

Section 12.4

More Exercise: Disk and Block



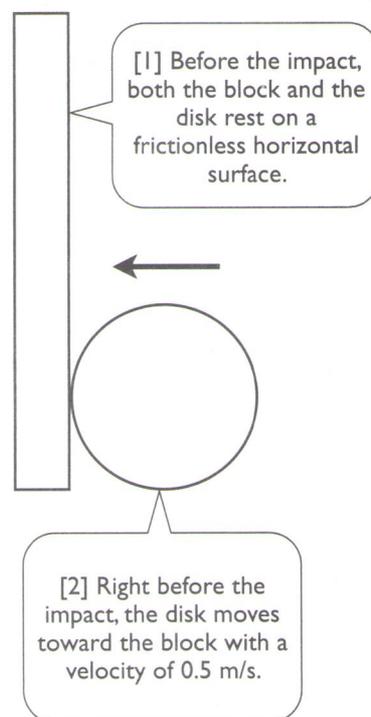
This exercise has two purposes: (1) to demonstrate how to apply a simple initial condition, namely uniform velocity, on a body, and (2) to show a limitation of <Transient Structural> analysis system for impact simulations, and to build a learning motivation for <Explicit Dynamics> (Chapter 15).

12.4-1 About the Disk and Block

Consider a disk of radius of 40 mm and a block of 200x20 mm², both have a thickness of 10 mm, on a frictionless horizontal surface [1]. Both are made of a very soft polymer of Young's modulus of 10 kPa, Poisson's ratio of 0.4, and mass density of 1000 kg/m³.

Right before the impact, the disk moves toward the block with a velocity of 0.5 m/s, and the positions of the disk and the block are as shown [1, 2].

We purposely use an extremely soft material (Young's modulus of 10 kPa) and consider a very slow-speed impact (velocity of 0.5 m/s) to relieve numerical difficulty. Increasing either of them will make the impact duration shorter, and in turn require a shorter integration time step (to find a solution). This will leave an exercise for you at the end of this chapter.



12.4-2 Start Up

Launch Workbench. Create a <Transient Structural> analysis system [1] by double-clicking it in <Toolbox>. Save the project as "Disk."

Double-click <Engineering Data> to prepare material data [2]. Create a new material, name it "Polymer" [3], and input the material properties as shown [4].

Return to <Project Schematic> and double-click <Geometry> to start up <DesignModeler>. Choose <Millimeter> as length unit.

| | | |
|---|----------------------|---|
| 1 | Transient Structural | ○ |
| 2 | Engineering Data | ○ |
| 3 | Geometry | ○ |
| 4 | Model | ○ |
| 5 | Setup | ○ |
| 6 | Solution | ○ |
| 7 | Results | ○ |

[1] Create a <Transient Structural> analysis system.

[2] Double-click <Engineering Data> to prepare material data.

Create a new material, name it "Polymer" [3], and input the material properties as shown [4]. Return to <Project Schematic> [5] and double-click <Geometry> to start up <DesignModeler>. Choose <Millimeter> as length unit.

Outline of Schematic A2: Engineering Data

| | A | B | C | D |
|---|----------------------------------|--------|-------------|------------------------|
| 1 | Contents of Engineering Data | | | |
| 2 | Material | Source | Description | |
| 3 | Structural Steel | | | stress |
| 4 | Polymer | | | tion 8, Div 2, Table 5 |
| * | Click here to add a new material | | | |

Properties of Outline Row 4: Polymer

| | A | B | C | D | E |
|---|----------------------|-------------------------------------|--------------------|---|---|
| 1 | Property | Value | Unit | | |
| 2 | Density | 1000 | kg m ⁻³ | | |
| 3 | Isotropic Elasticity | | | | |
| 4 | Derive from | Young's Modulus and Poisson's Ratio | | | |
| 5 | Young's Modulus | 10000 | Pa | | |
| 6 | Poisson's Ratio | 0.4 | | | |
| 7 | Bulk Modulus | 16667 | Pa | | |
| 8 | Shear Modulus | 3571.4 | | | |

[3] Create a new material and name it "Polymer."

[4] Input material properties as shown. Note that the SI units are used.

[5] Return to project.

