ABAQUS

Lecture 8

Impact and Postbuckling Analyses

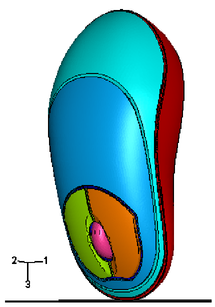
Overview

- Impact Analysis
- Geometric Imperfections for Postbuckling Analyses

Impact Analysis

Impact Analysis

- **Overview**
  - Why use ABAQUS/Explicit for impact?
  - Modeling Considerations (Example: Drop Test Simulation of a Cordless Mouse)
    - Initial model configuration
    - Elements
    - Materials
    - Constraints
    - Contact
    - Output
  - Other topics
    - Preloading
    - Submodeling
    - Mass scaling



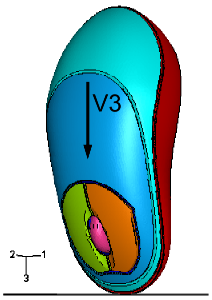
Cordless mouse assembly

Impact Analysis

- **Why use ABAQUS/Explicit for impact problems?**
  - A number of factors contribute to this decision:
    - High-speed dynamic response
    - Severe nonlinearities usually encountered:
      - Large number of contact constraints changing rapidly
      - Possible extensive plasticity
      - Possible structural collapse
    - Relatively short duration of simulation
      - Typically up to about 20 milliseconds for a drop test
      - Typically up to about 120 milliseconds for a full vehicle crash
    - Relatively large size of models

Impact Analysis

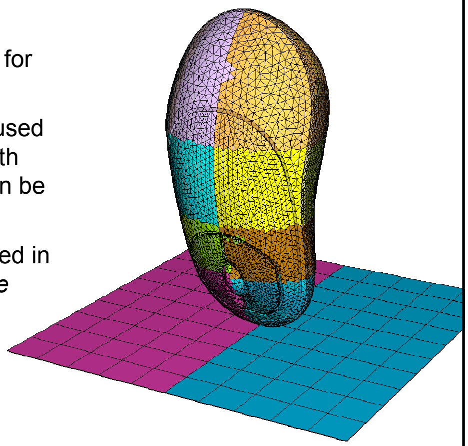
- **Example: Drop Test Simulation of a Cordless Mouse**
- **Initial model configuration**
  - Model built in “just before impact” position
  - Initial velocities applied to capture the effect of the drop
  - Changing drop height simply a case of changing initial velocity
  - Very simple to change drop orientation by adjusting floor position and changing initial velocity direction



Mouse initial velocity

Impact Analysis

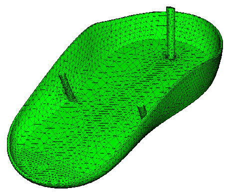
- For drop test simulations, the floor is generally rigid.
  - A meshed rigid body is required for general contact.
  - If domain parallelization will be used then the floor should meshed with enough elements such that it can be split between domains.
    - This functionality is discussed in Lecture 10, *Managing Large Models.*



Mouse drop assembly parallelization domains

Impact Analysis

- **Elements**
  - Modified tetrahedral elements (C3D10M) are typically used for complex geometries found in molded plastic parts.
    - Meshes are usually designed to have a single element through the thickness.
      - Reference: AUC 2000 – "Utilizing ABAQUS' 10-Node Modified Tet for Analyzing Impact Problems Involving Thin-Walled Structures", Motorola

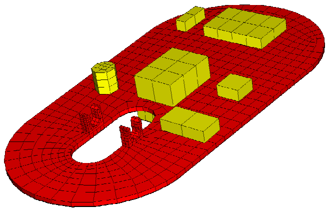


Mouse casing meshed with C3D10M elements

  - First-order tetrahedra (C3D4) are usually overly stiff, and extremely fine meshes would be required to obtain accurate results.
  - A good mesh of hexahedral elements (C3D8R) usually provides a solution of equivalent accuracy at less cost than tetrahedral elements.
    - C3D8R elements should be used where possible.

