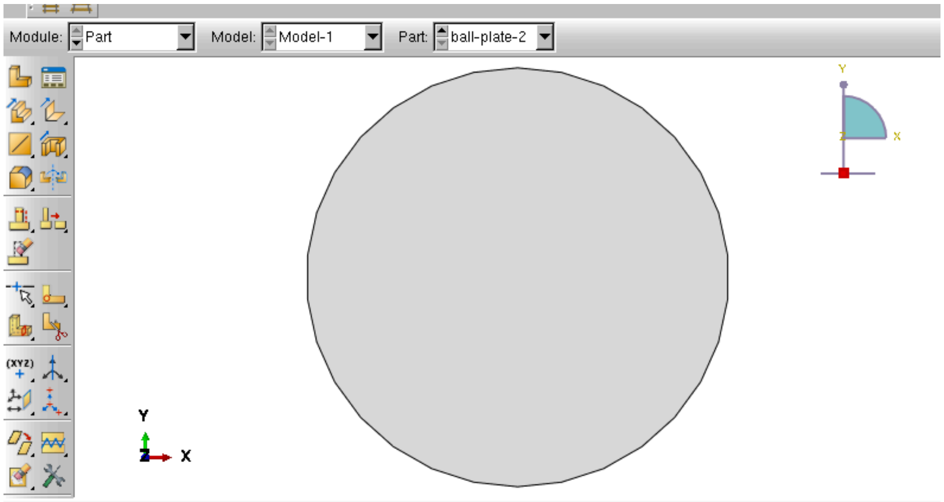
git clone https://github.com/rhk12/ball-impact

File-> Import-> Part, Scale the part by 0.001 to make it in meters.



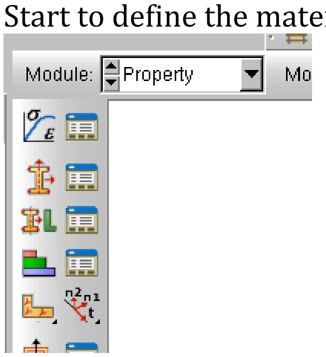


Should look like this:



2. Material and section properties

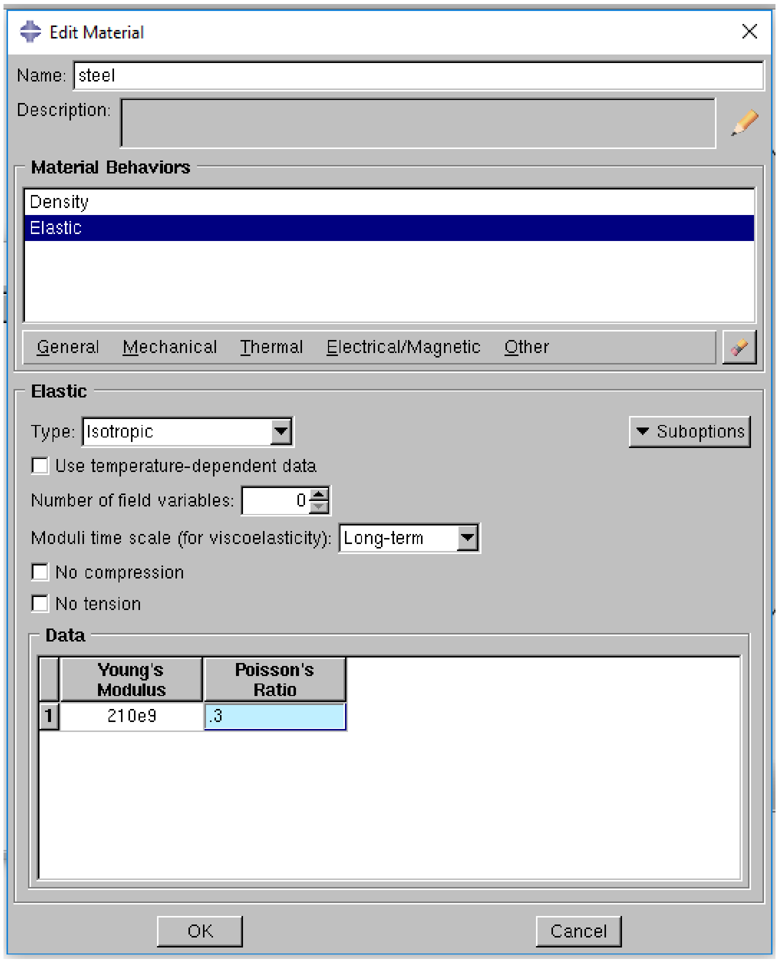
Start to define the material properties – go into the properties module



Define two materials – **Steel** which will only have linear properties and **Aluminum** which will have yield and post yield behavior.

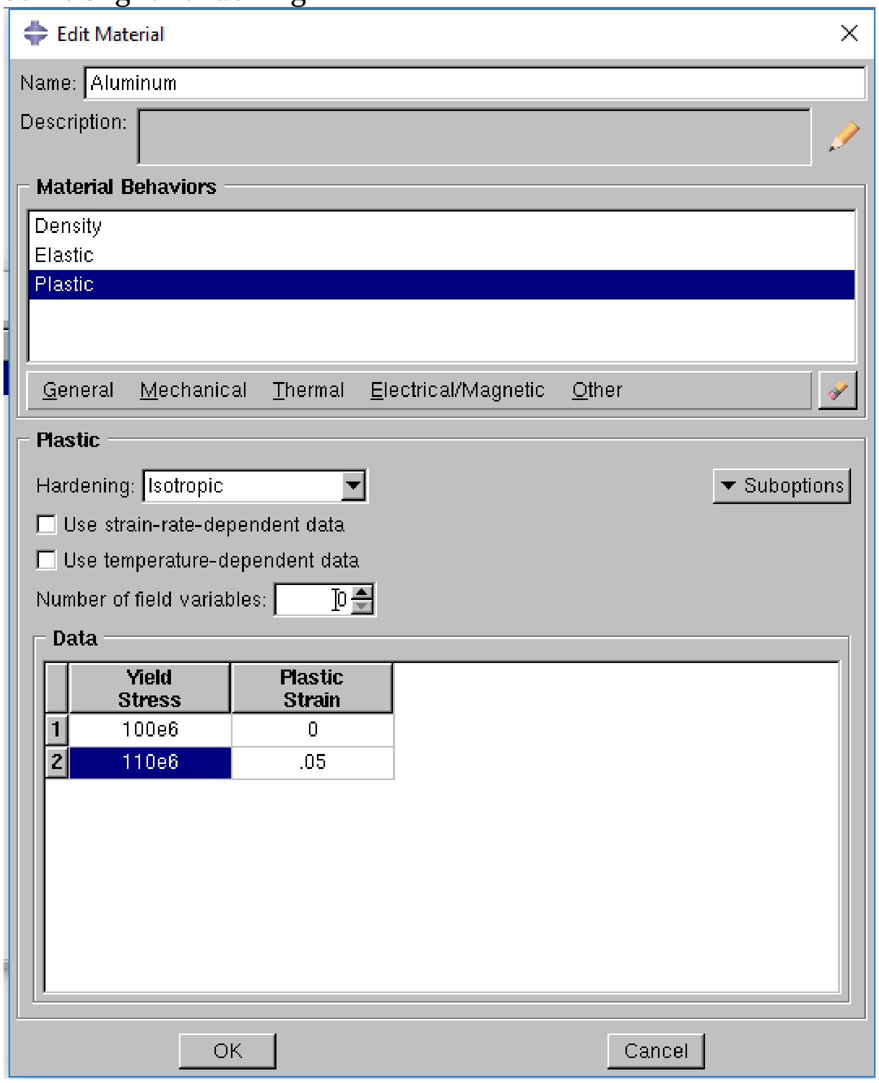
As the problem is dynamic we also need to define a density for both.