12.4-3 Create Geometry in <DesignModeler>

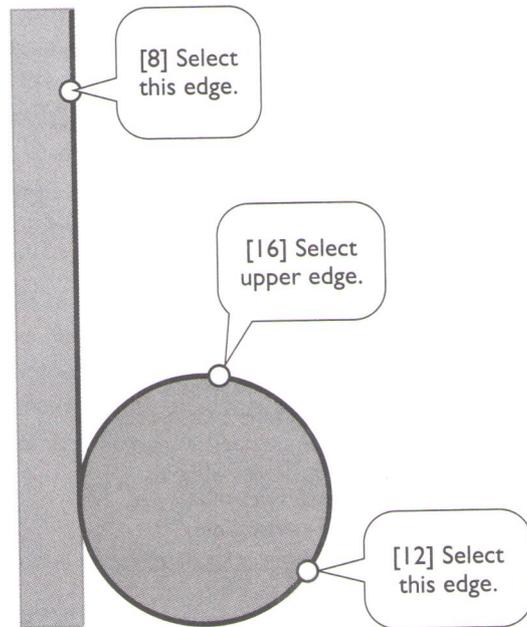
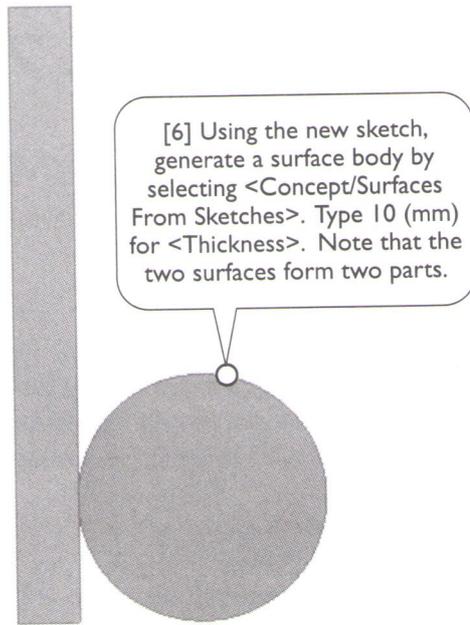
[1] On <XYPlane>, create a sketch for the block.

[2] Using the sketch, generate a surface body by selecting <Concept/Surfaces From Sketches>. Type 10 (mm) for <Thickness>.

[3] On <XYPlane>, create a new sketch and draw a circle for the disk. Remember to impose two <Tangent> constraints.

[4] Tangent.

[5] Tangent.

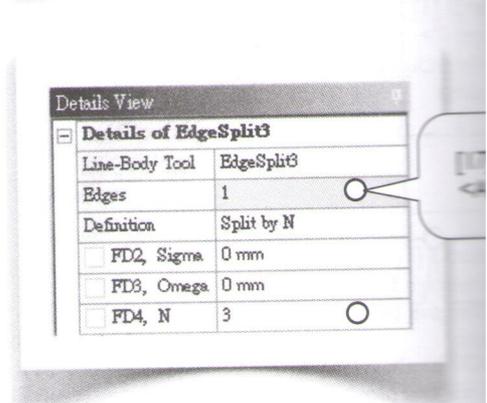
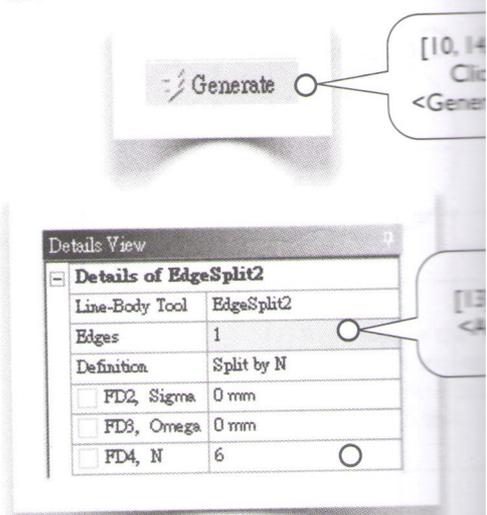
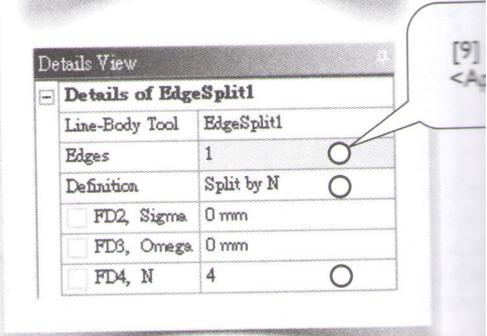
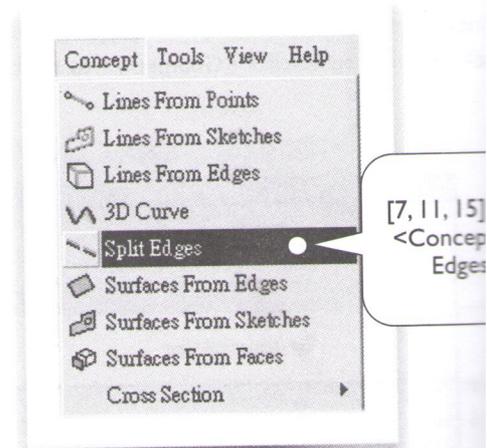