Impact Analysis

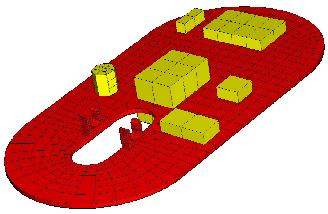
- Membrane elements are often used to coat solid element to extract detailed surface stress and strain data.
  - These can be used to represent strain gages and to extract results to compare with physical test data.
- Continuum shell elements may be used, where meshing allows, for parts whose main deformation mode is bending.
  - For example, circuit boards and electrical components
- Element selection is discussed in Lecture 2, *Elements*



Mouse circuit board

Impact Analysis

- Rigid bodies can be defined for stiff parts where stress and strain are not required.
  - The electrical components on the mouse circuit board (colored yellow) are modeled as rigid.
- The Part/Instance/Assembly format is very useful for building complex system-level models.
  - Component and sub-system models can be built and validated before being added into the system-level model.
  - This functionality is discussed in Lecture 10, *Managing Large Models.*



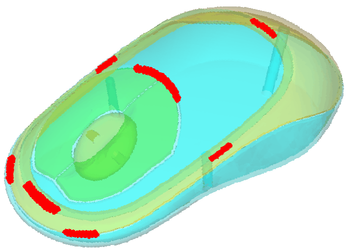
Mouse circuit board

Impact Analysis

- **Materials**
  - In the mouse model, all deformable components use elastic materials.
  - Constitutive models are available for all materials commonly found in impact analyses:
    - Elastic/plastic models for metals
    - Pressure-dependent plasticity models for thermoplastics
    - Hyperelastic models for solid rubbers
    - Hyperfoam models for foam rubbers
    - Failure models for vulnerability analysis
      - LCD and solder joints
      - This functionality is discussed in Lecture 9, *Material Damage and Failure*
    - Most materials can be made strain-rate and direction dependent.
  - The limiting factor is generally the availability of material data.
  - Material models are discussed in more detail in Lecture 3, *Materials.*

Impact Analysis

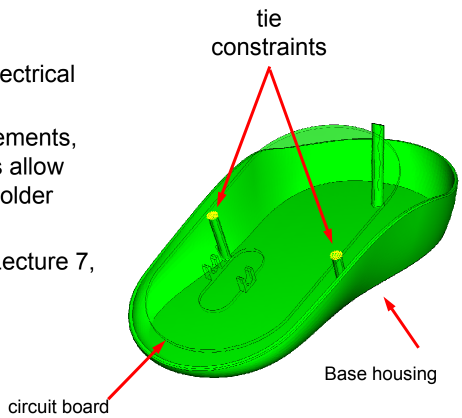
- **Constraints and connections in the mouse drop model**
  - Connector elements are used as a simple representation of the joints in the mouse model.
    - The button hinges and plastic clips are modeled as rigid connectors.
    - These connectors have failure defined at a critical force.



Regions of the mouse connected via connector elements

Impact Analysis

- Tie constraints are used to represent screw joints between the circuit board and the base housing.
  - These joints cannot fail.
- Ties are also used between the electrical components and the circuit board.
  - Regular elements, cohesive elements, or mesh-independent fasteners allow for more detailed modeling of solder joints.
- This functionality is discussed in Lecture 7, *Constraints and Connections.*

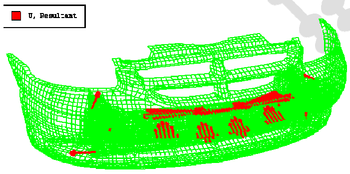


Impact Analysis

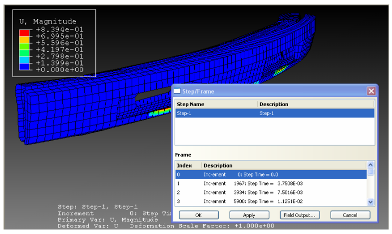
- **Modeling contact for impact analyses**
  - General contact is used in the computer mouse model.
    - An “ALL ELEMENT BASED” contact domain is used.
    - Default values are used for all contact settings.
    - This functionality is discussed in Lecture 4, *Contact Modeling*.
  - It is good practice to check the initial contact state.
    - This can be done from a datacheck analysis.
    - Need to check for:
      - Initial intersections leading to large adjustments
      - Interlocking surfaces
      - Surface thickness reductions
    - The large clearances between components in the mouse model avoid these problems.
  - Cell phones, lap tops, and digital cameras contain very tightly packed components, and very accurate CAD data are needed to avoid initial contact problems for these models.