1. **Steel** – Young’s Modulus = 210e9 Pa and Poisson’s Ratio = 0.3, density = 7800 kg/m^3

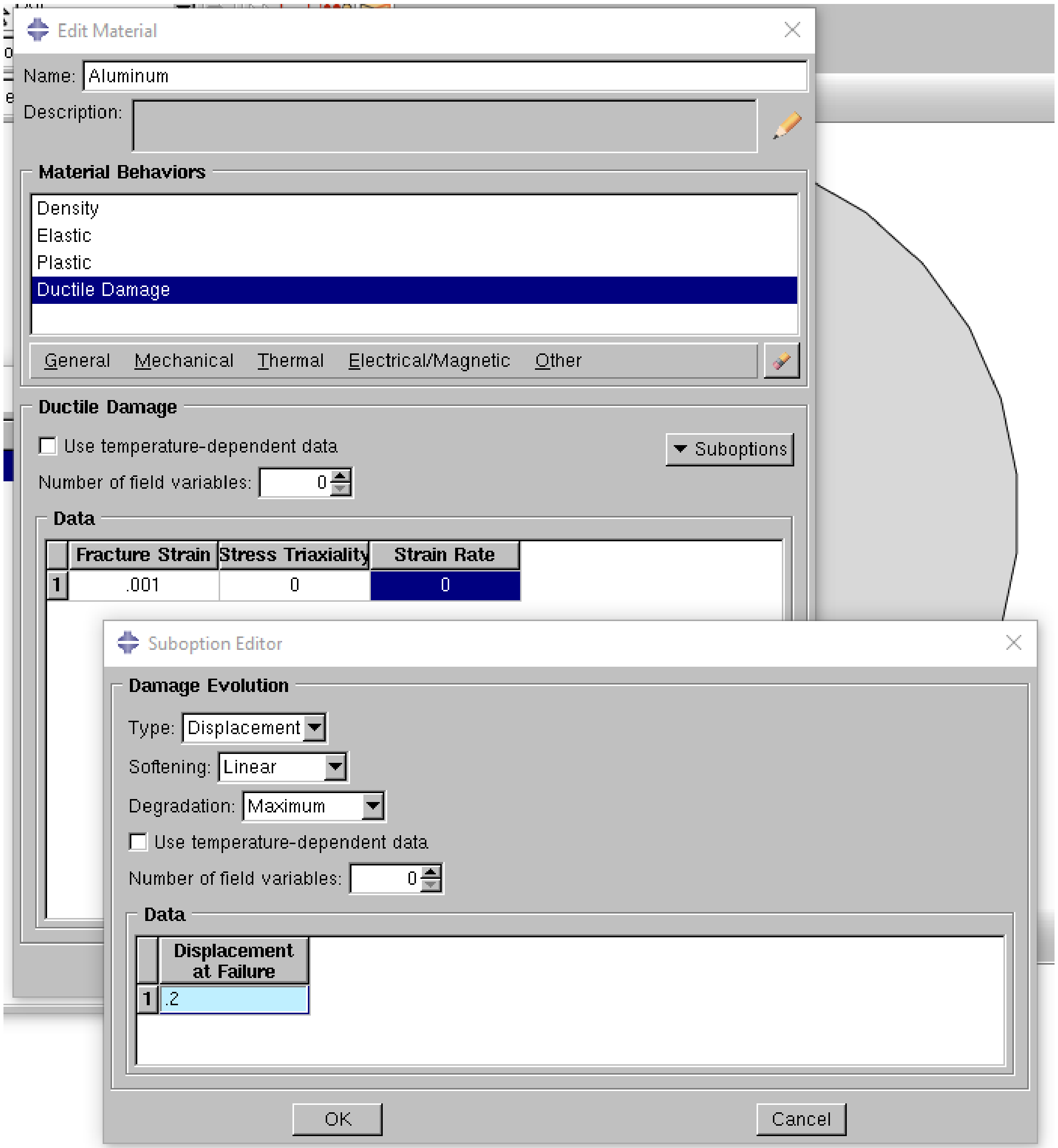

Edit Material
Name: steel
Description:
Material Behaviors
Density
Elastic
General
Mechanical
Thermal
Electrical/Magnetic
Other
Elastic
Type: Isotropic
Suboptions
Use temperature-dependent data
Number of field variables: 0
Moduli time scale (for viscoelasticity): Long-term
No compression
No tension
Data
Young's Modulus
Poisson's Ratio
1
210e9
.3
OK
Cancel

2. **Aluminum** – Young's Modulus = 70e9 Pa and Poisson's Ratio = 0.3, density = 2500 kg/m^3

A simple bi-linear material plasticity definition is used. Initial yield stress is set at 100e6 Pa with some slight hardening.



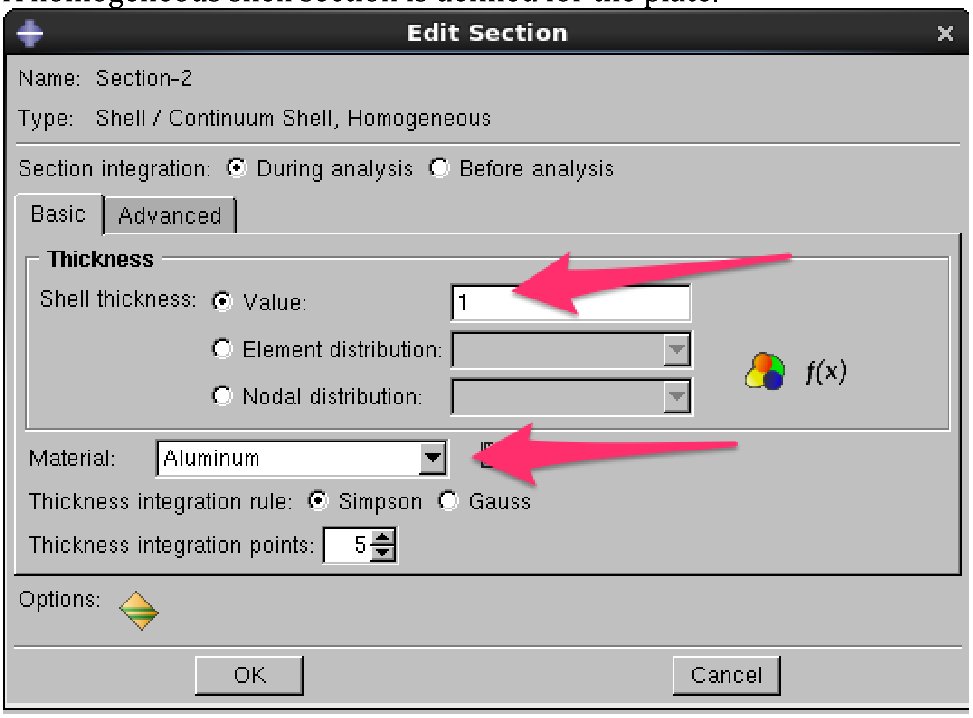
Add the Ductile Damage option. Element deletion criteria: Displacement at failure (damage evolution) is defined using the Suboption button.