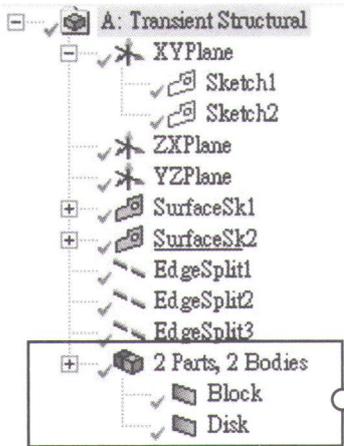
Why Split Edges?

The purpose of splitting the edges into segments is that, when meshed, each segment can have its own mesh density. Specifically, we need finer mesh around the contact region.

A Bug?

When I try to split the circular edge into 6 edges [11-14], oddly enough, it splits only the lower part of the circular edge. Therefore, I have to split the upper circular edge further into three edges [15-18]. Fortunately, we should be able to live with this kind of small problems.





[19] Rename two parts as shown.

| Advanced Geometry Options | | |
|---------------------------|--------------------------------|-------------------------------------|
| 10 | | |
| 11 | Analysis Type | 2D |
| 12 | Use Associativity | <input checked="" type="checkbox"/> |
| 13 | Import Coordinate Systems | <input checked="" type="checkbox"/> |
| 14 | Import Work Points | <input type="checkbox"/> |
| 15 | Reader Mode Saves Updated File | <input type="checkbox"/> |
| 16 | Import Using Instances | <input checked="" type="checkbox"/> |
| 17 | Smart CAD | <input type="checkbox"/> |
| 18 | Enclosure | <input checked="" type="checkbox"/> |
| 19 | Decompo | <input checked="" type="checkbox"/> |

[20] Close <DesignModeler>. Before starting up <Mechanical>, remember to set <Analysis Type> to <2D> (3.1-4[4-6]).

12.4-4 Simulation in <Mechanical>

| Details of "Geometry" | |
|----------------------------------|--------------------------------------|
| Definition | |
| Source | E:\ANSYS\Book14\Sec12.4\Disk_file... |
| Type | DesignModeler |
| Length Unit | Millimeters |
| Element Control | Program Controlled |
| 2D Behavior | Plane Stress |
| Display Style | Body Color |
| Bounding Box | |
| Properties | |
| Statistics | |
| Basic Geometry Options | |
| Advanced Geometry Options | |

[1] Start up <Mechanical>. Highlight <Geometry>, make sure <Plane Stress> is selected for <2D Behavior>.

| Details of "Disk" | |
|-------------------------------------|---------------------------|
| Graphics Properties | |
| Definition | |
| <input type="checkbox"/> Suppressed | No |
| Stiffness Behavior | Flexible |
| Coordinate System | Default Coordinate System |
| Reference Temperature | By Environment |
| <input type="checkbox"/> Thickness | 10. mm |
| Thickness Mode | Refresh on Update |
| Material | |
| Assignment | Polymer |
| Nonlinear Effects | Yes |
| Thermal Strain Effects | Yes |
| Bounding Box | |
| Properties | |
| Statistics | |

[2] Highlight <Geometry/Disk>, select <Polymer> as the material for the body. Note that mm-N-s unit system is used.

| Details of "Block" | |
|-------------------------------------|---------------------------|
| Graphics Properties | |
| Definition | |
| <input type="checkbox"/> Suppressed | No |
| Stiffness Behavior | Flexible |
| Coordinate System | Default Coordinate System |
| Reference Temperature | By Environment |
| <input type="checkbox"/> Thickness | 10. mm |
| Thickness Mode | Refresh on Update |
| Material | |
| Assignment | Polymer |
| Nonlinear Effects | Yes |
| Thermal Strain Effects | Yes |
| Bounding Box | |
| Properties | |
| Statistics | |

[3] Highlight <Geometry/Block>, select <Polymer> as the material for the body.