Impact Analysis

- **Surface adjustments**
  - Overclosures generally mean that the geometry is wrong.
    - The only exception to this would be an interference fit, which would need special consideration as the adjustments done here are strain-free.
  - Symbol (vector) plots and contour plots of initial contact overclosure adjustments can be plotted at time zero using ABAQUS/Viewer.
  - Adjusted nodes are saved in the node set *InfoNodeOverclosureAdjust*.
  - Nodes with unresolved initial overclosures are saved in set *InfoNodeUnresolvInitOver*.
  - Examine status and message file for additional information.



Symbol plot of surface adjustments

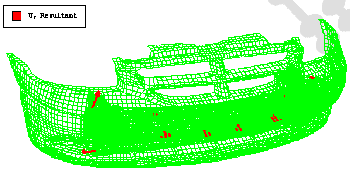


Contour plot of surface adjustments

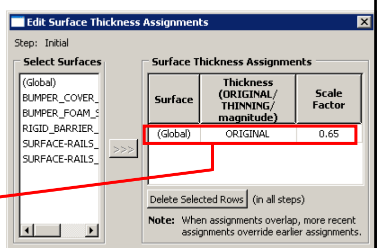
Impact Analysis

- The best way to avoid surface adjustments is to correct the geometry.
- However, in cases where the affected gemetry is not critical to the overall response of the model, other options are available.
  - It may acceptable to use the adjusted geometry.
  - For surfaces based on shell, membrane, and rigid elements the surface thickness used for general contact can be modified.

```
*SURFACE PROPERTY ASSIGNMENT,
   PROPERTY=THICKNESS
,ORIGINAL,0.65
```

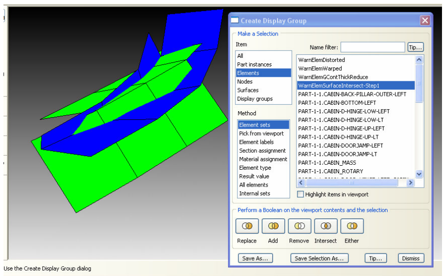


Reduced surface adjustments with shell contact thickness reduced



Impact Analysis

- **Surface intersections**
  - Surface intersections generally mean that the geometry is wrong.
  - Surfaces that are initially crossed are “locked” together for the duration of the analysis.
  - Surface intersections can be viewed by displaying element set *WarnElemSurfaceIntersect* using the Display Group dialog.
  - Examine status and message file for additional information.
  - It may be necessary to manually adjust the configuration of the intersecting surfaces
    - e.g., employing the *CONTACT CLEARANCE ASSIGNMENT option.



Intersecting surfaces displayed in ABAQUS/Viewer

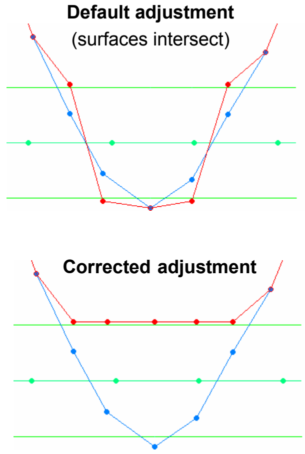
Impact Analysis

Green – master shell surface
Blue – original slave location
Red – adjusted slave location

- Example: Using *CONTACT CLEARANCE ASSIGNMENT to prevent entangled shell surfaces:
  - For the surfaces shown, the default adjustments made for entangled shell surfaces (assuming the master surface is fully constrained).
  - These adjustment can be corrected using the following clearance definition and assignment:

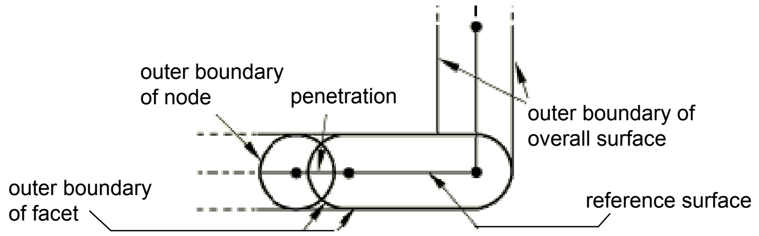
```
*CONTACT CLEARANCE, NAME=c2, ADJUST=YES,
SEARCH BELOW=1.5, CLEARANCE=0

*CONTACT
*CONTACT CLEARANCE ASSIGNMENT
<master_surf>, <slave_surf>, c2
```