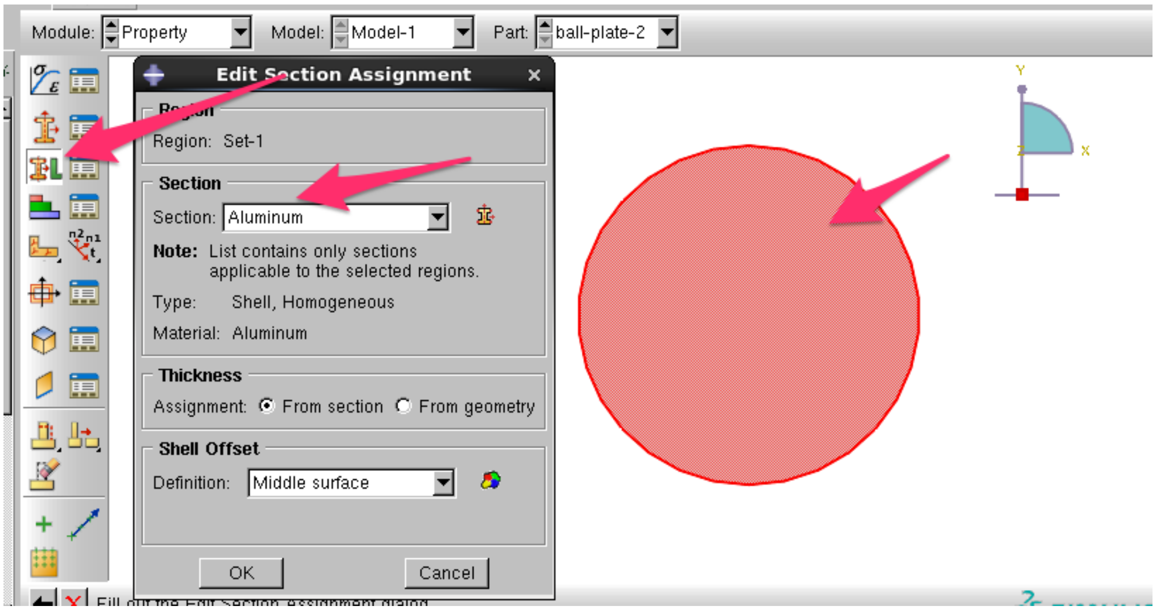
3. Define Sections

A simple steel section (solid., homogeneous) is defined and applied to the ball.

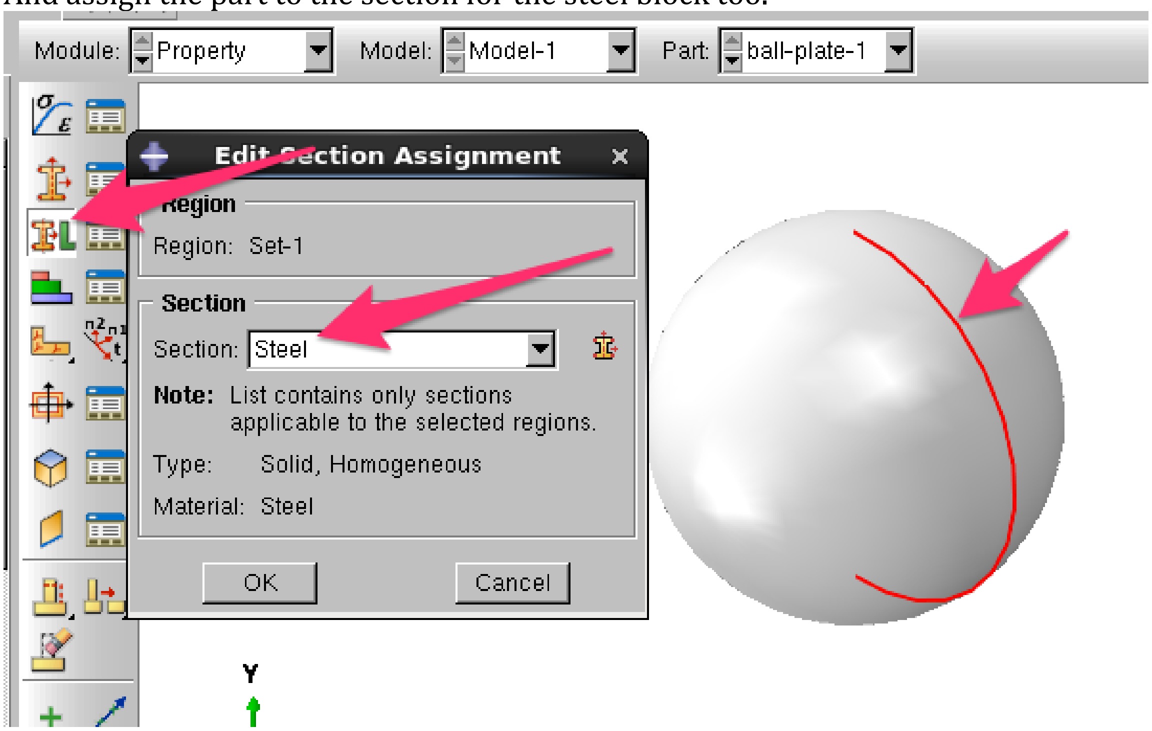
A homogeneous shell section is defined for the plate.



Assign sections:
For Al:

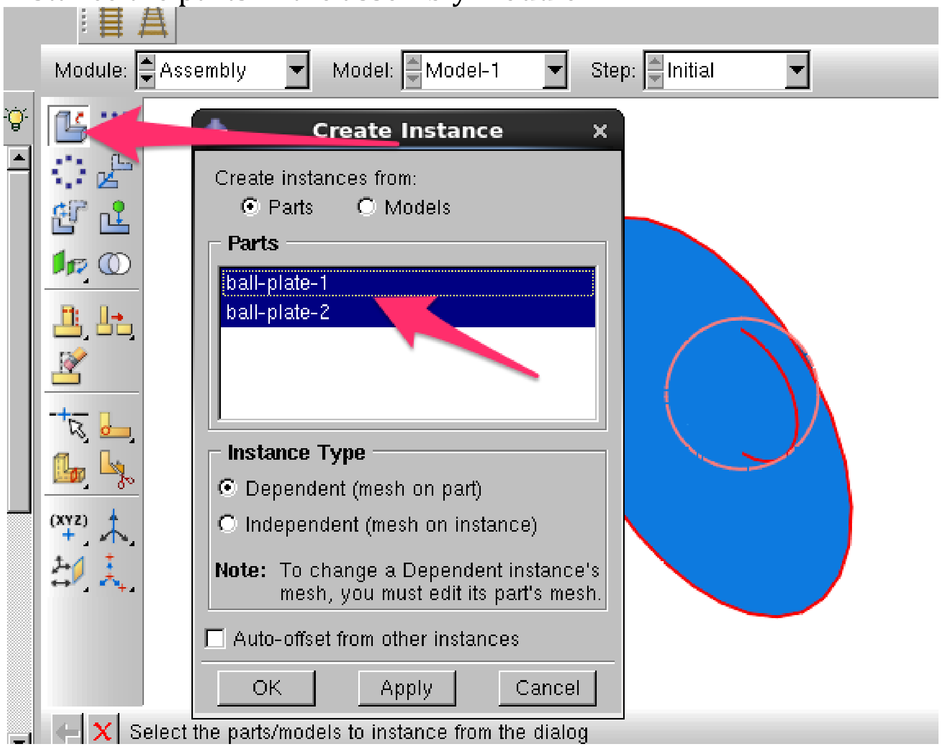


And assign the part to the section for the steel block too.