Details of "Frictionless - Block To Disk"

| | |
|---------------------------------|------------------------|
| Scope | |
| Scoping Method | Geometry Selection |
| Contact | 2 Edges |
| Target | 2 Edges |
| Contact Bodies | Block |
| Target Bodies | Disk |
| Definition | |
| Type | Frictionless |
| Scope Mode | Automatic |
| Behavior | Program Controlled |
| Suppressed | No |
| Advanced | |
| Formulation | Program Controlled |
| Detection Method | Program Controlled |
| Interface Treatment | Add Offset, No Ramping |
| <input type="checkbox"/> Offset | 0. mm |
| Normal Stiffness | Program Controlled |
| Update Stiffness | Each Iteration |
| Stabilization Damping Factor | 0. |
| Pinball Region | Program Controlled |
| Time Step Controls | None |

[5] Change the contact type to <Frictionless>.

[6] Update stiffness at each equilibrium iteration. It usually helps when contacts are involved.

[4] Highlight <Connection/Contacts/Contact Region>. The contact region is correctly detected by Workbench.

Mesh Control | Metric Gr

| |
|---------------------|
| Method |
| Mesh Group |
| Sizing |
| Contact Sizing |
| Refinement |
| Mapped Face Meshing |
| Match Control |
| Pinch |
| Inflation |
| Sharp Angle |
| Gap Tool |

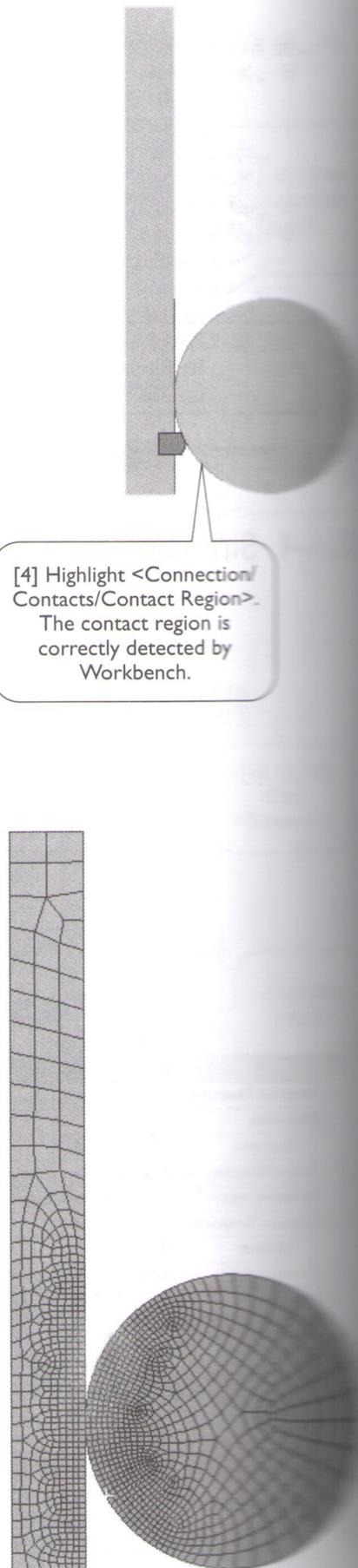
[7] With <Mesh> highlighted, select <Mesh Control/Sizing>.