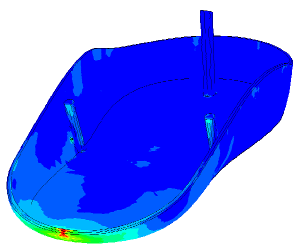
Impact Analysis

- **Surface thickness reductions**
  - ABAQUS may automatically reduce the contact thickness associated with structural elements to avoid issues of self-intersection.
  - Reducing the contact thickness of a surface may mean that contact occurs later than expected – think of a pinched shell.
  - Contact thickness reductions can be viewed by displaying element set *WarnElemGContThickReduce* using the Display Group dialog.
  - Examine status and message file for additional information.



Impact Analysis

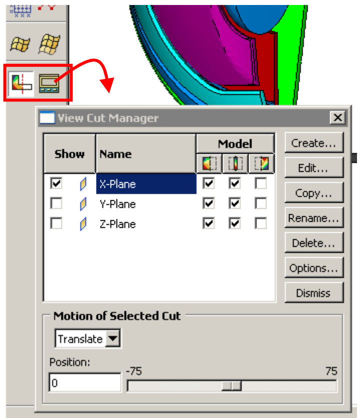
- **Typical results of interest from an impact analysis:**
  - Deformed shape animations
    - This allows for an assessment of which parts come into contact during the analysis.
  - Stresses and strains in critical components
    - This allows for an assessment of whether material yield or failure might occur.
  - Interface or joint loads and stresses
    - This allows for an assessment of whether joints or bonds might break.
    - If failure models are included in the analysis then parts can physically separate during the analysis.
      - In this case, failure could be detected by looking at the deformed shape.

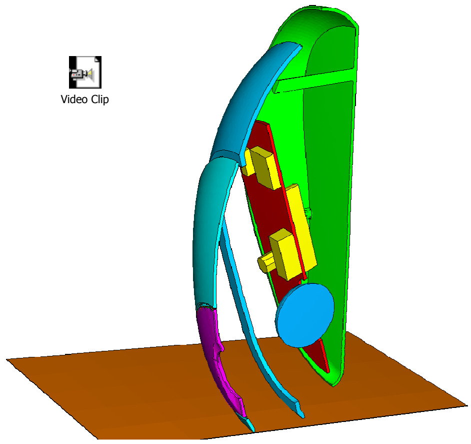


mouse base housing
Mises stress contours are shown at impact

Impact Analysis

- Cutting-planes are very useful for electronic drop test models.

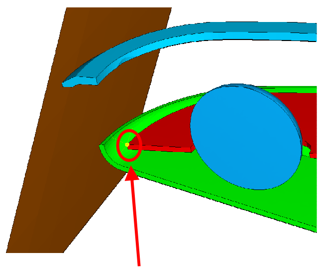


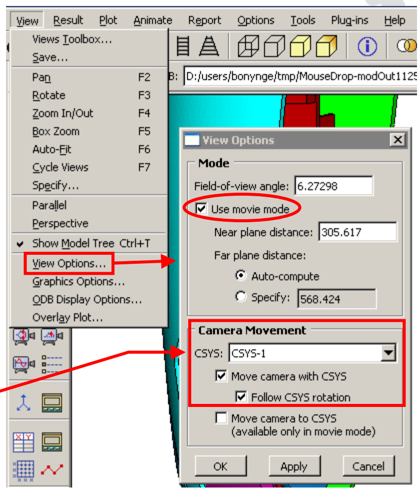


 ABAQUS

Impact Analysis

- Movie-mode animations are also very useful for these models.