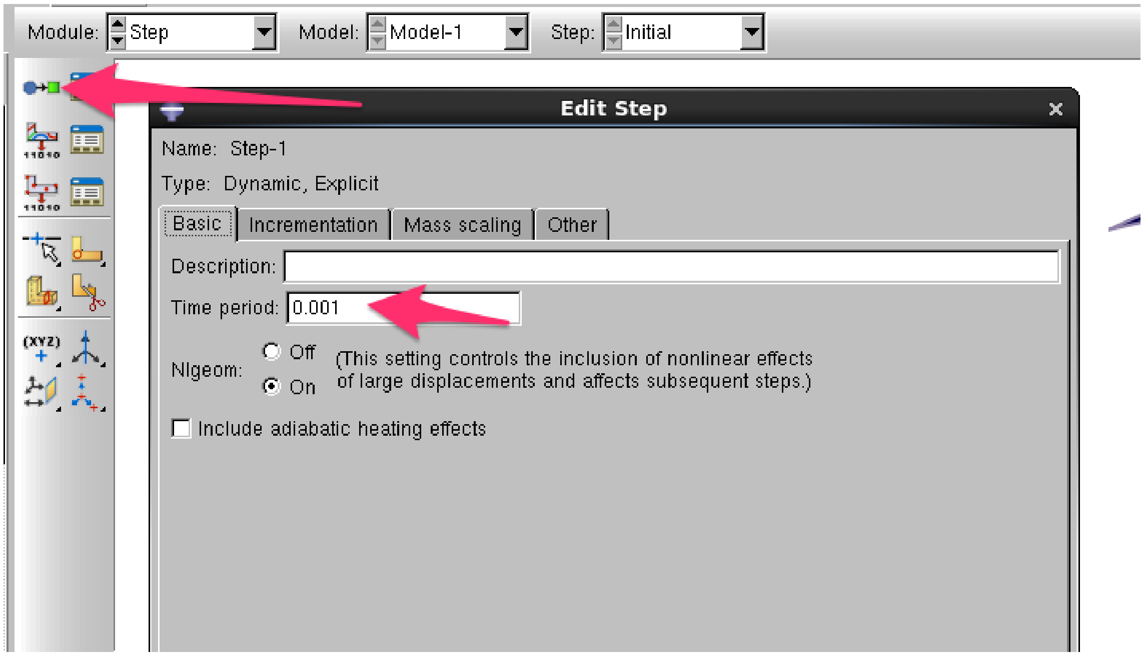
4. Create an assembly

Instance the parts in the assembly module.



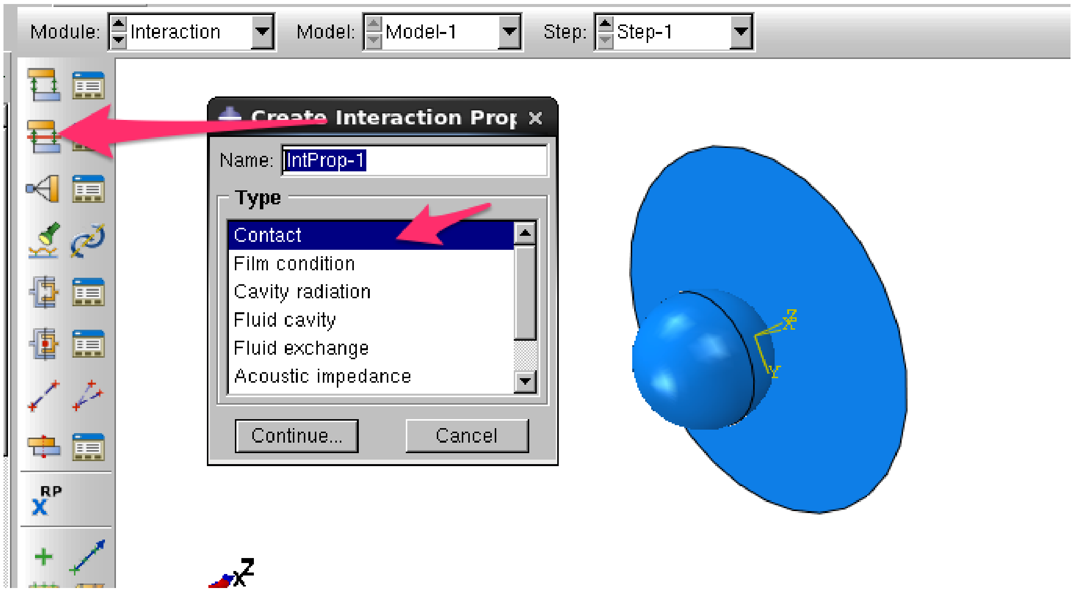
5. Define the analysis step

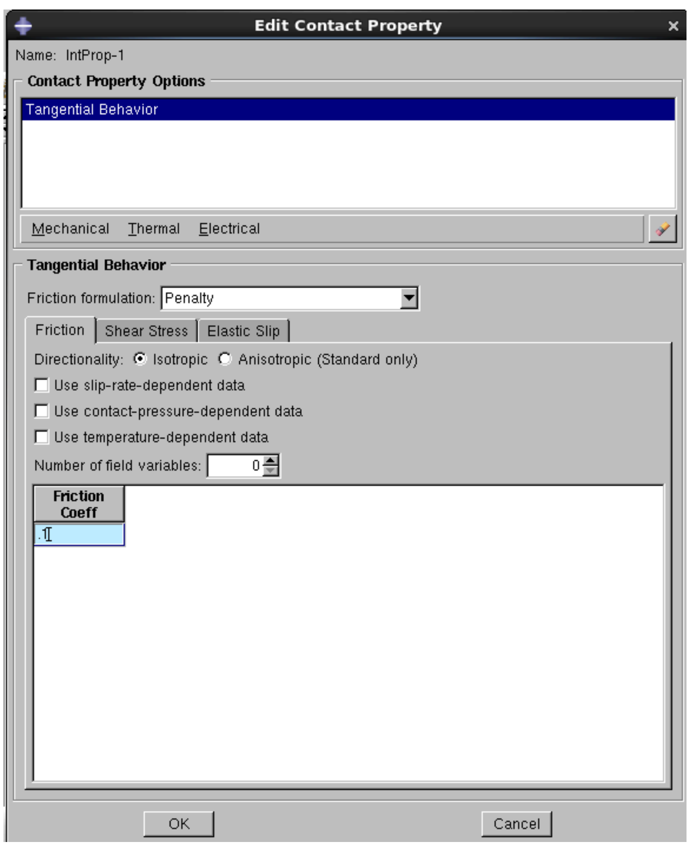
Create a dynamic explicit step and include Nlgeom. Set the time period = 0.001 s

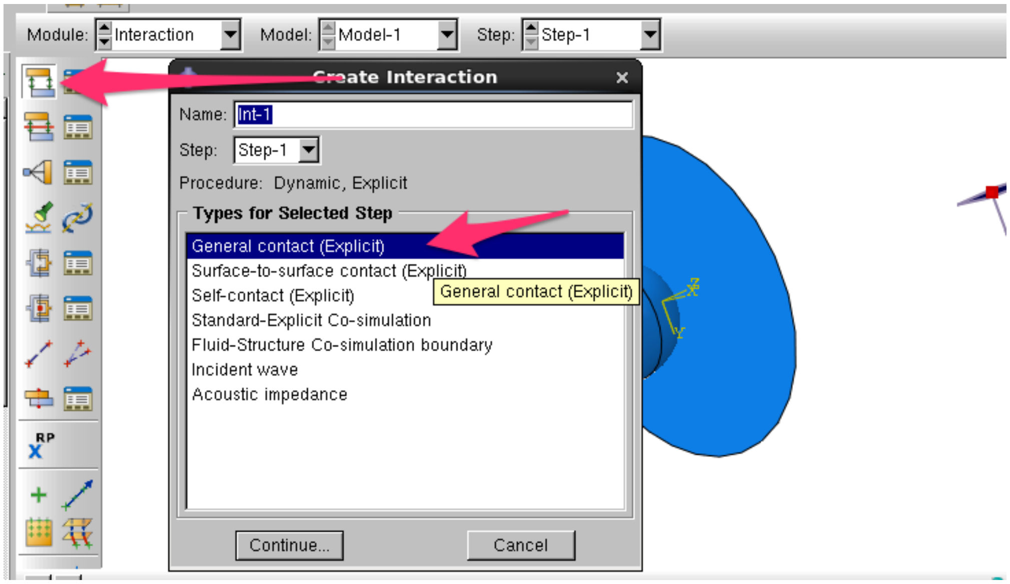

Module: Step
Model: Model-1
Step: Initial
Edit Step
Name: Step-1
Type: Dynamic, Explicit
Basic
Incrementation
Mass scaling
Other
Description:
Time period: 0.001
Nlgeom:
Off
On
(This setting controls the inclusion of nonlinear effects of large displacements and affects subsequent steps.)
Include adiabatic heating effects