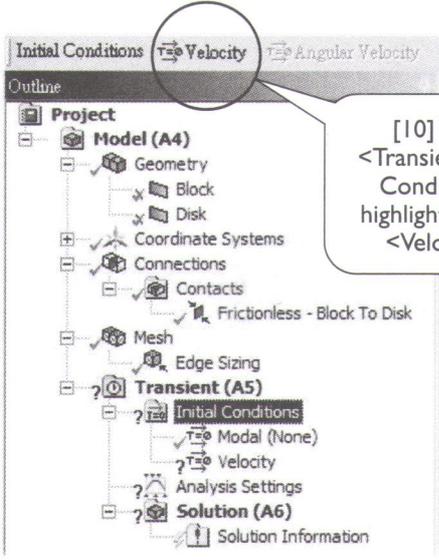
[8] Select all four edges in the contact region (two in the block and two in the disk, see [4]).

Details of "Edge Sizing" - Sizing

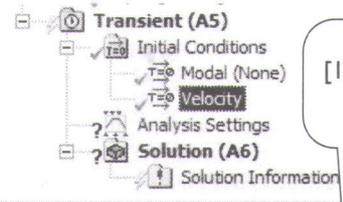
| | |
|---|--------------------|
| Scope | |
| Scoping Method | Geometry Selection |
| Geometry | 4 Edges |
| Definition | |
| Suppressed | No |
| Type | Element Size |
| <input type="checkbox"/> Element Size | 1. mm |
| Behavior | Soft |
| <input type="checkbox"/> Curvature Normal Angle | Default |
| <input type="checkbox"/> Growth Rate | Default |
| Bias Type | No Bias |

[9] Type 1 (mm) for <Element Size>. Generate mesh.





[10] With <Transient/Initial Conditions> highlighted, click <Velocity>.



[11] Select the disk body.

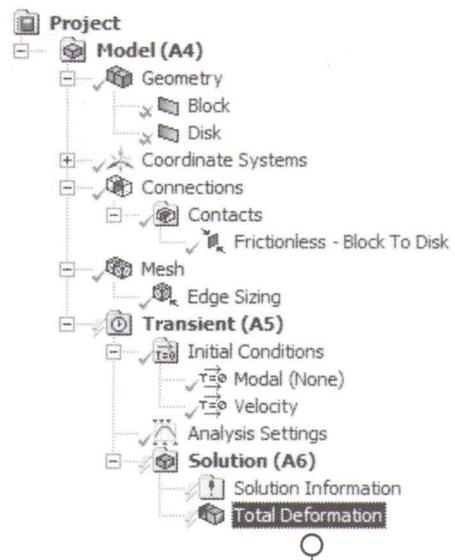
Details of "Velocity"

| | |
|--------------------------------------|--------------------------|
| Scope | |
| Scoping Method | Geometry Selection |
| Geometry | 1 Body |
| Definition | |
| Input Type | Velocity |
| Define By | Components |
| Coordinate System | Global Coordinate System |
| <input type="checkbox"/> X Component | -500. mm/s |
| <input type="checkbox"/> Y Component | 0. mm/s |
| Suppressed | No |

Details of "Analysis Settings"

| | |
|---------------------------------|-----------------------|
| Step Controls | |
| Number Of Steps | 1. |
| Current Step Number | 1. |
| Step End Time | 0.2 s |
| Auto Time Stepping | On |
| Define By | Time |
| Initial Time Step | 1.e-004 s |
| Minimum Time Step | 1.e-005 s |
| Maximum Time Step | 1.e-003 s |
| Time Integration | On |
| Solver Controls | |
| Solver Type | Program Controlled |
| Weak Springs | Program Controlled |
| Large Deflection | On |
| Restart Controls | |
| Nonlinear Controls | |
| Output Controls | |
| Stress | No |
| Strain | No |
| Nodal Forces | No |
| Contact Miscellaneous | No |
| General Miscellaneous | No |
| Calculate Results At | Equally Spaced Points |
| - Value | 100. |
| Max Number of Result Sets | Program Controlled |
| Damping Controls | |
| Analysis Data Management | |

[12] Highlight <Analysis Settings>. Type 0.2 (s) for <Step End Time>, 0.0001 (s) for <Initial Time Step>, 0.00001 (s) for <Minimum Time Step>, and 0.001 (s) for <Maximum Time Step>.

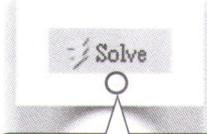


[14] Insert a <Total Deformation>.