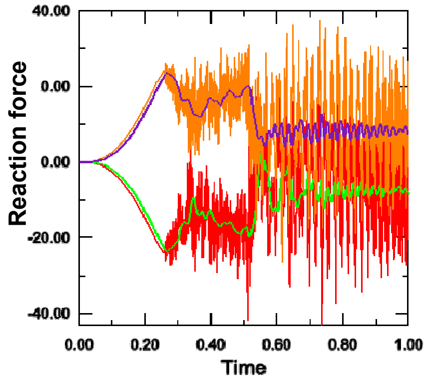


Camera moves with coordinate system CSYS-1, which is connected to the circled node on the circuit board.

 ABAQUS

Impact Analysis

- **Results of impact analyses are noisy and so care needs to be taken when looking at history plots.**
  - Pre-filtering of output can avoid issues with aliasing and decimation of results.
    - This functionality is discussed in Lecture 10, *Managing Large Models.*



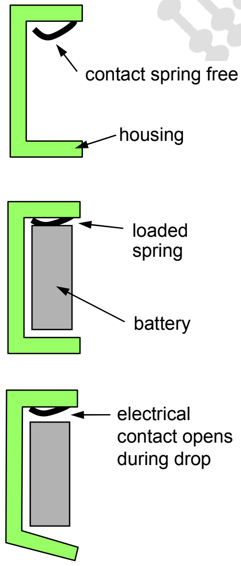
Comparison of raw history data with real-time filtered history data

Impact Analysis

- **Energy output**
  - Comparisons between different energy components can be used to evaluate whether an analysis is yielding a physically correct response.
  - The whole model energies are useful indicators of solution stability.
    - For example, an unrealistic growth in one of the energies may indicate that the analysis has become unstable.
    - Total energy in the model (ETOTAL) should remain approximately constant during the simulation.
      - The energy balance equation is discussed in Lecture 1, *Overview*.

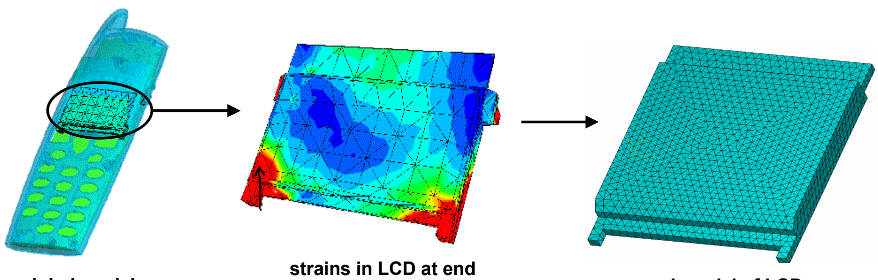
Impact Analysis

- **Preloading**
  - Various system-level components have preloads:
    - Snap-fit joints in plastic cases
    - Screw and bolt loads
    - Precompression of gaskets and seals
    - Deformation of battery contacts and other electrical joints
  - The effects can be included using *IMPORT.
    - The preload is best suited to ABAQUS/Standard.
    - Deformed shape and material state are imported into ABAQUS/Explicit as start point for impact analysis.
    - This functionality is discussed in Lecture 11, *ABAQUS/Explicit – ABAQUS/Standard Interface*.



Impact Analysis

- **Submodeling**
  - Submodeling allows detailed results to be extracted from a relatively coarse global model.
  - Submodel can be run as either a transient dynamic analysis or a static snapshot.
  - This functionality is discussed in Lecture 10, *Managing Large Models.*



global model | strains in LCD at end of global analysis | submodel of LCD

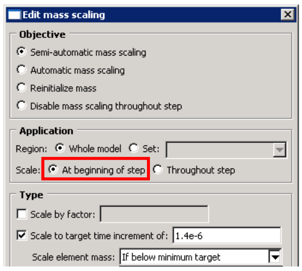
Impact Analysis

- **Mass Scaling in Impact Problems**
  - Mass scaling for truly dynamic events should almost always occur only for a limited number of elements and should never significantly increase the overall mass properties of the model.
    - The percentage change in total model mass caused by mass scaling can be found in the status (`.sta`) file.
  - Impact analyses generally use the mass scaling option to only scale the masses of elements whose element stable time increments are less than a specified value.
    - Other mass scaling options are discussed in Lecture 5, *Quasi-Static Analyses*.

