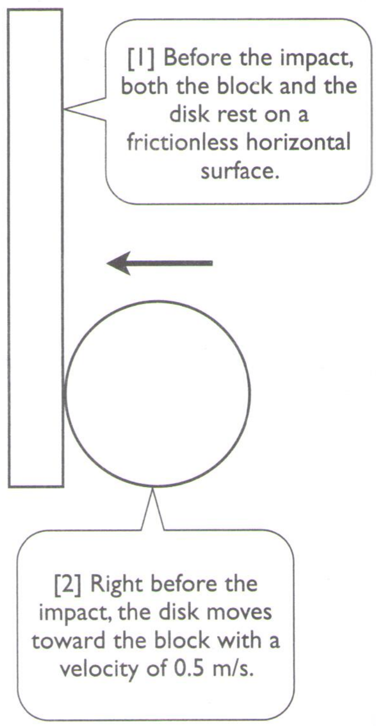
Section 12.4

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 mm^2, 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/m^3.

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].

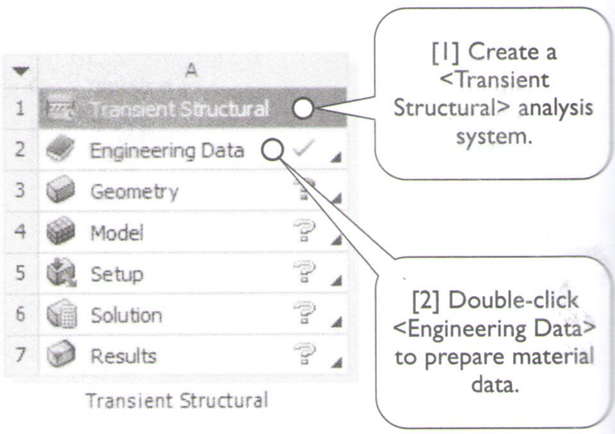
We purposely use an extremely soft material (Young's modulus of 10 kPa) and consider a very slow-speed impact (velocity of 0.5 m/s) to relieve numerical difficulty. Increasing either of them will make the impact duration shorter, and in turn require a shorter integration time step (to find a solution).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

12.4-2 Start Up

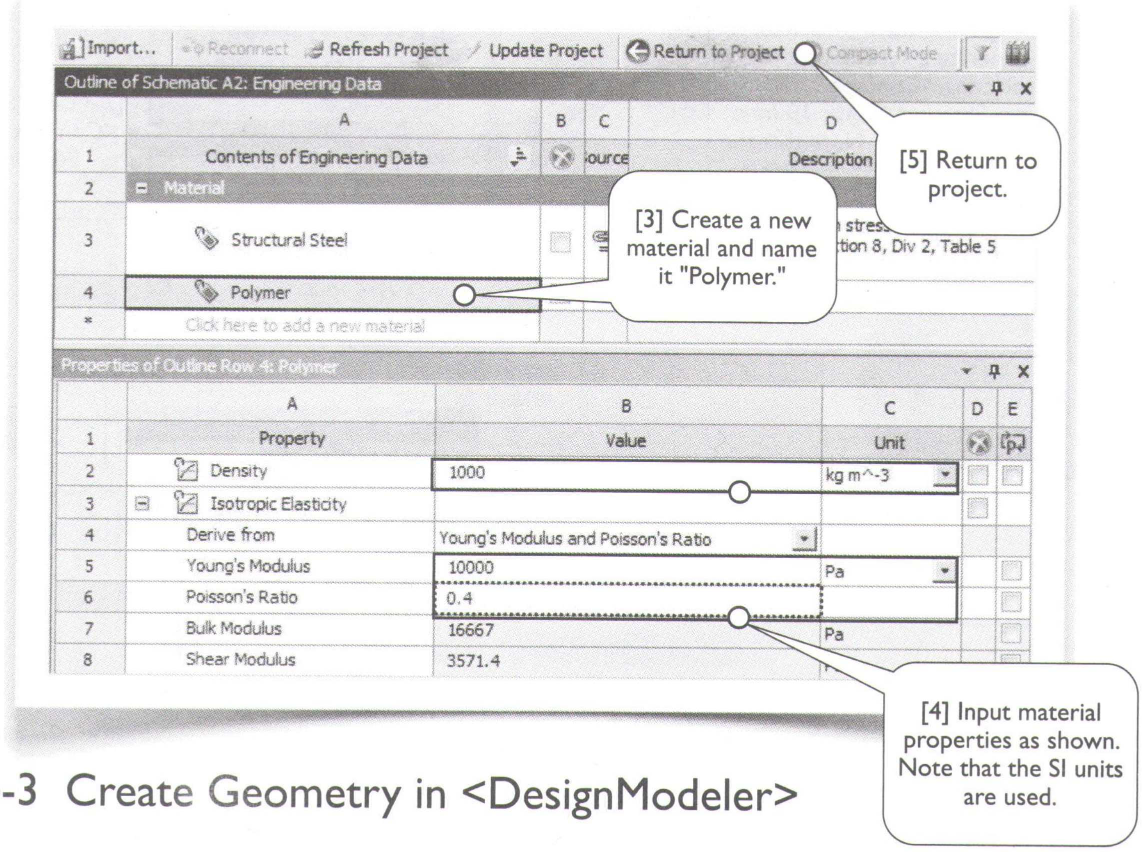
Launch Workbench. Create a <Transient Structural> analysis system [1] by double-clicking it in <Toolbox>. Save the project as "Disk."