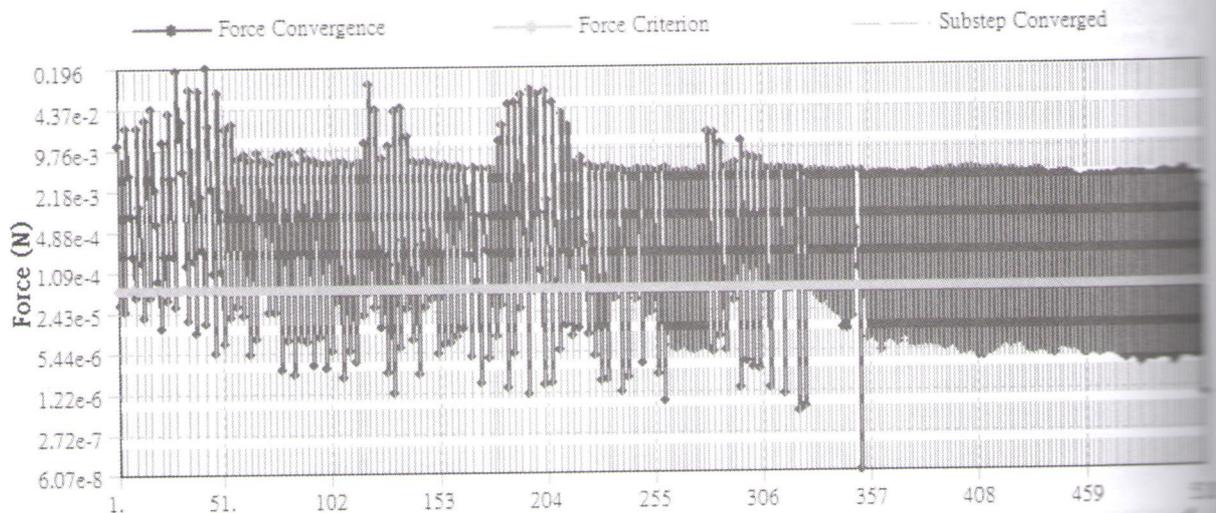
Details of "Solution Information"

| | |
|-----------------------------|-------------------|
| Solution Information | |
| Solution Output | Force Convergence |
| Newton-Raphson Residuals | 0 |
| Update Interval | 2.5 s |
| Display Points | All |

[15] With <Solution/Solution Information> highlighted, select <Force Convergence>, to watch how the solution proceeds.

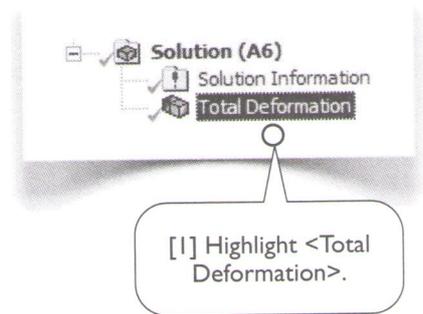


[16] Click <Solve>.

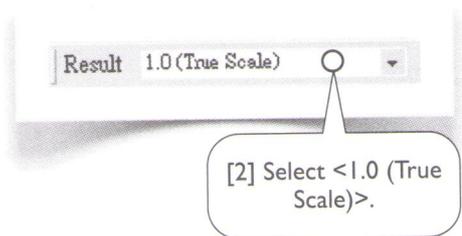


[17] It takes many iterations to complete the simulations.

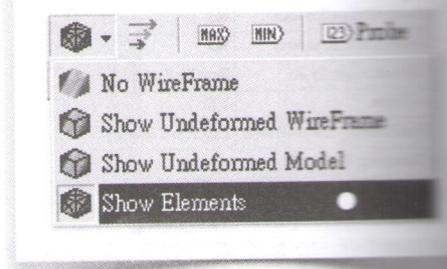
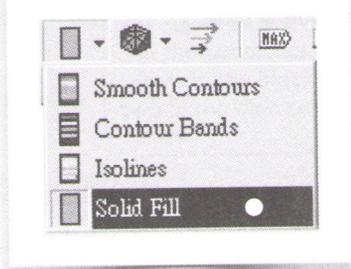
12.4-5 Animate the Impact



[1] Highlight <Total Deformation>.



[2] Select <1.0 (True Scale)>.



[4] Click <Play>.

[3] Select <Result Sets>.