Impact Analysis

- Complex models may contain a few very small elements, which forces ABAQUS/Explicit to use a small time increment to integrate the equations of motion in time.
  - These small elements are often the result of a difficult mesh generation task.
  - By scaling the masses of these elements at the beginning of the step, the stable time increment can be increased significantly, yet the effect on the overall dynamic behavior of the model may be negligible.
- Fixed mass scaling provides a simple means to modify the masses of a few small elements in a dynamic model so that they do not control the stable time increment size.

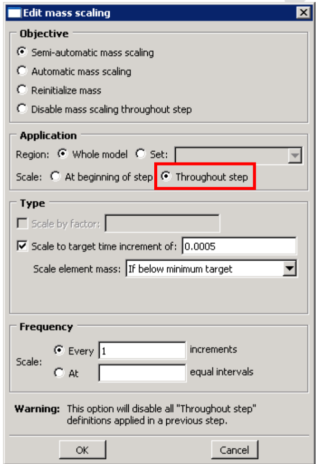
```
*FIXED MASS SCALING, TYPE=BELOW MIN,
 DT=1.4E-6, ELSET=elset
```



Impact Analysis

– When a structure impacts a fixed rigid body, severe element distortion often occurs.
  - Scaling the mass of the critical (smallest and most distorted) elements that are controlling the stable time increment for the model often can be done without significantly affecting the results.
    – The nodes associated with these elements often have very little mass and, hence, contribute very little kinetic energy to the model.
  - Mass scaling for this type of analysis must be done periodically during a step.

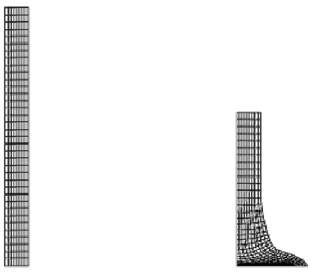
```
*VARIABLE MASS SCALING, FREQUENCY=1,
DT=0.0005, TYPE=BELOW MIN
```



ABAQUS

Impact Analysis

- **Example – Rod impact analysis**
  – Severe distortion of the elements near the impact zone would normally result in a dramatic reduction in the stable time increment.
  – Variable mass scaling is used to increase the mass of any elements with stable time increments below, 1e-8 seconds.



Undeformed and deformed shapes

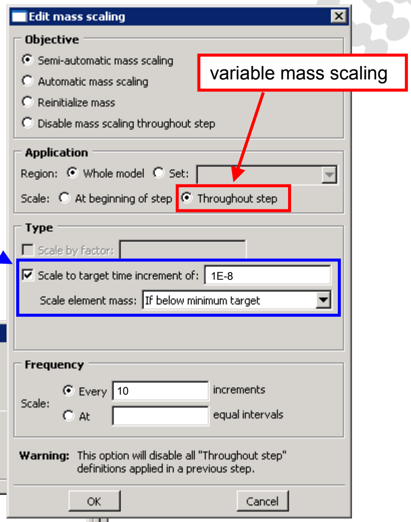
Video Clip

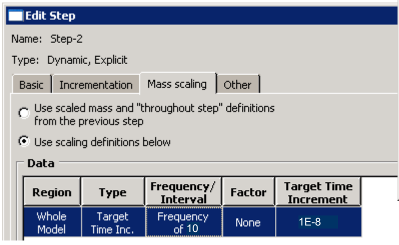
ABAQUS

Impact Analysis

```
*VARIABLE MASS SCALING, FREQUENCY=10,
 DT=1E-8, TYPE=BELOW MIN
```

Scale density of elements whose stable time increment is less than 1E-8 s





 ABAQUS

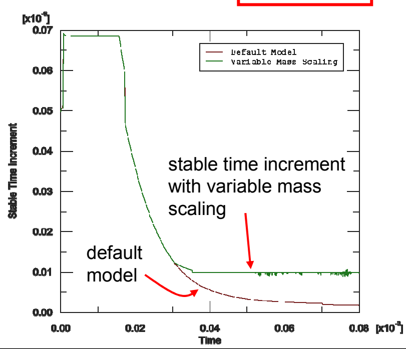
Impact Analysis

– From the status file, the maximum reported change in mass was just over 6%:

STEP INCREMENT	TOTAL TIME	CPU TIME	STABLE TIME	CRITICAL INCREMENT	KINETIC ELEMENT	PERCENT ENERGY	CHNG MASS
0	0.000E+00	0.000E+00	00:00:00	4.972E-08	1	2.375E+02	0.000E+00
:							
5767	8.000E-05	8.000E-05	00:00:04	1.000E-08	1	8.858E+00	6.260E+00

– Analysis results were unaffected by the added mass and the job ran almost twice as fast.