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].

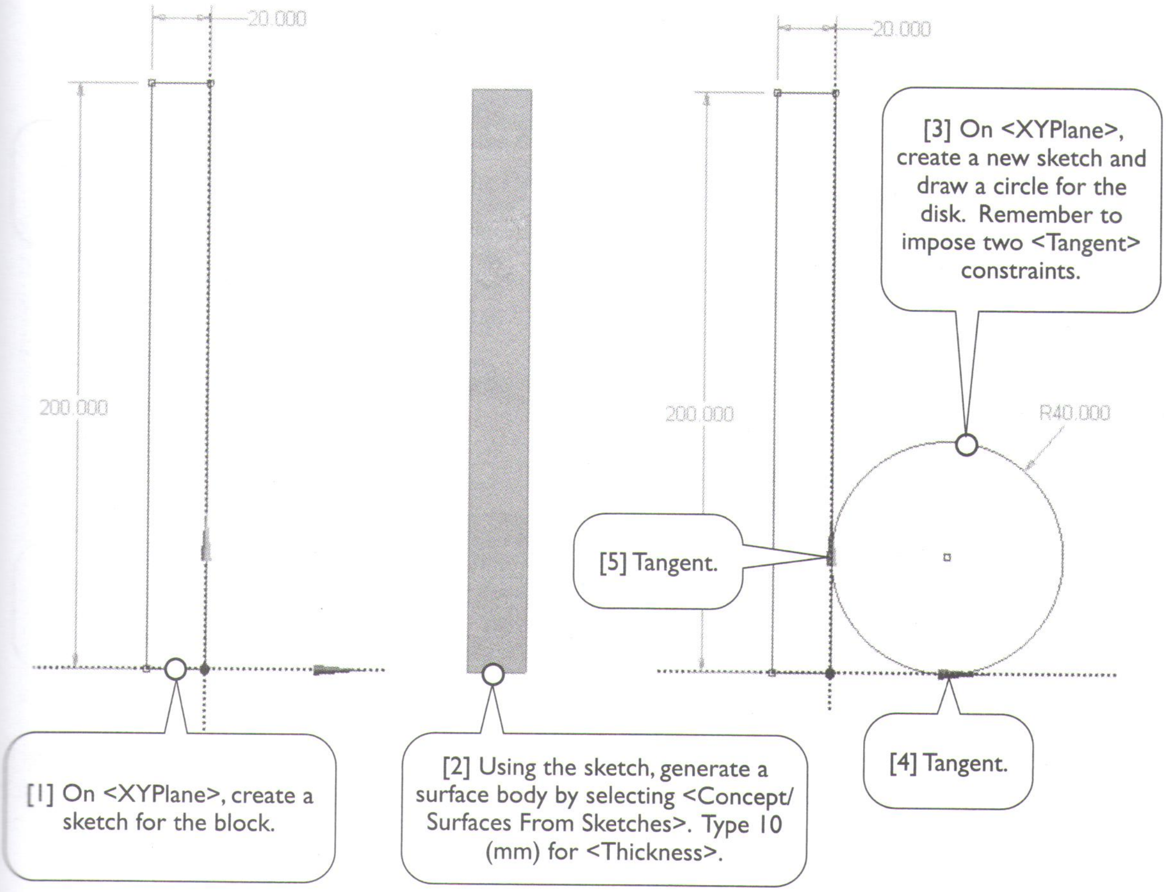
Return to <Project Schematic> and double-click <Geometry> to start up <DesignModeler>. Choose <Millimeter> as length unit.