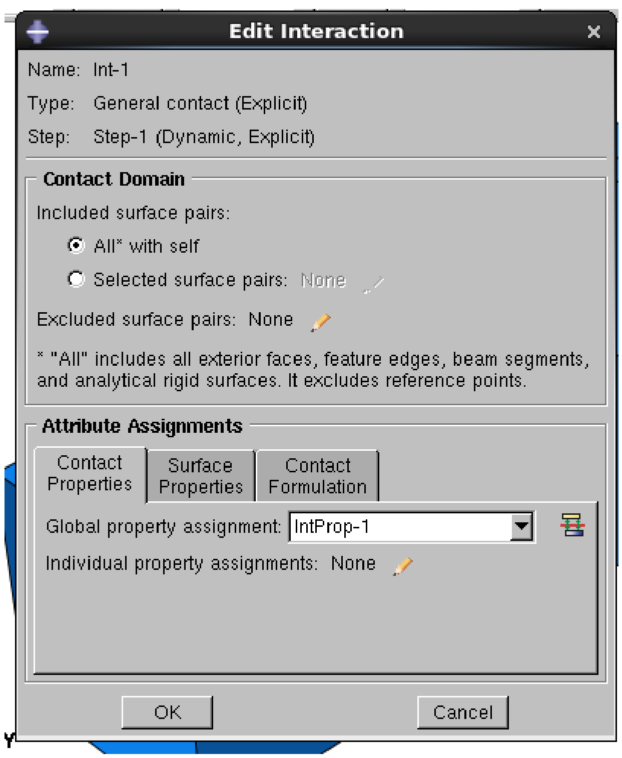
6. Define the Interactions (contact conditions)

The "catch all" general contact with the default All* with self option provides all the contact settings necessary. In this case a global interaction property defining a coefficient of friction of 0.1 is used.



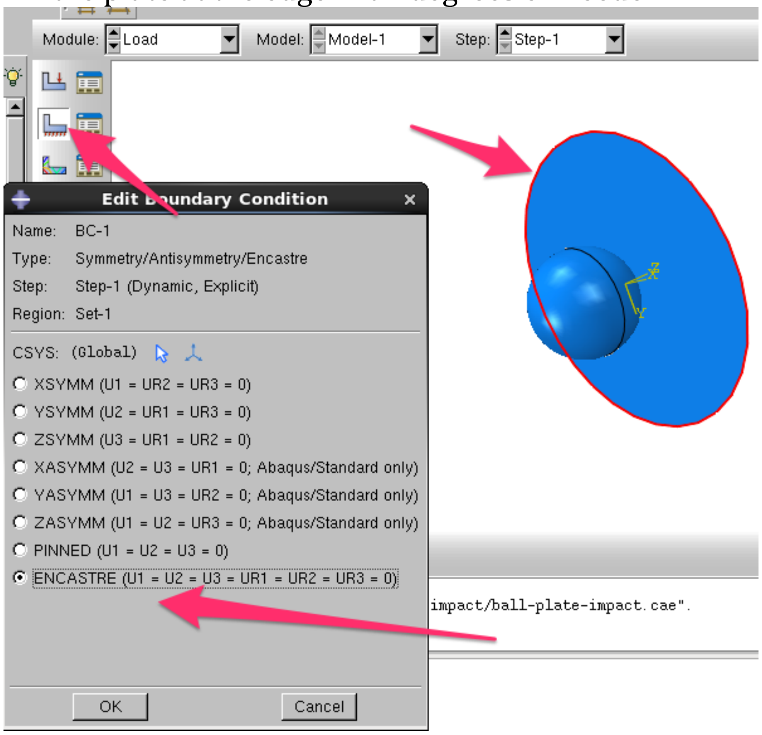



Module: Interaction
Model: Model-1
Step: Step-1
Create Interaction
Name: Int-1
Step: Step-1
Procedure: Dynamic, Explicit
Types for Selected Step
General contact (Explicit)
Surface-to-surface contact (Explicit)
Self-contact (Explicit)
Standard-Explicit Co-simulation
Fluid-Structure Co-simulation boundary
Incident wave
Acoustic impedance
General contact (Explicit)
Continue...
Cancel

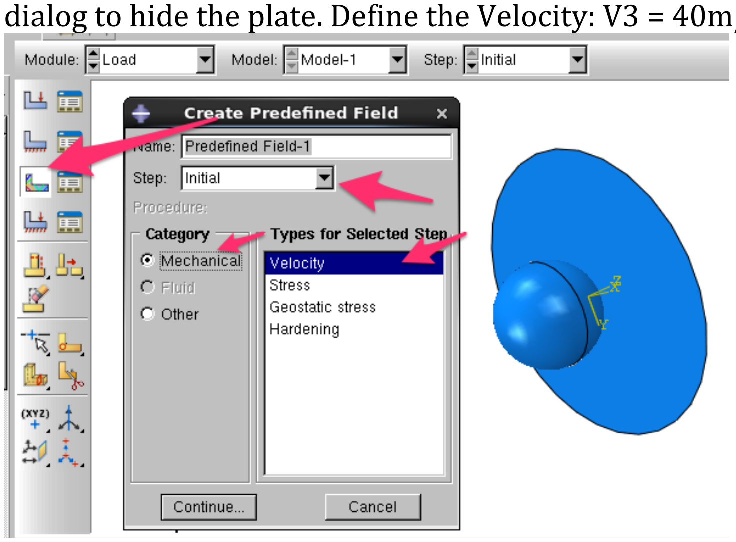

Edit Interaction
Name: Int-1
Type: General contact (Explicit)
Step: Step-1 (Dynamic, Explicit)
Contact Domain
Included surface pairs:
All* with self
Selected surface pairs: None
Excluded surface pairs: None
* "All" includes all exterior faces, feature edges, beam segments, and analytical rigid surfaces. It excludes reference points.
Attribute Assignments
Contact Properties
Surface Properties
Contact Formulation
Global property assignment: IntProp-1
Individual property assignments: None
OK
Cancel