– Reference: ABAQUS Benchmark problem 1.3.10: *Impact of a copper rod*



 ABAQUS

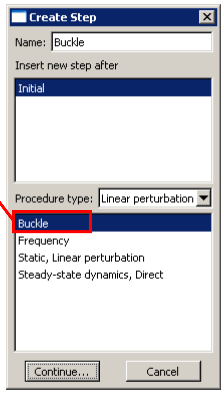
Geometric Imperfections for Postbuckling Analyses

Geometric Imperfections for Postbuckling Analyses

- **A geometric imperfection pattern is generally introduced into an otherwise "ideal" model for a postbuckling load-displacement analysis.**
  - ABAQUS offers three ways to define an imperfection:
    - As a linear superposition of buckling eigenmodes
    - From the displacements of an ABAQUS/Standard static analysis
    - By specifying the node number and imperfection values directly on the data lines
  - Only the translational degrees of freedom are modified.

Geometric Imperfections for Postbuckling Analyses

- **Using buckling modes for imperfections**
  - Using the buckling modes of the structure to define the imperfection in the model involves two analysis runs with the same model definition:
    - ABAQUS/Standard is used with the ∗BUCKLE procedure to establish the probable collapse modes.
    - ABAQUS/Explicit is used to perform the postbuckling analysis.
  - Write the buckling eigenmodes in the default global system to the results file using the ∗NODE FILE option with the (default) GLOBAL=YES parameter.
    - The ∗NODE FILE option is not currently supported by ABAQUS/CAE.
      - It can be included using the Keywords Editor.



 ABAQUS

Geometric Imperfections for Postbuckling Analyses

- Introduce an imperfection to the geometry in the ABAQUS/Explicit analysis by adding these buckling modes to the “perfect” geometry with the ∗IMPERFECTION option.

```
*IMPERFECTION, FILE=file, STEP=step [, NSET=nset]
<Mode number>, <Scaling factor for this mode>
:
```

  - The ∗IMPERFECTION option is not currently supported by ABAQUS/CAE.
    - It can be included using the Keywords Editor.
- The lowest buckling modes are assumed to provide the most critical imperfections, so they are usually given the largest scaling factor.
  - The scaled deformation patterns of the buckling modes are added to the perfect geometry to create the perturbed mesh.
  - The magnitudes of the perturbations used are typically a few percent of a relative structural dimension such as a beam cross-section or shell thickness.

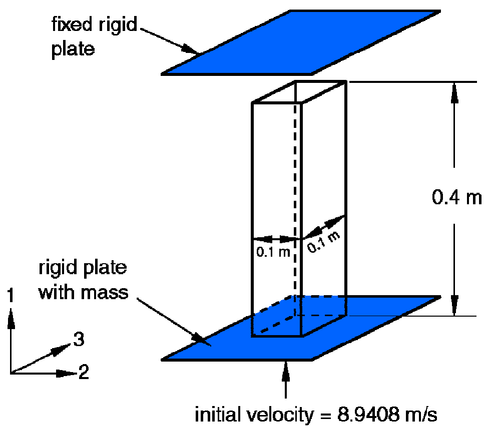
 ABAQUS

Geometric Imperfections for Postbuckling Analyses

- **Imperfection sensitivity**
  - The postbuckling behavior of some structures, such as thin-walled cylindrical shells, depends strongly on the imperfections in the original geometry.
    - Hence, imperfections based on a single buckling mode tend to yield nonconservative results for such structures.
  - By adjusting the magnitude of the scaling factors of the various buckling modes, the imperfection sensitivity of the structure can be assessed.
    - Normally, a number of analyses should be conducted to investigate the sensitivity of a structure to imperfections.