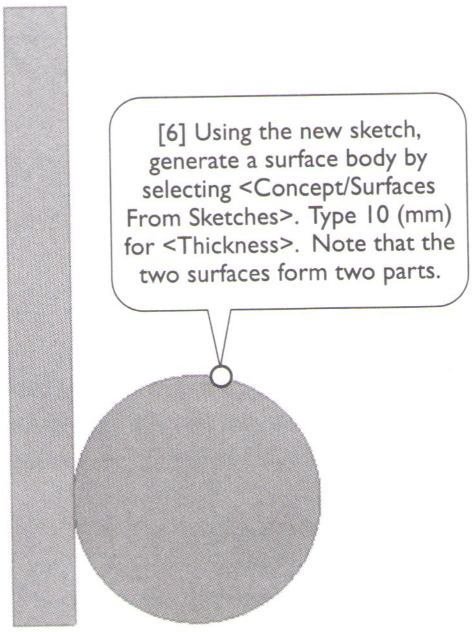


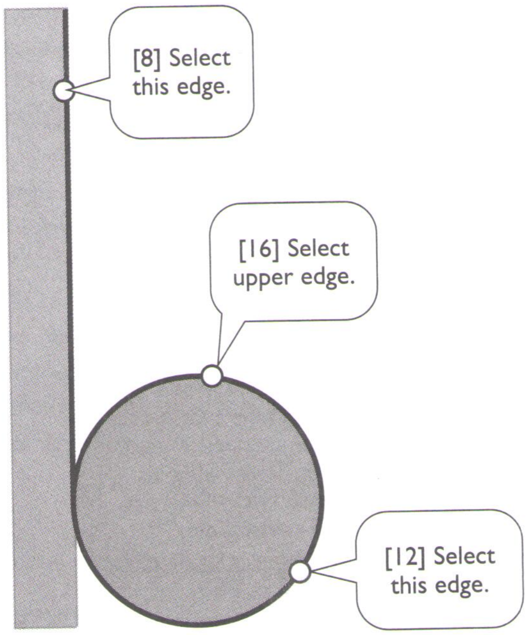
Create a new material, name it "Polymer" [3], and input the material properties as shown [4]. Return to <Project Schematic> [5] and double-click <Geometry> to start up <DesignModeler>. Choose <Millimeter> as length unit.



12.4-3 Create Geometry in <DesignModeler>





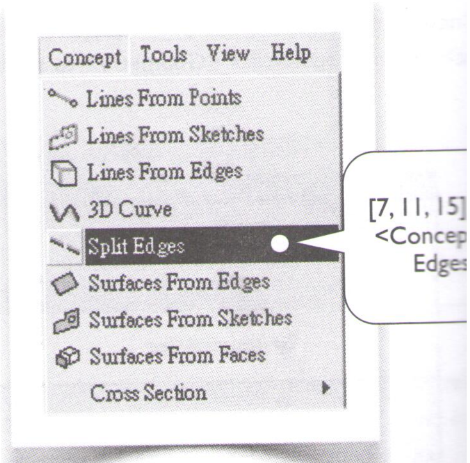


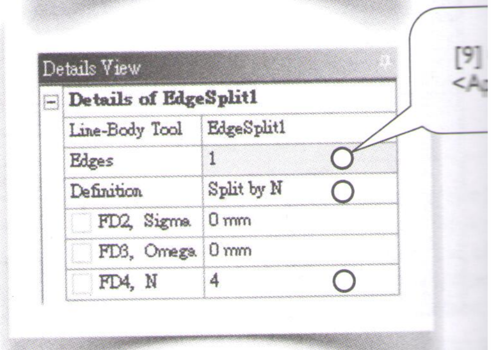
Why Split Edges?

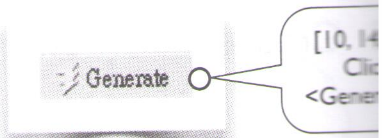
The purpose of splitting the edges into segments is that, when meshed, each segment can have its own mesh density. Specifically, we need finer mesh around the contact region.

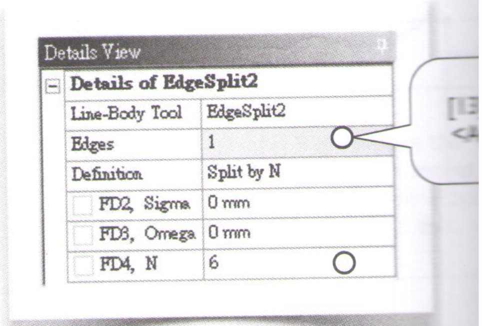
A Bug?

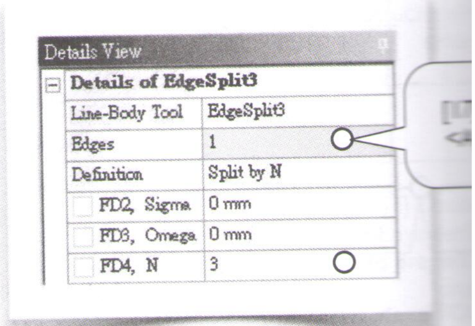
When I try to split the circular edge into 6 edges [11-14], oddly enough, it splits only the lower part of the circular edge. Therefore, I have to split the upper circular edge further into three edges [15-18]. Fortunately, we should be able to live with this kind of small problems.

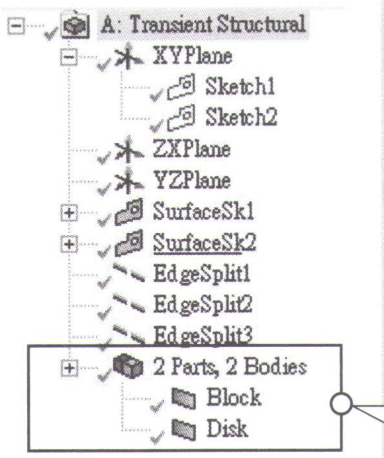




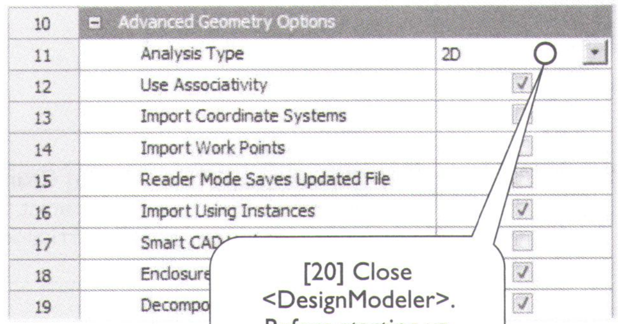






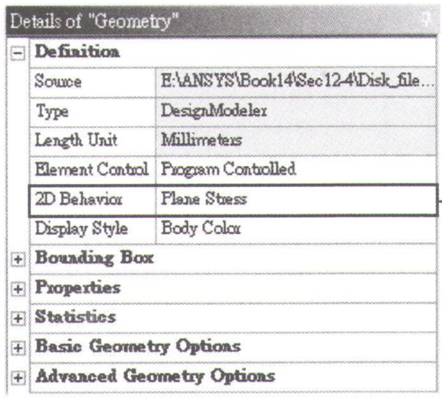


[19] Rename two parts as shown.