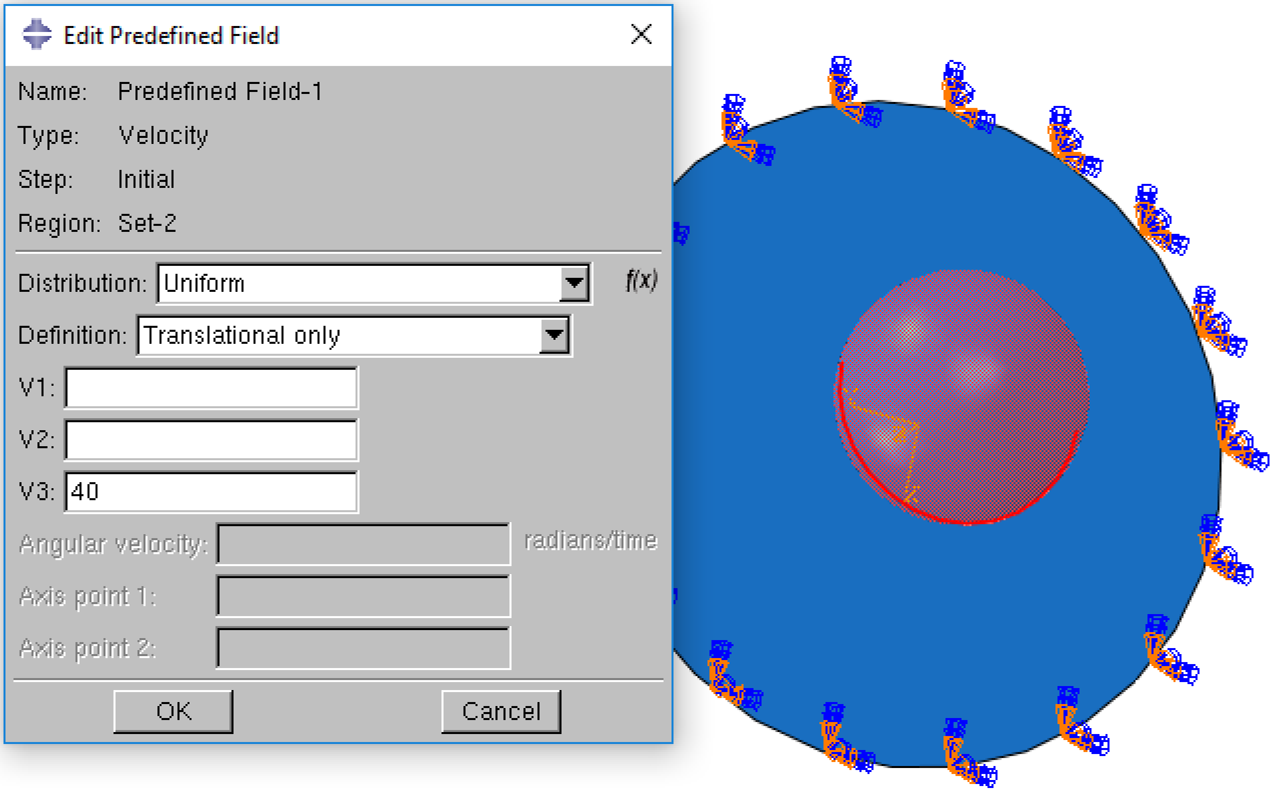
7. Define loads and restraints

Fix the plate at the edge in all degrees of freedom.

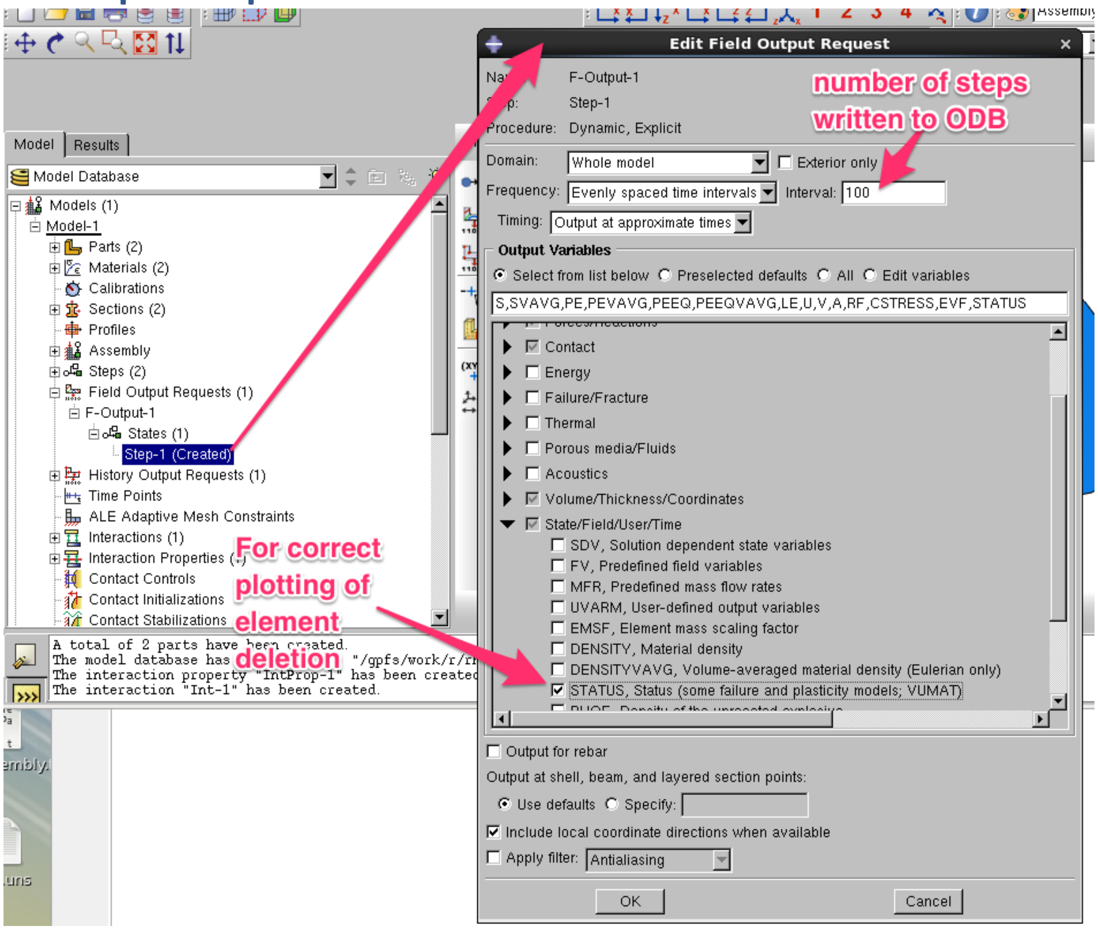


The ball is given an initial velocity, predefined field > mechanical > velocity. Ensure the entire ball is selected and not just the outer surface, this is best done using the assembly display options dialog to hide the plate. Define the Velocity: V3 = 40m/s.



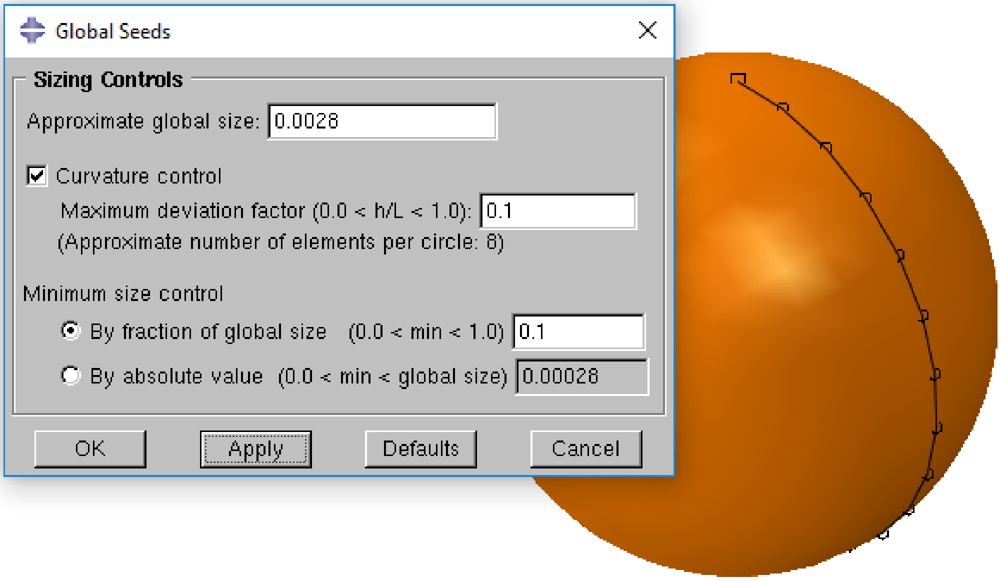

Edit Predefined Field
Name: Predefined Field-1
Type: Velocity
Step: Initial
Region: Set-2
Distribution: Uniform
f(x)
Definition: Translational only
V1:
V2:
V3: 40
Angular velocity:
radians/time
Axis point 1:
Axis point 2:
OK
Cancel