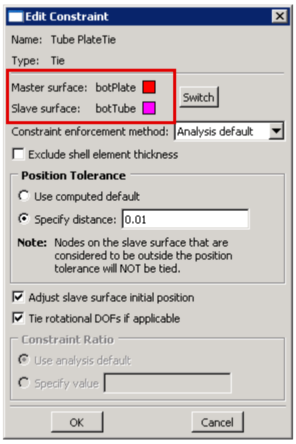
Geometric Imperfections for Postbuckling Analyses

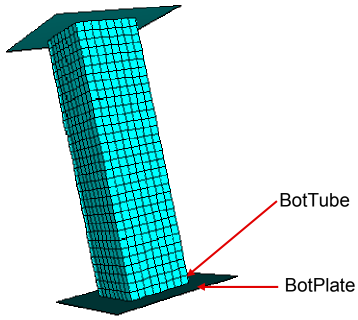
- **Example: Tube crush**
  - The crushing of a square, steel box tube between two rigid plates is considered.
    - The tube is rigidly attached to the bottom plate.
  - During impact, the tube dissipates a large amount of the initial kinetic energy into plastic deformation.
  - The primary interest of the analysis is to evaluate the tube's ability to absorb kinetic energy.



Geometric Imperfections for Postbuckling Analyses

- **The bottom of the tube is attached to the bottom plate via tie constraints.**

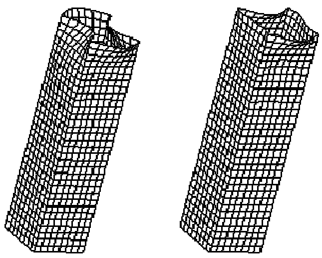




 ABAQUS

Geometric Imperfections for Postbuckling Analyses

– The tube's deformation will involve buckling.
  - It is necessary to seed the mesh with the lowest buckling modes of the tube (the imperfection factors) to obtain a smooth postbuckling response.
  - Thus, before performing the ABAQUS/Explicit crushing analysis an ABAQUS/Standard buckling analysis is performed to obtain the 10 lowest buckling modes of the tube.



First two buckling modes

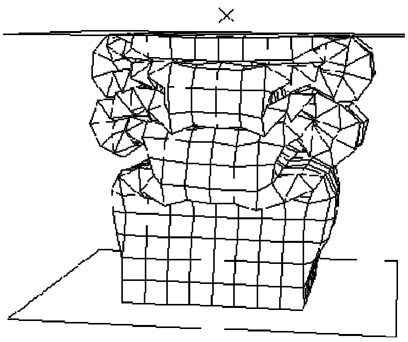
 ABAQUS

Geometric Imperfections for Postbuckling Analyses

- **Postbuckling response with imperfections**

```
*Imperfection, file=TubeBuckle, step=1
 1, 2.0E-5
 2, 0.25E-5
 3, 0.25E-5
 4, 0.18E-5
 5, 0.16E-5
 6, 0.10E-5
 7, 0.10E-5
 8, 0.08E-5
 9, 0.02E-5
10, 0.02E-5
```

  – These imperfections allow the postbuckling behavior proceed smoothly.



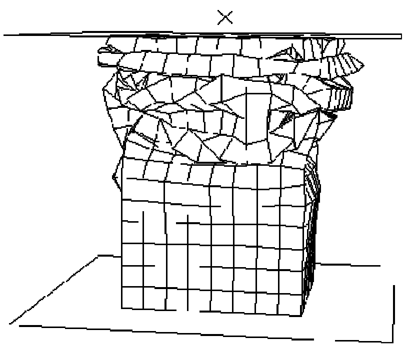
Final deformed mesh shows smooth folds, as expected.

 ABAQUS

Geometric Imperfections for Postbuckling Analyses

- **Postbuckling response without imperfections**
  – In contrast to the smoothly buckled shape of the perturbed mesh, the unperturbed mesh deforms into sharp folds.
    - The shape is clearly **not physically correct**.

Video Clip



 ABAQUS

Geometric Imperfections for Postbuckling Analyses

- **Energy histories**
  - By the end of the analysis 3600 J of energy has been dissipated as plastic deformation.
  - There is a corresponding decrease of 4400 J in the total kinetic energy of the model.