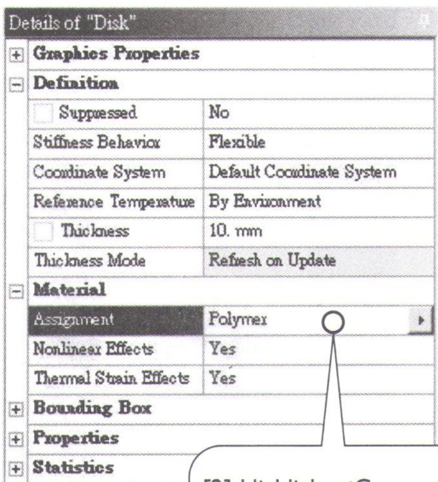


[20] Close <DesignModeler>. Before starting up <Mechanical>, remember to set <Analysis Type> to <2D> (3.1-4[4-6]).

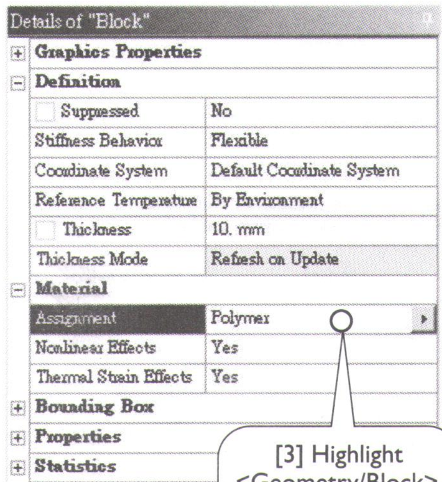
12.4-4 Simulation in <Mechanical>



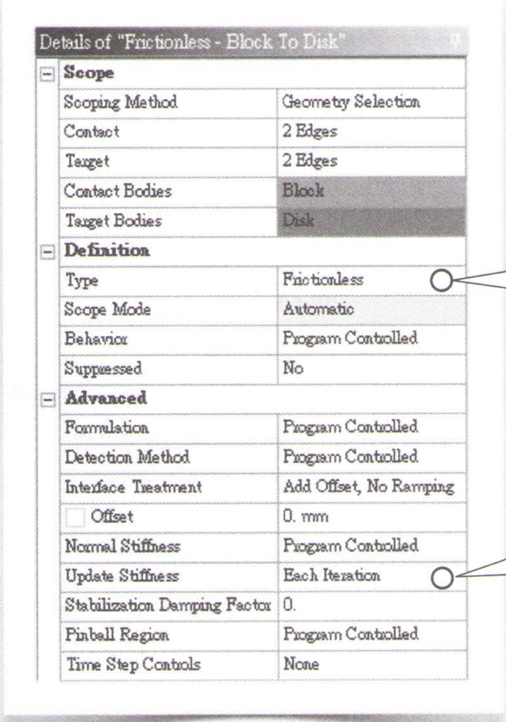
[1] Start up <Mechanical>. Highlight <Geometry>, make sure <Plane Stress> is selected for <2D Behavior>.



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



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

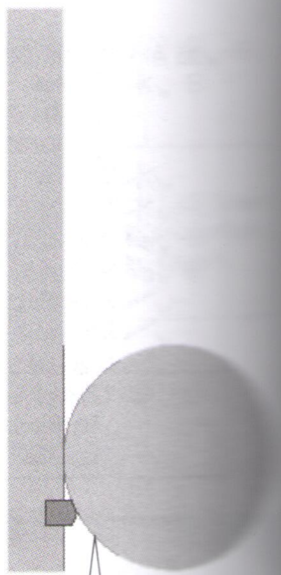


Details of "Frictionless - Block To Disk"

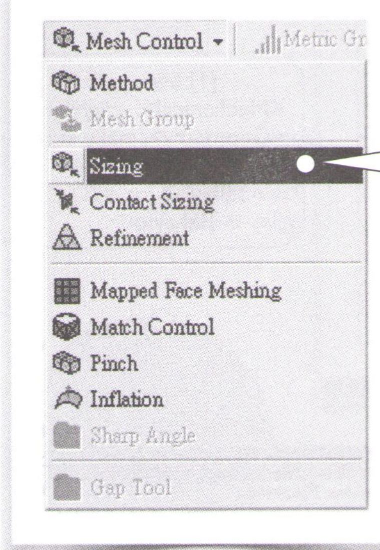
Scope	
Scoping Method	Geometry Selection
Contact	2 Edges
Target	2 Edges
Contact Bodies	Block
Target Bodies	Disk
Definition	
Type	Frictionless
Scope Mode	Automatic
Behavior	Program Controlled
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Interface Treatment	Add Offset, No Ramping
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.

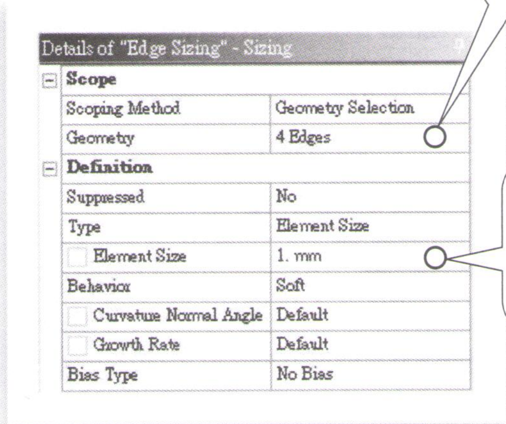


[4] Highlight <Connection/Contacts/Contact Region>. The contact region is correctly detected by Workbench.



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

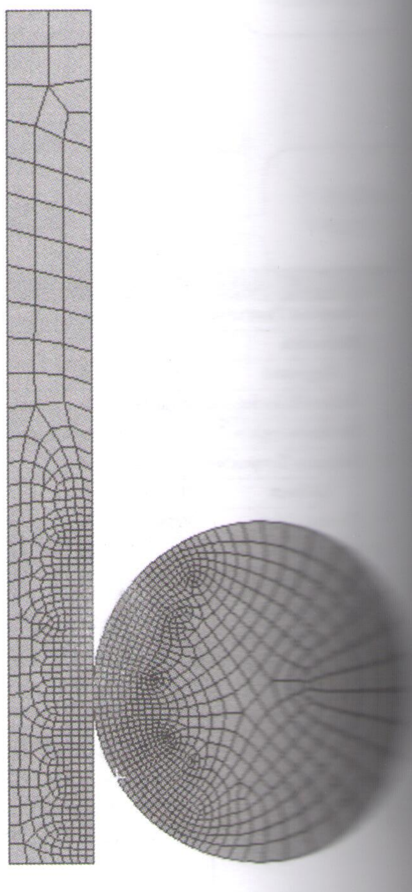
[8] Select all four edges in the contact region (two in the block and two in the disk, see [4]).

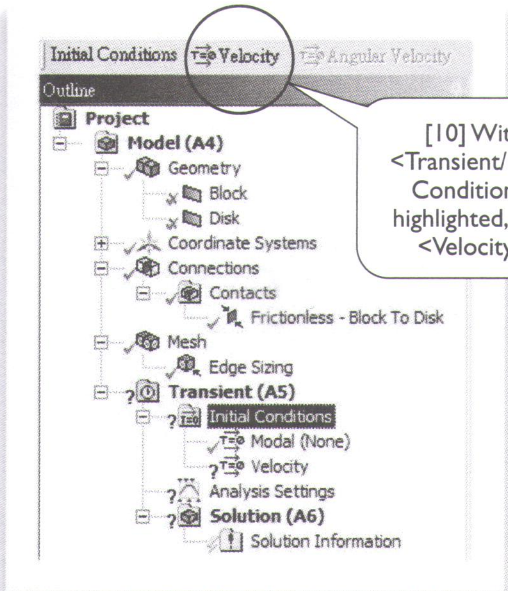


Details of "Edge Sizing" - Sizing

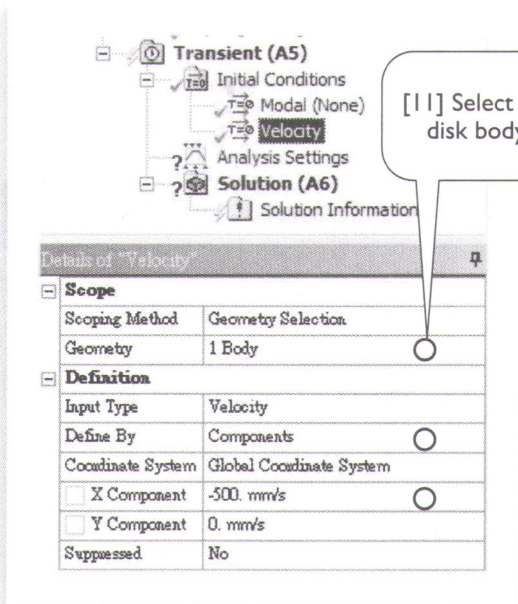
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Suppressed	No
Type	Element Size
Element Size	1. mm
Behavior	Soft
Curvature Normal Angle	Default
Growth Rate	Default
Bias Type	No Bias

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

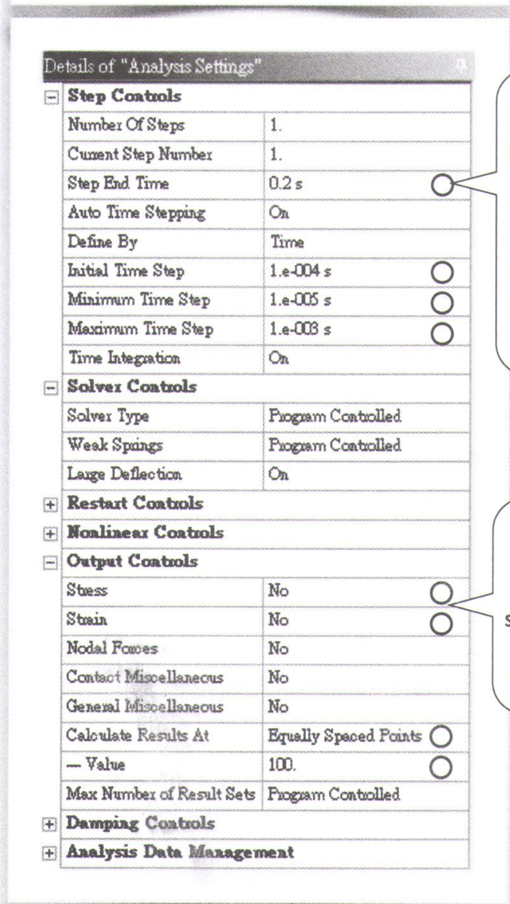




[10] With <Transient/Initial Conditions> highlighted, click <Velocity>.

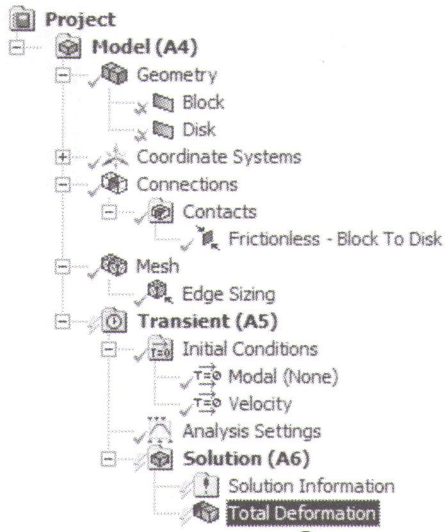


[11] Select the disk body.

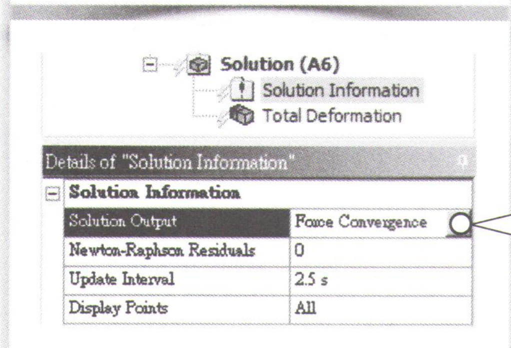


[12] Highlight <Analysis Settings>. Type 0.2 (s) for <Step End Time>, 0.0001 (s) for <Initial Time Step>, 0.00001 (s) for <Minimum Time Step>, and 0.001 (s) for <Maximum Time Step>.

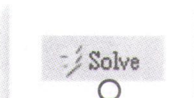
[13] Set up <Output Controls> to save data storage space and computing time.



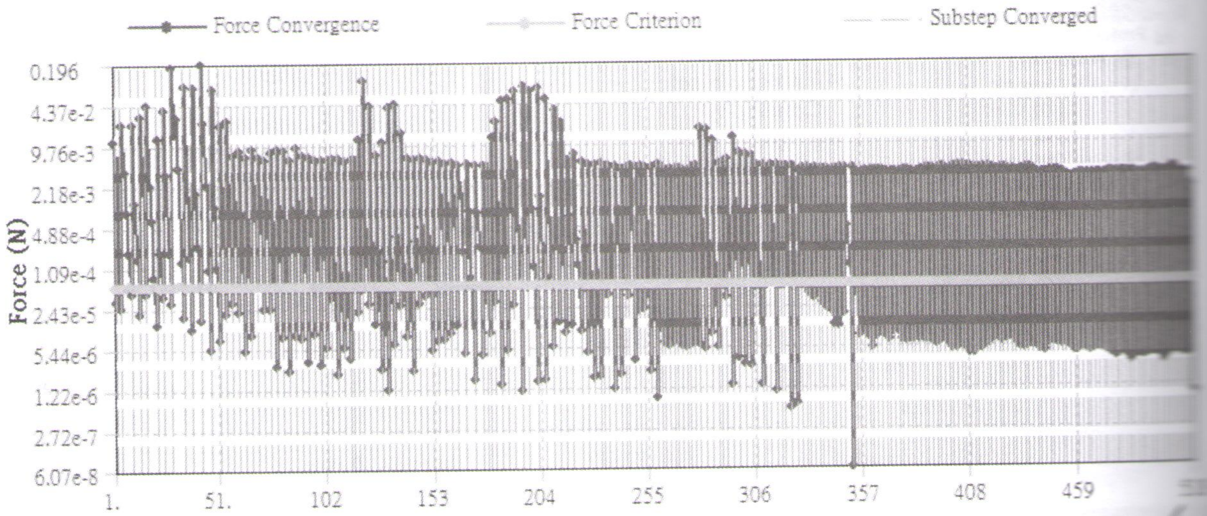
[14] Insert a <Total Deformation>.



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

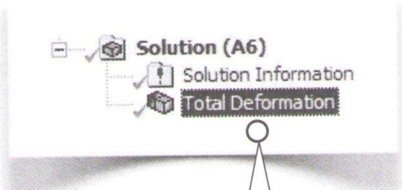


[16] Click <Solve>.

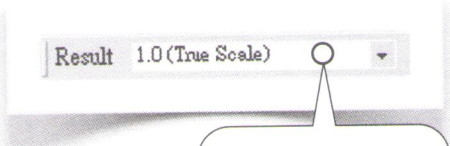


[17] It takes many iterations to complete the simulations.

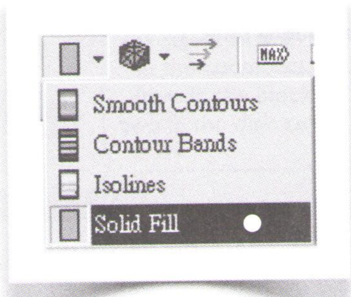
12.4-5 Animate the Impact

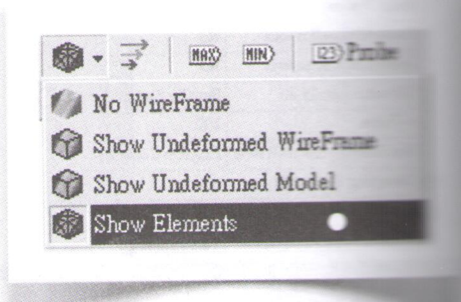


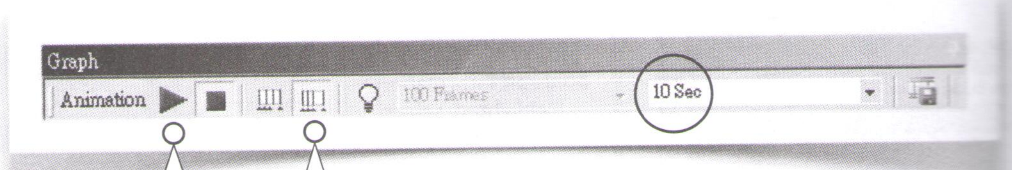
[1] Highlight <Total Deformation>.



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







[4] Click <Play>.

[3] Select <Result Sets>.