8. Output Requests

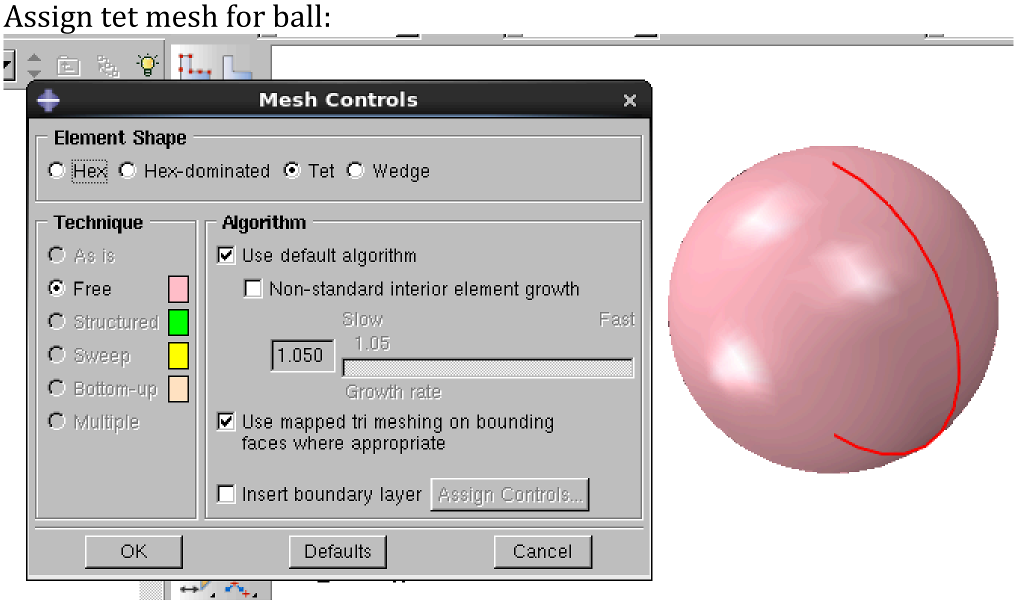


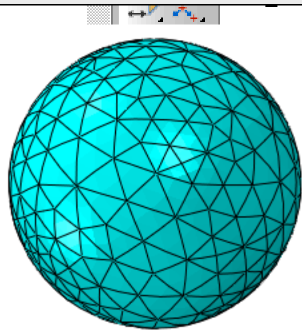
9. Create the Mesh

Mesh the ball. Use seed of .0028

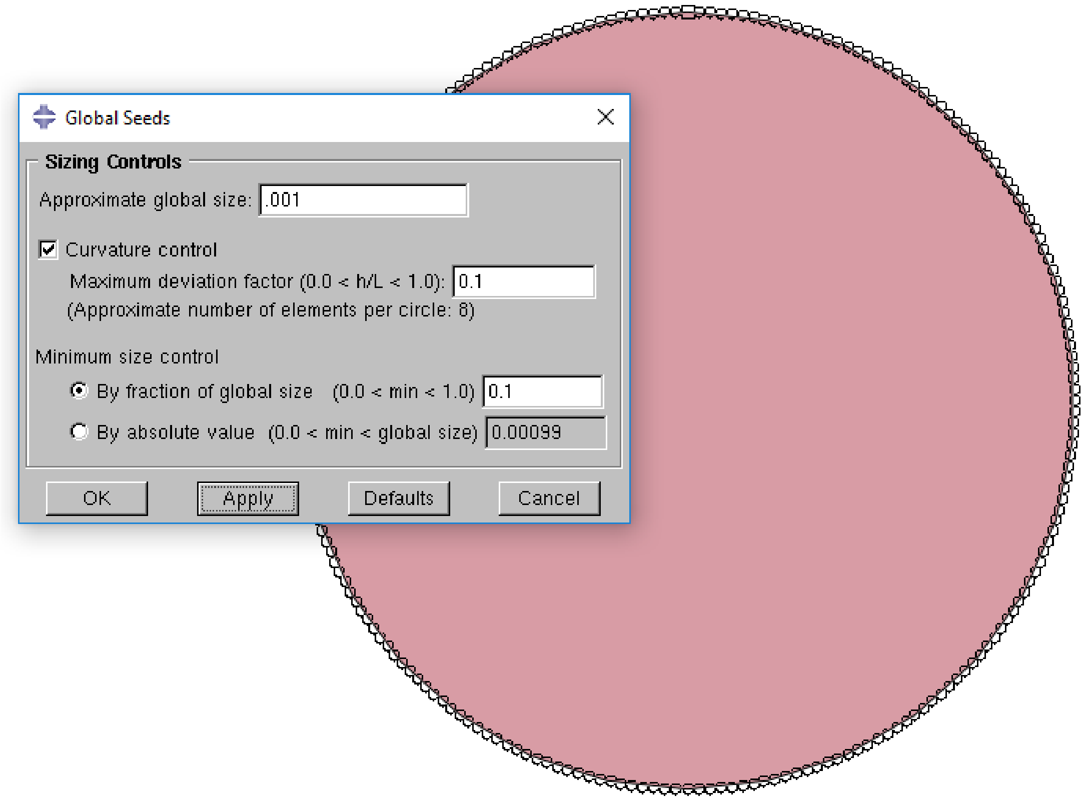


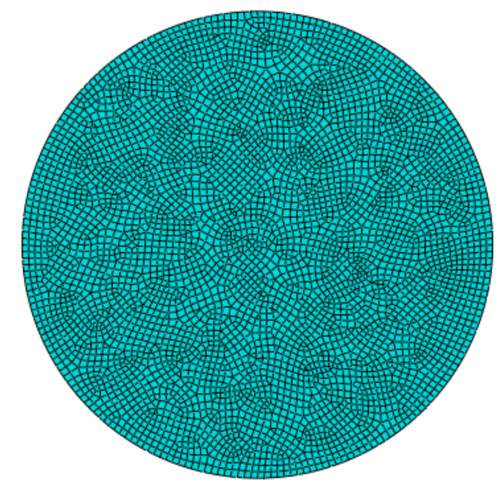
Assign tet mesh for ball:





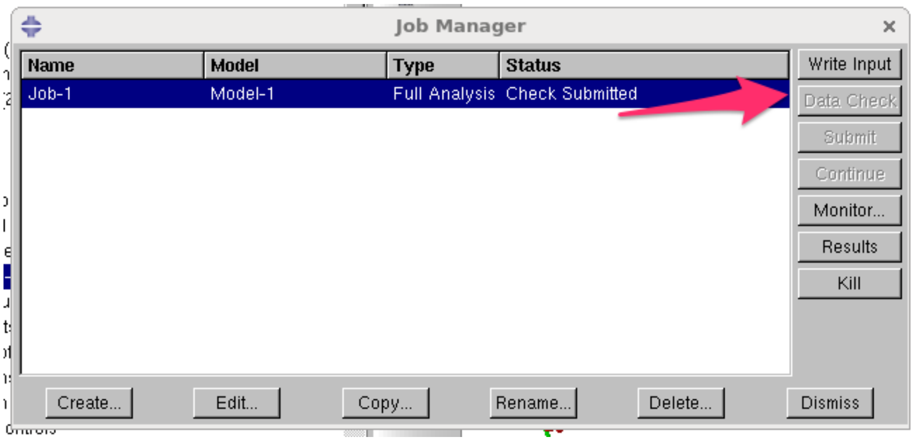
Use a mesh seed size of 0.001 for the plate:



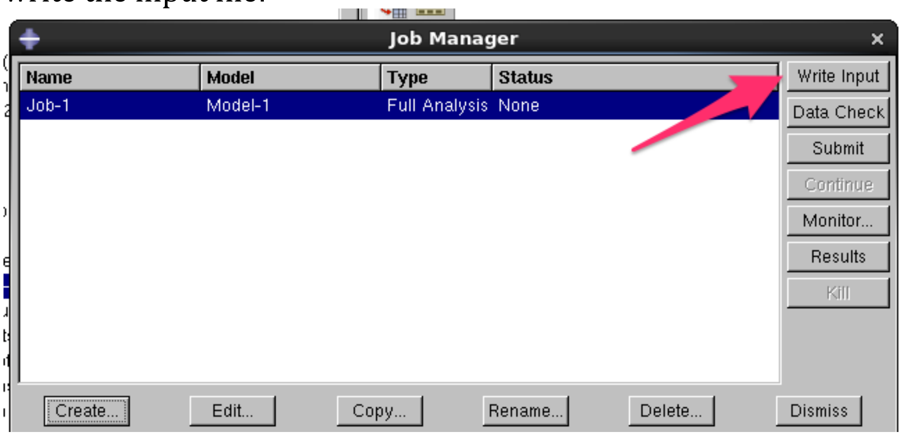


10. Create a job and then submit it using PBS

Submit the job and run a data check:



write the input file:



Open a terminal and ssh to aci-b. Navigate to your directory. Submit pbs script:

```
-rwxrwx--- 1 rhk12 rhk12_collab     917 Feb  7 14:55 Job-1.log
-rw-rw---- 1 rhk12 rhk12_collab  917508 Feb  7 14:57 Job-1.inp
-rw-rw---- 1 rhk12 rhk12_collab  344064 Feb  7 15:14 app4_v1.cae
-rw-rw---- 1 rhk12 rhk12_collab   10341 Feb  7 15:14 abaqus.rpy
-rw-rw---- 1 rhk12 rhk12_collab   10734 Feb  7 15:14 app4_v1.jnl
-rw-r----- 1 rhk12 rhk12_collab     281 Feb  7 15:24 abaqus.pbs
[rhk12@aci-lgn-007 ball-impact]$ qsub -A open abaqus.pbs
```

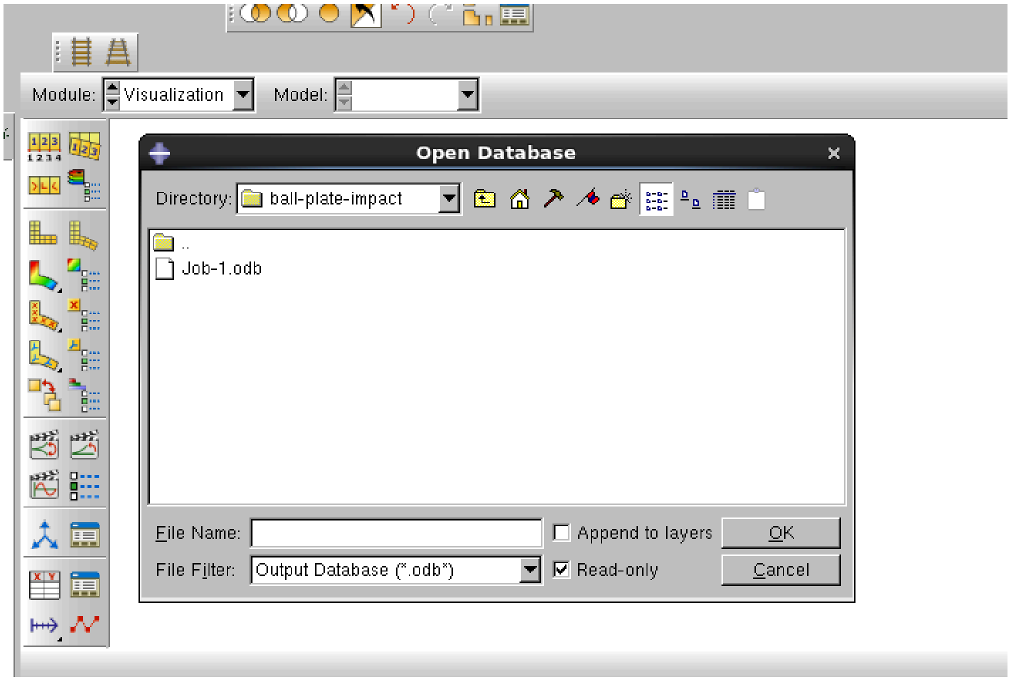
Monitor the job:

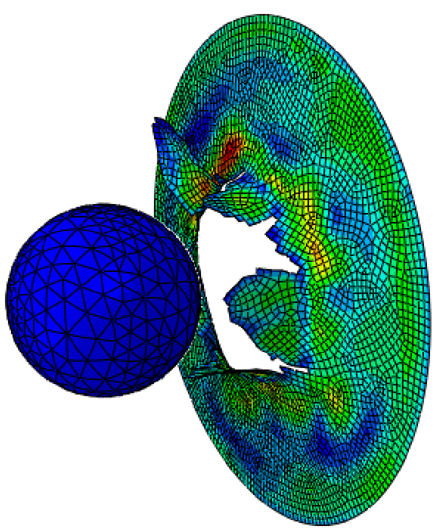
```
[rhk12@aci-lgn-007 ball-impact]$
[rhk12@aci-lgn-007 ball-impact]$ qstat -u rhk12

torque01.util.production.int.aci.ics.psu.edu:
                                                                           Req'd       Req'd        Elap
Job ID                  Username    Queue    Jobname          SessID  NDS   TSK    Memory      Time     S   Time
----------------------- ----------- -------- ---------------- ------ ----- ------ --------- --------- - ---------
10536102.torque01.util  rhk12       open     abaqus.pbs          --      1      2       4gb  00:30:00 Q        --
[rhk12@aci-lgn-007 ball-impact]$
```

11. Open the results file and plot the results

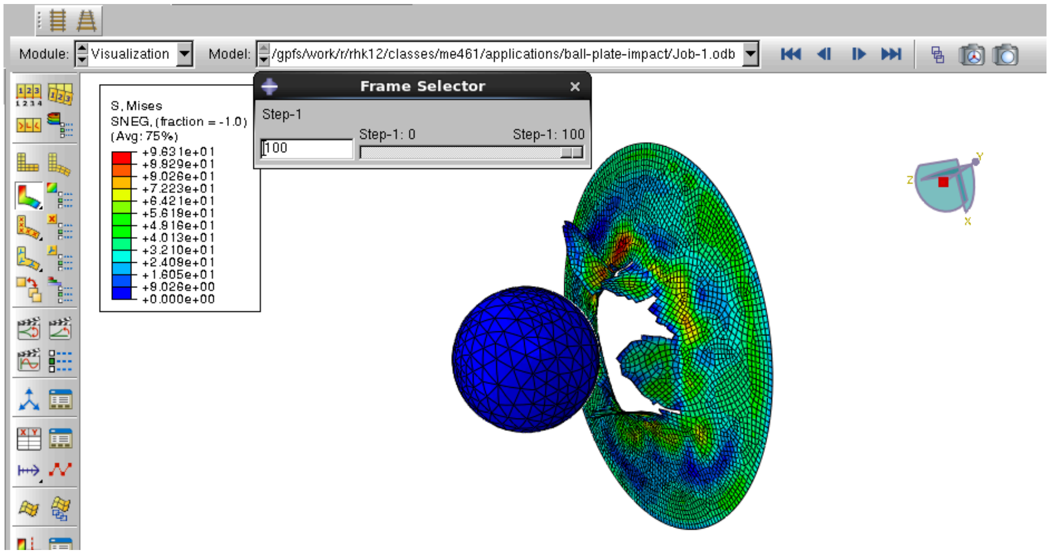
Once the job is complete, go to the visualization module and open the ODB file:





baqus/Explicit 6.14-2 Tue Mar 01 14:59:40 EST 2016

Use the frame selector to quickly move through the time steps:



In the undeformed configuration you can see the fully damaged/ deleted elements: