MBD for ANSYS
Multi-Body Dynamics

MBD for ANSYS Tutorial ANSYS 15

Hoekens Mechanism

FunctionBay

Registered Trademarks of FunctionBay, Inc. or Subsidiary

RecurDyn is a registered trademark of FunctionBay, Inc.

Multi-Body Dynamics for Ansys, MBD for Ansys, RecurDyn™/Professional, RecurDyn™/Solver, RecurDyn™/Modeler, RecurDyn™/FFlex, RecurDyn™/RFlex, RecurDyn™/Linear, RecurDyn™/CoLink, RecurDyn™/AutoDesign, RecurDyn™/Control, RecurDyn™/Hydraulic, RecurDyn™/Tire, RecurDyn™/Driver, RecurDyn™/EHD, RecurDyn™/MeshInterface, RecurDyn™/Chain, RecurDyn™/Track HM, RecurDyn™/Track LM, RecurDyn™/MTT2D, RecurDyn™/MTT3D, RecurDyn™/Belt, RecurDyn™/HAT, RecurDyn™/Gear, RecurDyn™/Crank, RecurDyn™/Piston, RecurDyn™/Bearing, RecurDyn™/Valve, RecurDyn™/Spring, RecurDyn™/TimingChain, RecurDyn™/Engine, RecurDyn™/ECFD_HD, RecurDyn™/Ch Script are trademarks of FunctionBay, Inc.

Third-Party Trademarks

ANSYS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries.
Windows 7 and Windows 8 are registered trademarks of Microsoft Corporation.
FLEXlm is a registered trademark of GLOBEtrotter Software, Inc.
All other brand or product names are trademarks or registered trademarks of their respective holders.

Edition Note

These documents describe the release information of MBD for ANSYS and ***RecurDyn™*** V8R3, working with ANSYS V15.0 and V15.0.7.

Revision History

First printed, March 2015

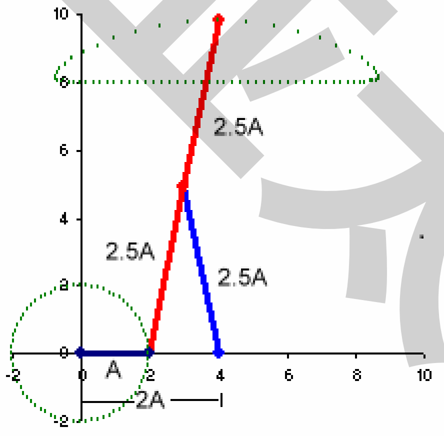
Table of Contents

Chapter 1

Getting Started

Objective

The Hoekens (or Hoeckens) mechanism (http://en.wikipedia.org/wiki/Hoeckens_linkage) is named after Karl Hoecken (1874–1962) and is well-known because it is a four-bar mechanism that converts rotational motion to approximate straight-line motion for a portion of the path of a point on the connector link of the 4-bar (see figure, from http://mechanical-design-handbook.blogspot.com/2011/02/hoekens-straight-line-mechanism.html). As can be seen from the spacing of the dots in the figure, the velocity in during the straight line motion is also quite constant.



The Hoekens mechanism is sometimes used to provide an intermittent straight-line motion (push) to an object in a manufacturing environment.

In this tutorial a Hoekens mechanism will be defined and the loads on the connector body will be evaluated. Interesting (peak) loads at certain output times in the simulation will be selected and MBD for ANSYS will prepare projects for linear statics analysis for those load cases.

The purpose of this tutorial is to provide a simple introduction to the load transfer capability of MBD for ANSYS.

Audience

This tutorial is intended for new users of the MBD for ANSYS module who have basic working knowledge of ANSYS Workbench and ANSYS Mechanical. Most new tasks are explained carefully.

Prerequisites

We assume that you have a basic knowledge of physics. The Geneva Wheel tutorial should be done before doing this tutorial.

Procedures

The tutorial is comprised of the following procedures. The estimated time to complete each procedure is shown in the table.

Procedures	Time (minutes)
Importing the system geometry	8
Define a body	5
Adding joints	15
Defining contacts	10
Analysis / Review	17
Total:	55

Estimated Time to Complete

This tutorial takes approximately 55 minutes to complete.

Chapter 2

Importing the System Geometry

Task Objective

Learn how to import the rigid geometry that is used to create the Geneva wheel system.

Estimated Time to Complete

5 minutes

Starting ANSYS Workbench

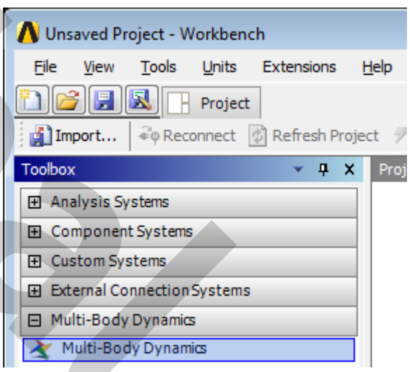
To start ANSYS Workbench and create a new custom system:

1. Run ANSYS Workbench from your Start button (single-click) or from your Desktop (double-click).

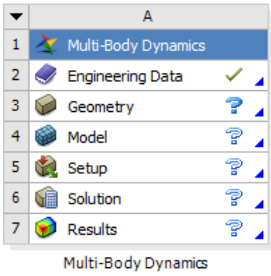
   Workbench 15.0

2. The ANSYS Workbench window appears. If the Getting Started window appears, click OK to dismiss it.

3. Expand the Multi-Body Dynamics group in the Toolbox Window and double-click the Multi-Body Dynamics item (or drag it) to create a new custom Multi-Body Dynamics system in the Project Schematic window.



4. The Multi-Body Dynamics Project Schematic appears.



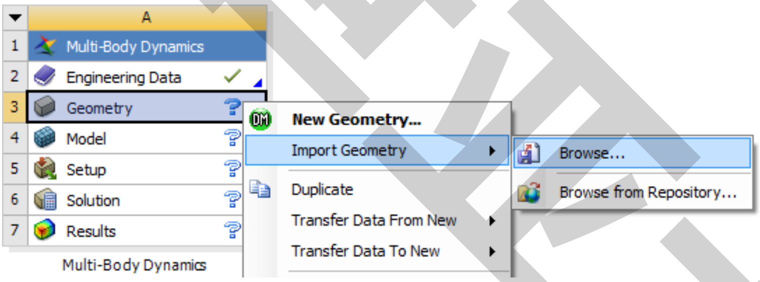
Importing the Hoekens Geometry

You will set up the geometry import in the system and then you will enter the Design Modeler in order to import the geometry and set up your modeling environment. The geometry was previously modeled in CAD.

To set up the geometry import:

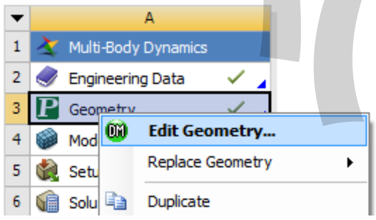
1. With the cursor over the Geometry field in the system, click the right mouse button and select the **Import Geometry** option.

2. Slide the cursor to the right and select the **Browse** option.



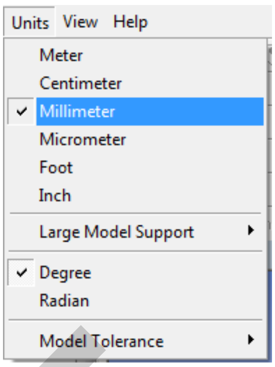
3. Navigate to the folder that contains the tutorial files and select the Parasolid file **Hoekens.x_t**. Then click the **Open** button.

4. Again, with the cursor over the Geometry field in the system, click the right mouse button and select the **Edit Geometry...** option.



5. The DesignModeler application will open in a new window.

6. Click on the **Units** menu. The default length unit is Meter, but select **Millimeter** instead.



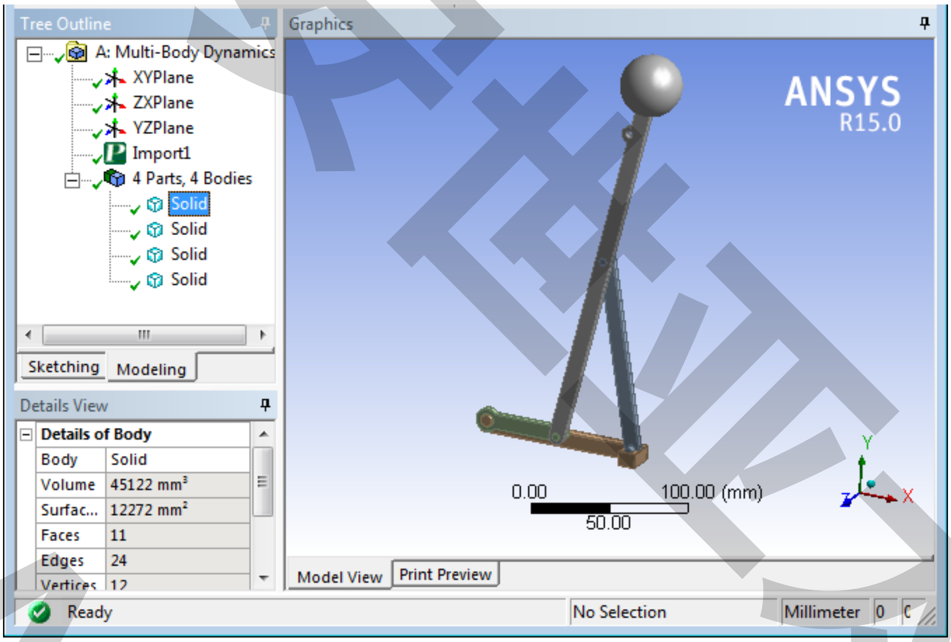
7. You will notice that a message in the lower left corner of the DesignModeler window states: Import Creation – Click Generate to complete the Import Feature. Click the Generate tool in the toolbar to execute the import of the geometry.

8.

8. The geometry will appear in the Graphics window and it will be transparent. The Tree Outline window also shows that there are 7 parts and 7 bodies in the system. Expand the **4 Parts, 4 Bodies** item in the Tree Outline window if it is not already expanded.

To improve the geometry display and save the project:

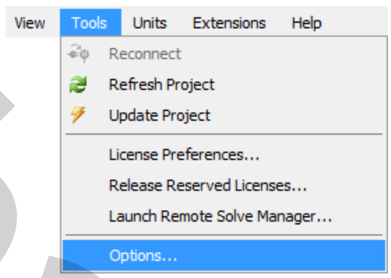
1. Select all of the **Solid** items, go to the **View Menu**, and **uncheck** the **Frozen Body Transparency** option. The geometry should appear to be similar to the image below.



2. At the bottom of the **File** menu select the **Close DesignModeler** option in order to exit the DesignModeler application.

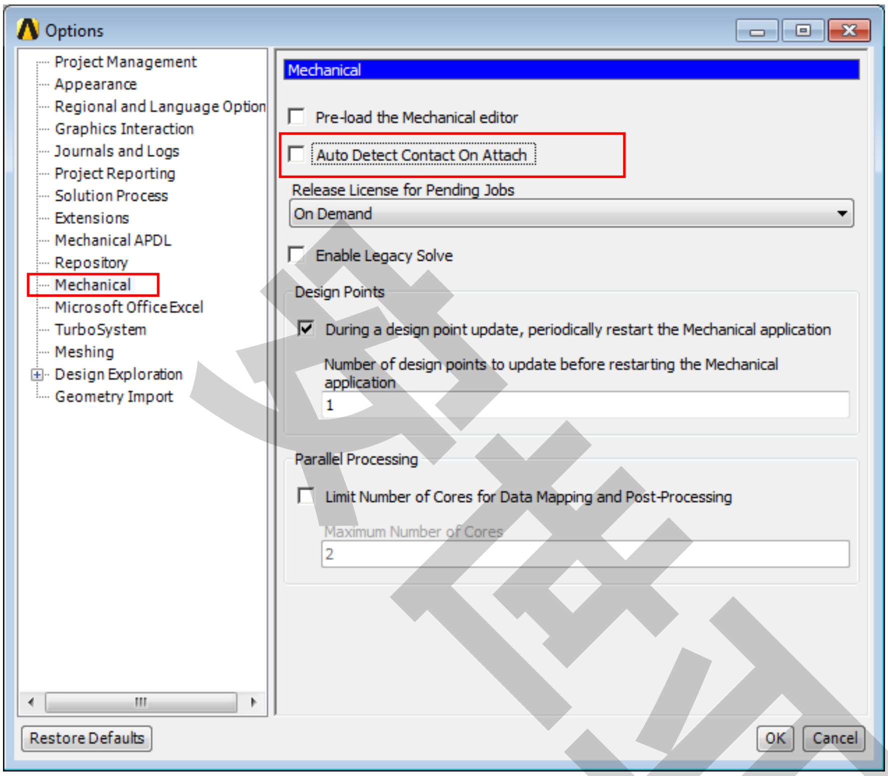
To adjust Workbench settings:

You should uncheck the "Auto Detect Contact on Attach" option for Mechanical if you have not already done so.



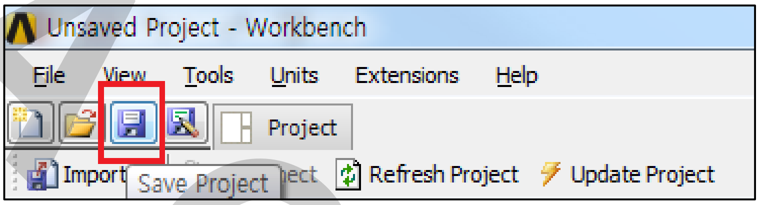
1. Back in the Workbench window, under the **Tools** menu select **Options…**. The options dialog box will appear.
   - **Select Mechanical** in the column on the left.
   - **Uncheck** the **Auto Detect Contact on Attach** option so that contacts are not automatically created when you invoke ANSYS Mechanical.

The dialog box should appear as shown below.



To save the project:

1. Back in the Workbench window select the **Save** icon in the toolbar.



Chapter 3

Adding Joints

You will now create the joints needed to logically combine the geometry to become a mechanism with a driven Crank.

Task Objective

Learn to:

- Set up ANSYS Mechanical for an MBD model
- Rename the geometry so that they have unique names and are easier to identify.
- Use a fixed joint to attach the frame to ground.
- Create revolute joints to connect the Crank and the Rocker to the Frame.
- Connect the Connector to the Rocker with a spherical joint.
- Connect the Crank to the Connector with a cylindrical joint.
- Create an expression which will be used to control the rotational motion of the Crank.

Estimated Time to Complete

15 minutes

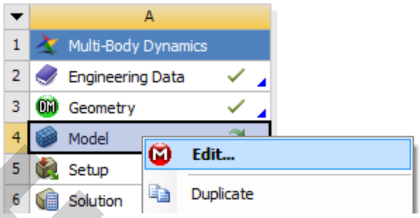
Opening Mechanical and Changing Settings

The remainder of the system definition and running the analysis will be done in the ANSYS Mechanical application. You will open Mechanical through the Workbench project window. Some important settings will be adjusted and then the bodies will be created.

To open ANSYS Mechanical:

1. With the cursor over the Model field in the system, click the right mouse button and select the **Edit...** option.

   The ANSYS Mechanical application will open in a new window.

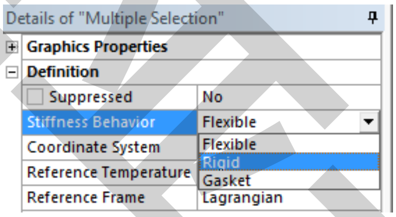


To adjust key settings:

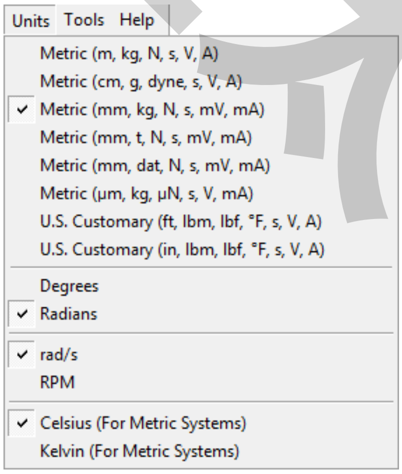
1. Expand the **Geometry** group in the Outline window.

2. Select all 4 of the **Solid** items in the Outline window.

3. In the Details of "Multiple Selection" window pull down the menu of the Stiffness Behavior field and select the **Rigid** option.



4. In the Units menu make sure that the unit system is set to the third option and that the angular units of **Radians** are selected.

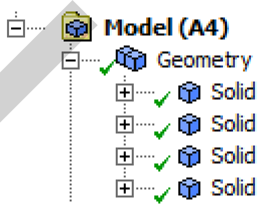


Renaming the geometry

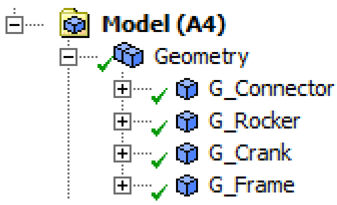
The geometry names must be different or an error will occur when you do an analysis. Also it is easier to work with the model if descriptive names are used.

To rename the geometry:

1. Expand the geometry group in the **Outline** window and the 4 geometric solids will be listed, all of them being named Solid.



2. In the **Outline** window click on the first solid name. The geometry will highlight. Click the right mouse button and select Rename (or type the F2 button). Type in the name of the first solid, which is **G_Connector**.



3. Repeat the above operations until all of the geometry is renamed as shown in the figure.

Setting Up the Joint Icon Size

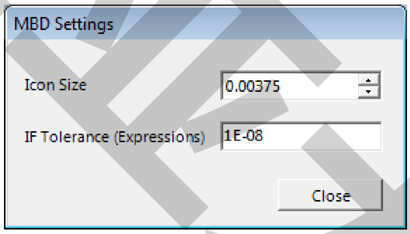
Joint icons will be displayed as the joints are created. You can control the size of the joint icons.

To adjust the scale factor of the joint icons:

1. In the **MBD Tools** toolbar, select the **Settings** icon. A dialog box will appear.

2. Use the up and down arrows to set the **Icon Size** scale factor to 0.00375 as shown.

3. Click on the **Close** button.

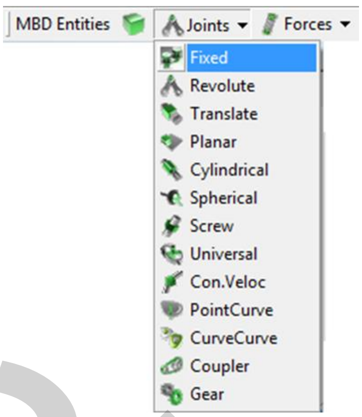


Fix Frame to Ground

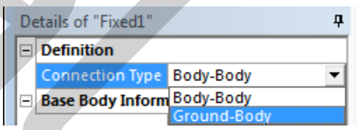
The **Frame** body needs to be attached to the ground. This is done by creating a **Fixed** joint, which constrains all six degrees-of-freedom between two bodies.

To fix the Frame to Ground:

1. In the **MBD Entities** toolbar, pull down the **Joints** menu and select the **Fixed** joint icon.

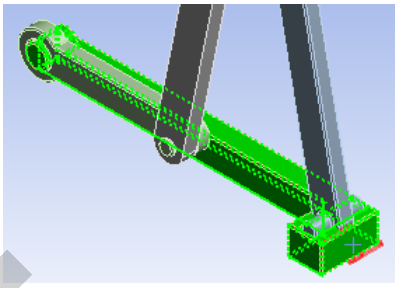


2. In the **Details** window, pull down the menu for the **Connection Type** field and select the **Ground-Body** option. The Details window will update to show that the Base Body is Ground.

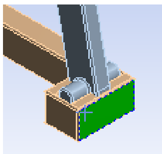


3. Click in the **Scope** field for the Action Body, click on the **Frame** body (highlighted in green in the figure) in the graphics window, and then click on the **Apply** button. The Details window will show that a body was selected for the Action Body.

Action Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	Solid



4. Click in the **Origin** field, make sure the select mode is set to **Face**, and select the right face of the G_Frame solid, and click on the **Apply** button. The Details window will update to show that the origin is set to the center of the right face (you will see more digits).



Action Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	G_Frame
Origin and Orientation	
Origin	Click to Change
Origin (x, y, z)	106,-5,-5

Tip: Understanding the display of the Origin fields in this tutorial
ANSYS displays many digits for each number in the dialog boxes. In the case of the Origin field, the size of each number results in not being able to see the x, y, and z values at one time. Therefore, for ease of viewing, only 5-6 significant digits were retained in the numbers in the Origin field so that you could see all 3 numbers at a glance in the tutorial. Your numbers will be longer in Ansys, and you can scroll through the Origin field in ANSYS in order to check that the numbers match.

Creating Revolute Joints

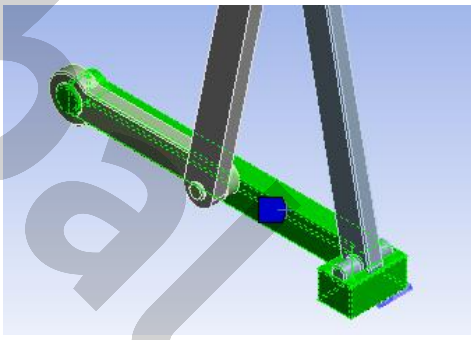
Now you will create two revolute joints, one to attach the Crank to the Frame and second to attach the Rocker to the Frame.

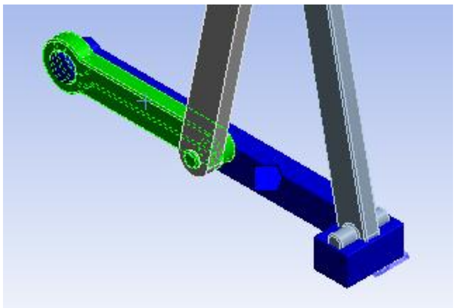
To attach the G_Crank solid to the G_Frame solid:

1. Pull down the **Joints** menu of the **MBD Entities** toolbar and select the **Revolute** joint icon.

2. Click in the **Scope** field for the **Base Body**, and select the **G_Frame** solid in the graphics window. Click on the **Apply** button.

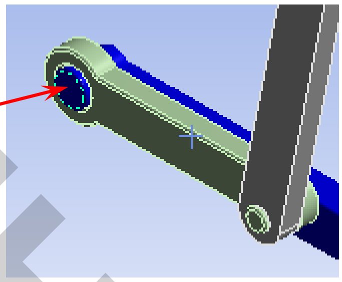
Base Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	G_Frame





3. Click in the **Scope** field for the Action Body, and select the **G_Crank** solid as shown in the figure. Click on the **Apply** button.

Action Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	G_Crank



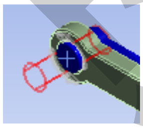
4. Define the joint origin with these steps:
   - Click in the **Origin** field,
   - Make sure the select mode is set to **Face**,
   - **Select** the front face of the rod that holds the G_Crank solid, and
   - Click on the **Apply** button.

   The Origin (x, y, z) field appears as shown (you will see more digits).

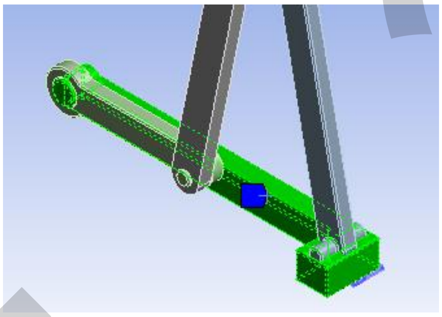
Origin and Orientation	
Origin	Click to Change
Origin (x, y, z)	0,0,-1.5

5. The Rotational Axis is in the +Z direction, so it is easy to directly type 0, 0, 1 into the field. The joint icon appears.

Origin and Orientation	
Origin	Click to Change
Origin (x, y, z)	0,0,-1.5
Rotational Axis	Click to Change
Rotational Axis (x, y, z)	0,0,1



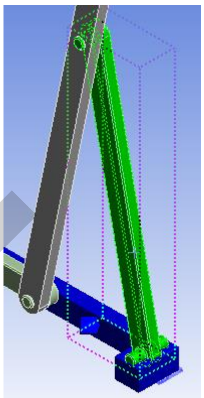
To attach the G_Rocker solid to the G_Frame solid:



1. Select the **Revolute** joint icon in the **MBD Entities** toolbar.

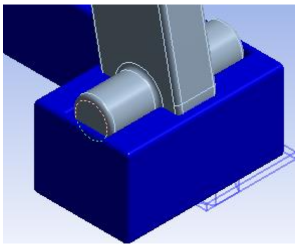
2. Click in the **Scope** field for the **Base Body**, and select the **G_Frame** solid in the graphics window. Click on the **Apply** button.

Base Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	G_Frame



3. Click in the **Scope** field for the Action Body, and select the **Drivewheel** body. Click on the **Apply** button.

Action Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	G_Rocker

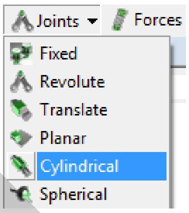


4. Click in the **Origin** field, make sure the select mode is set to **Face**, select the front face of the pin that holds the G_Rocker solid, and click on the **Apply** button. The Origin (x, y, z) field should appear as shown (you will see more digits). Since the rotational axis should be in the positive Z direction you can simply type in 0,0,1 into the Rotational (x, y, z) field.

Origin and Orientation	
Origin	Click to Change
Origin (x, y, z)	100,0,3.5
Rotational Axis	Click to Change
Rotational Axis (x, y, z)	0, 0, 1

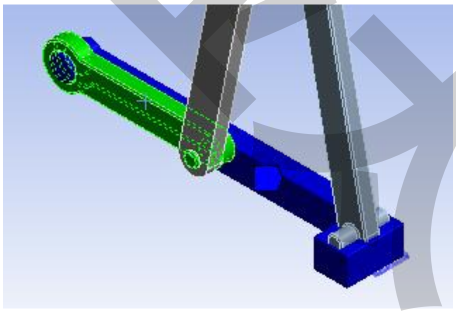
To attach the G_Crank solid to the G_Connector solid:

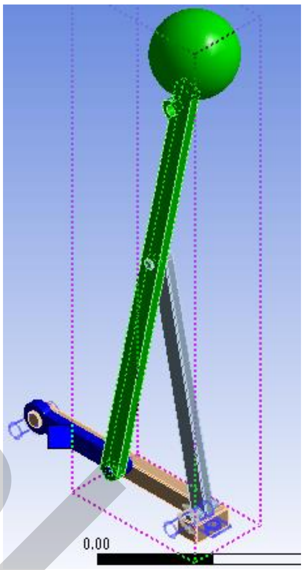
1. Select the **Cylindrical** joint icon in the **MBD Entities** toolbar.



2. Click in the **Scope** field for the **Base Body**, and select the **G_Crank** solid in the graphics window. Click on the **Apply** button.

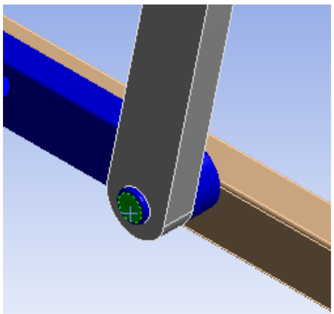
Base Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	Solid





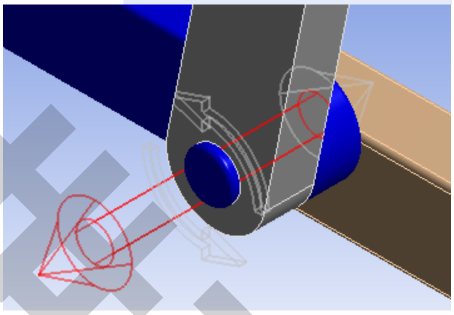
3. Click in the **Scope** field for the **Action Body**, and select the **G_Connector** solid. Click on the **Apply** button.

Action Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	Solid



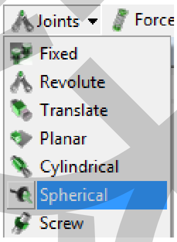
4. Click in the **Origin** field, make sure the select mode is set to **Face**, select the top face of the rod on the **G_Crank** solid that holds The **G_Connnector** solid, and click on the **Apply** button. The Origin (x, y, z) field should appear as shown (you will see more digits). Since the rotational axis should be in the positive Z direction you can simply type in 0,0,1 into the Rotational (x, y, z) field. The cylindrical joint icon then appears.

Origin and Orientation	
Origin	Click to Change
Origin (x, y, z)	50,0,3.5
Trans.Rot. Axis	Click to Change
Trans.Rot. Axis (x, y, z)	0,0,1



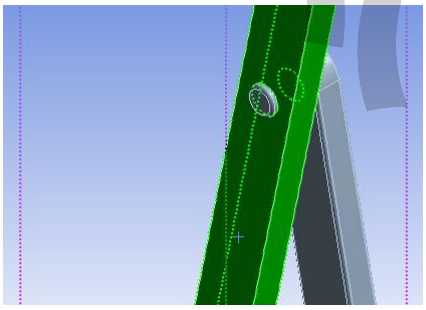
To attach the G_Connector solid to the G_Rocker solid:

1. Select the **Spherical** joint icon in the **MBD Entities** toolbar.



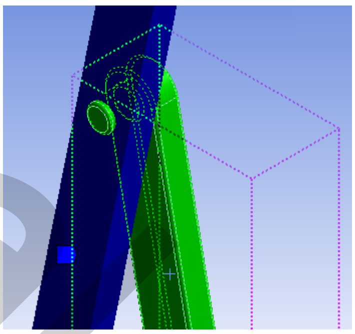
2. Click in the **Scope** field for the **Base Body**, and select the **G_Connector** solid in the graphics window. Click on the **Apply** button.

Base Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	G_Connector

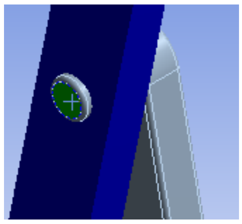


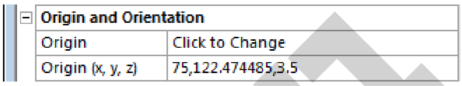
3. Click in the **Scope** field for the Action Body, and select the **G_Rocker** solid. Click on the **Apply** button.

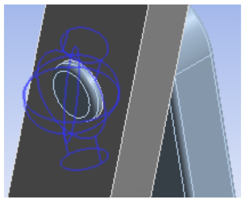
Action Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	G_Rocker



4. Click in the **Origin** field, make sure the select mode is set to **Face**, select the top face of the rod on the **G_Rocker** solid that connects to the **G_Connector** solid, and click on the **Apply** button. The Origin (x, y, z) field should appear as shown (you will see more digits). The spherical joint icon will appear.

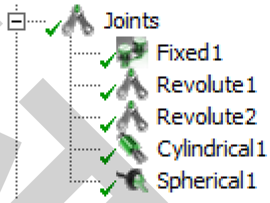






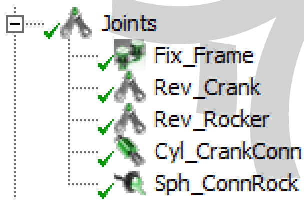
To check and rename the joints:

1. The list of joints in the **Outline** window should appear as shown. If any of the joints do not have a check mark, please review the instructions above and make sure that the base body, action body, origin, and rotational axis direction are defined for that joint.



2. In the **Outline** window click on each joint name. The joint icon will highlight. Click the right mouse button and select Rename. Type in the name of the joint as given in the table below. The list of joints in the Outline window should appear as shown after the renaming is complete

Default Name	Rename to:
Fixed1	Fix_Frame
Revolute1	Rev_Crank
Revolute2	Rev_Rocker
Cylindrical1	Cyl_CrankConn
Spherical1	Sph_ConnRock



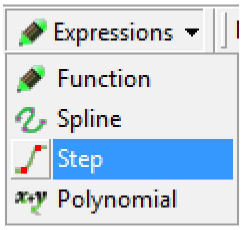
To save the project:

1. Back in the Workbench window click on the Save icon in the toolbar.

Creating the Crank Motion Expression

You will now create a function expression to control the motion of the Crank. In actual steady-state operation the Crank would rotate in a constant velocity, but for this simulation we want to see the dynamics of starting and stopping the motion. We will define a function expression that will smoothly ramp up the rotational velocity of the Crank and then smoothly bring it to a stop, all while rotating the Crank exactly one revolution in one second. Once the expression is defined we will link it to the Motion section of the Rev_Crank joint.

To create the motion expression for the Rev_Crank joint:

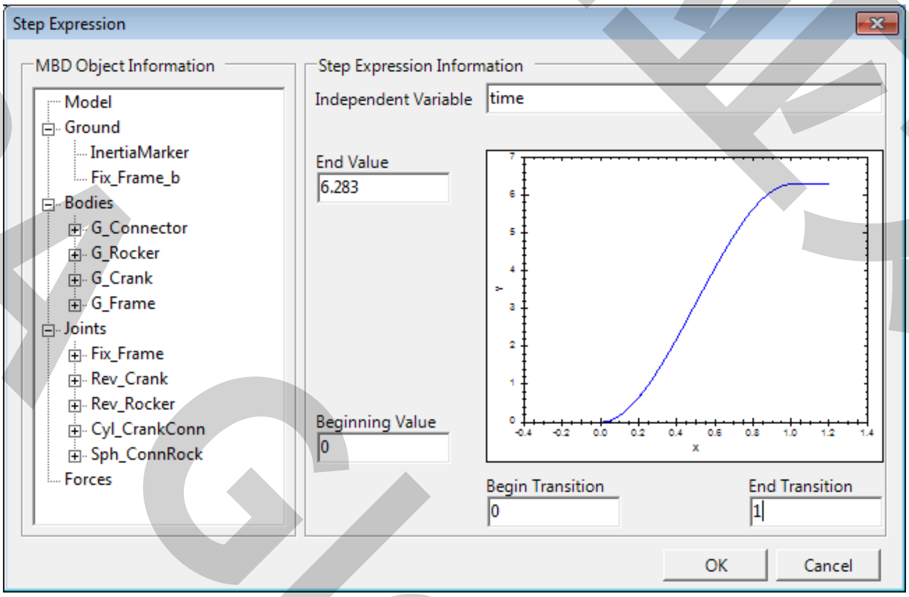


1. In the **MBD Entities** toolbar pull down on the **Expressions** menu and select the **Step** option.

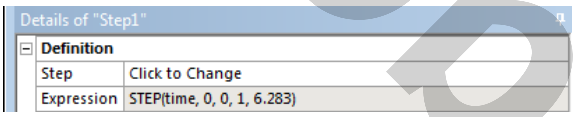
2. In the Details window click in the **Step** field and a dialog box will appear.

3. The Begin and End Transition fields indicate the beginning and ending times of the smooth transition of the Step function. In this model we are controlling the transition of the motion of the Crank body during the simulation. Therefore set the **Begin Transition** value to **0** (zero) and the **End Transition** value to **1**.

4. The Begin and End Value fields indicate the motion of the Crank in radians. Keep in mind that the rotational units of expressions are always radians. Set the **Begin Value** to **0** (zero). The Crank needs to rotate one revolution counter-clockwise, or 2π radians. A value of 6.283 can be used to approximate 2π.. Set the **End Value** to **6.283**. Note that the default value of the Independent Variable, time, is what is needed for a motion expression, and no adjustment is needed to that field.



5. Click **OK** to exit the Step Expression dialog box. The Expression field should be filled as shown.

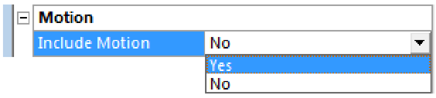


Also note the addition of the Step1 expression in the Expressions group in the Outline window.

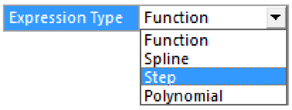
To link the expression with Rev_Crank:

1. In the **Outline** window select the **Rev_Crank** joint.

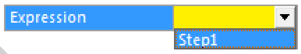
2. Near the bottom of the **Details** window, in the Motion group, click in the **Include Motion** field, pull down on the menu and select the **Yes** option.



3. Additional fields will appear. Click in the **Expression Type** field, pull down on the menu and select the **Step** option.



4. The default value of the Motion Type, Displacement, is good. Click in the **Expression** field and pull down the menu to select **Step1**, the step expression just created.



To save the project:

1. Back in the Workbench window click on the Save icon in the toolbar.

Chapter

4

Adding a Spring, Gravity

You will now create a spring between ground and the Connector body. Then you will define the gravity acceleration in the model.

Task Objective

You will learn to:

- Define a spring and its end points
- Set the stiffness and damping coefficients for the spring.

Estimated Time to Complete

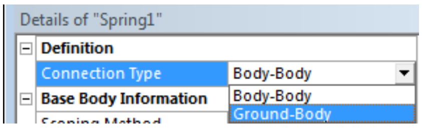
5 minutes

Adding a Spring

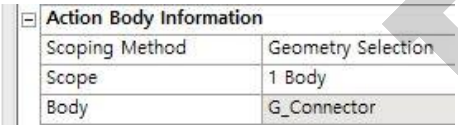
To add a spring between the ground and the connector body:

1. Pull down the **Forces** menu of the **MBD Entities** toolbar and select the **Spring** force icon.

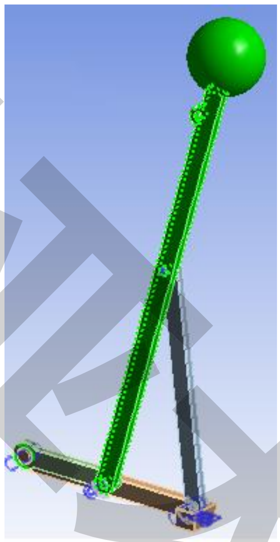
2. In the **Details** window, pull down the menu for the **Connection Type** field and select the **Ground-Body** option. The Details window will update to show that the Base Body is Ground.



3. Click in the **Scope** field for the Action Body, and select the **G_Connector** solid as shown in the figure. Click on the **Apply** button.



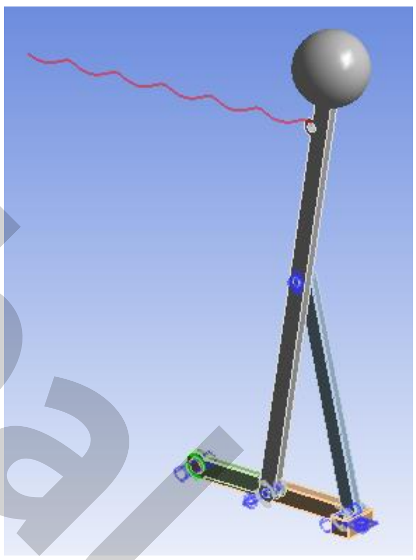
Action Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	G_Connector



4. Click in the **Origin1 (x, y, z)** field (the attachment point of the spring on ground), and type in the coordinates of **-100, 200, 0**.

5. Click in the **Origin2 (x, y, z)** field (the attachment point of the spring on the Connector body), and type in the coordinates of **87.8, 210.25, 0**.

The spring will appear as shown in the figure.



To adjust the spring properties:

1. Make the following changes to the **Properties** section of the Details window for the spring:
   - Stiffness Coefficient = 4.0
   - Damping Coefficient = 0.04.
   - Free Length = 25

   The Details window should appear as shown in the figure.

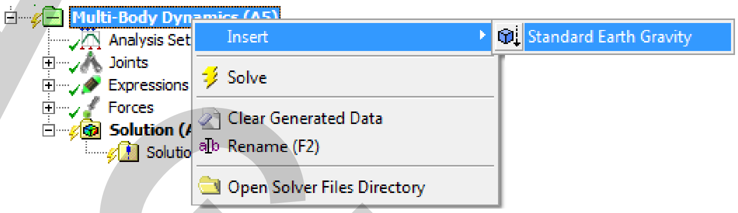
Properties	
Defined By	Spring Parameters
Stiffness Coefficient	4 [N mm^-1]
Damping Coefficient	0.04 [N sec mm^-1]
Stiffness Exponent	No
Damping Exponent	No
Free Length	25 [mm]
Pre Load	0 [N]
Current Length of Spring	188.079511111657 [mm]
Current Force in Spring	652.318044446628 [N]

Adding Gravity

To add gravity acting in the –Y direction:

1. Place the cursor over the **Multi-Body Dynamics** item in the Outline window and **Click** on the right mouse button.

2. Select **Insert** and then select **Standard Earth Gravity** as shown in the figure.



A new Acceleration item appears in the Outline window.

1. In the **Details** window go to the **Direction** field and then select the **–Y Direction** option as shown in the figure.

Definition	
Coordinate System	Global Coordinate System
X Component	0. mm/s² (ramped)
Y Component	0. mm/s² (ramped)
Z Component	-9806.6 mm/s² (ramped)
Suppressed	No
Direction	-Z Direction

+X Direction
-X Direction
+Y Direction
-Y Direction
+Z Direction
-Z Direction

The gravity components should now appear as shown below:

X Component	0. mm/s² (ramped)
Y Component	-9806.6 mm/s² (ramped)
Z Component	0. mm/s² (ramped)

To save the project:

1. Back in the Workbench window click on the Save icon in the toolbar.

Chapter

5

Analysis / Review

We can now run a simulation (analysis) with this model and then review an animation of its motion and plot some of the outputs.

Task Objective

You will learn to:

- Set up and run a simulation.
- View the motion of the Toekens mechanism

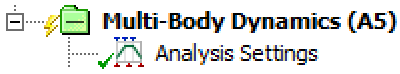
Estimated Time to Complete

10 minutes

Setting up and Running an Analysis

To set the parameters controlling the analysis:

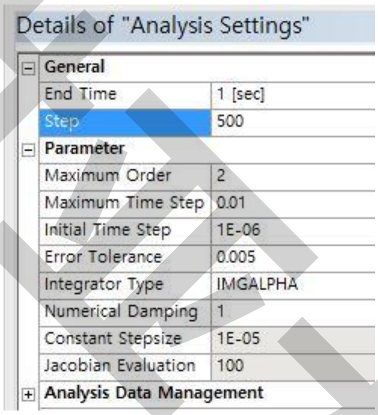
1. In the Outline window, select the **Analysis Settings** item in the Multi-Body Dynamics group.



2. In the Details window set the **End Time** value to **1**.

3. Set the number of steps to **500** using the **Step** field. This sets the number of animation output steps that are saved in the results files. When you play an animation you will see a maximum of this number of steps.

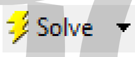
The Details window should appear as shown.


Details of "Analysis Settings"

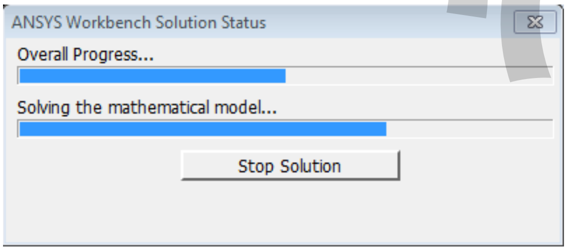
General	
End Time	1 [sec]
Step	500
Parameter	
Maximum Order	2
Maximum Time Step	0.01
Initial Time Step	1E-06
Error Tolerance	0.005
Integrator Type	IMGALPHA
Numerical Damping	1
Constant Stepsize	1E-05
Jacobian Evaluation	100
Analysis Data Management	

To run an analysis:

1. Select the Solve Icon in the ANSYS Mechanical toolbar.



After a pause (5-10 seconds) the ANSYS Workbench Solution Status box will appear and provide ongoing feedback on the progress of the solution.

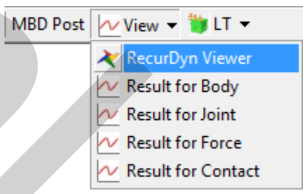


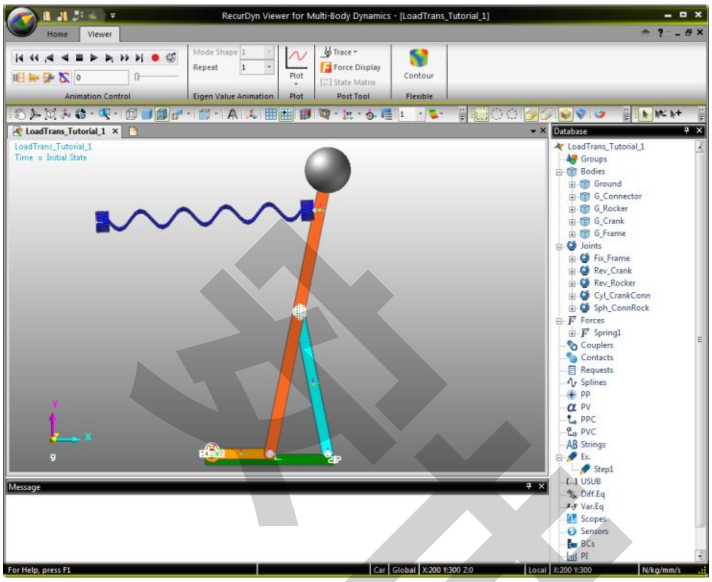
Viewing the Animation Results

An external viewer is provided in order to show animations and MBD output plots with high performance and capability.

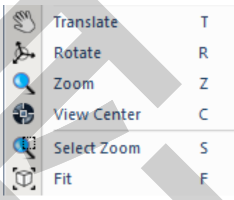
To invoke the results viewer:

1. In the **MBD Post** toolbar, pull down the **View** menu and select the **RecurDyn Viewer** option. At this point all of the geometry and results are being prepared and written to several files for use by the viewer. A startup graphic will appear and then 5-10 seconds later the RD Viewer will appear with the Hoekens mechanism model loaded as shown in the figure.





To adjust the view in the viewer:

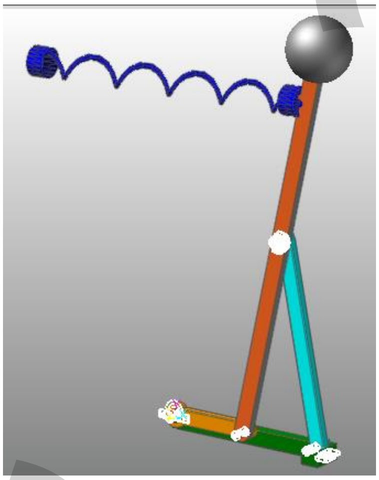


The RD Viewer has its own methods for controlling the view on the screen. In the RD Viewer graphics window, click on the right mouse button and the pop-up menu that appears contains information about view control, as shown in the figure. You can see the typical Translate, Rotate and View commands. Each view command can be accessed in three ways:

1. Select the icon associate with each view command from the toolbar on the left of the screen, just below the ribbon.

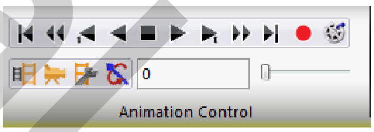
2. Use the keyboard equivalent for each command, as shown in the figure.

3. Click the right mouse button to pop up the menu and then select the view command.



Note that each view command is not persistent. Once the view command is given the user can click and hold down on the left mouse button and perform the view command. Once the left mouse button is released the cursor mode reverts back to select. Use the RD Viewer view commands to position the model at a good angle to see the 3D animation, as shown in the figure.

To play the animation:



1. At the left side of the **Ribbon**, there is an Animation Control group. The buttons can be used to play the animation following standard conventions. Click the **Play** button in order to see the motion of the Toekens mechanism. You will see that the extreme left and right positions of the sphere geometry occur at approximately 0.24 and 0.75 seconds.

Chapter

6

Load Transfer

The purpose of transferring loads for use in ANSYS Static Structural systems is to assess component reliability. A sudden failure could occur if a peak stress exceeds the yield stress of the component material. With cyclic loading the damage to the component is a function of the variation of stress throughout the component during the work cycle of the assembly.

In this section we are going to do the load transfer for the Connector body. Since the Hoekens mechanism causes cyclical loading of the Connector body we want to identify the load cases that capture the extremes of the cyclical loading. Then we can use ANSYS to do a set of linear statics stress analyses, using the Static Structural systems in Workbench.

Task Objective

You will learn to:

- Set up the transfer of the load cases of interest.
- Associate the loads with the loaded geometry of the body
- Run the linear statics analysis and review the results.
- Evaluate the results and determine the need for changes to the Connector design.

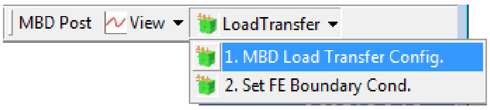
Estimated Time to Complete

25 minutes

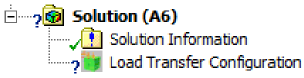
Setting up the Load Transfer

To select the Connector body for load transfer:

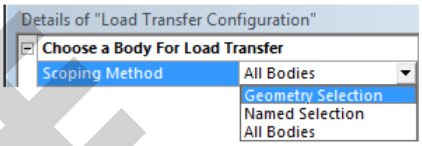
1. In the **MBD Post** toolbar, pull down the Load Transfer menu and select the **1. MBD Load Transfer Config.** item.



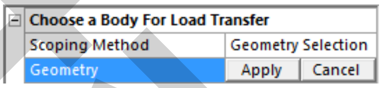
2. A **Load Transfer Configuration item** will appear in the **Outline** Window, within the **Solution** group.



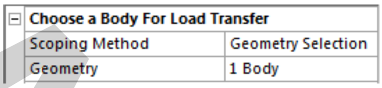
3. In the Details window pull down on the menu in the **Scoping Method** field and select **Geometry Selection**.



4. Define the Connector as the body to receive the loads by doing the following:
   - Click in the new **Geometry** field that appears.
   - Select the **Connector** body and click the **Apply** button.



The **Geometry** field shows that one body has been selected.

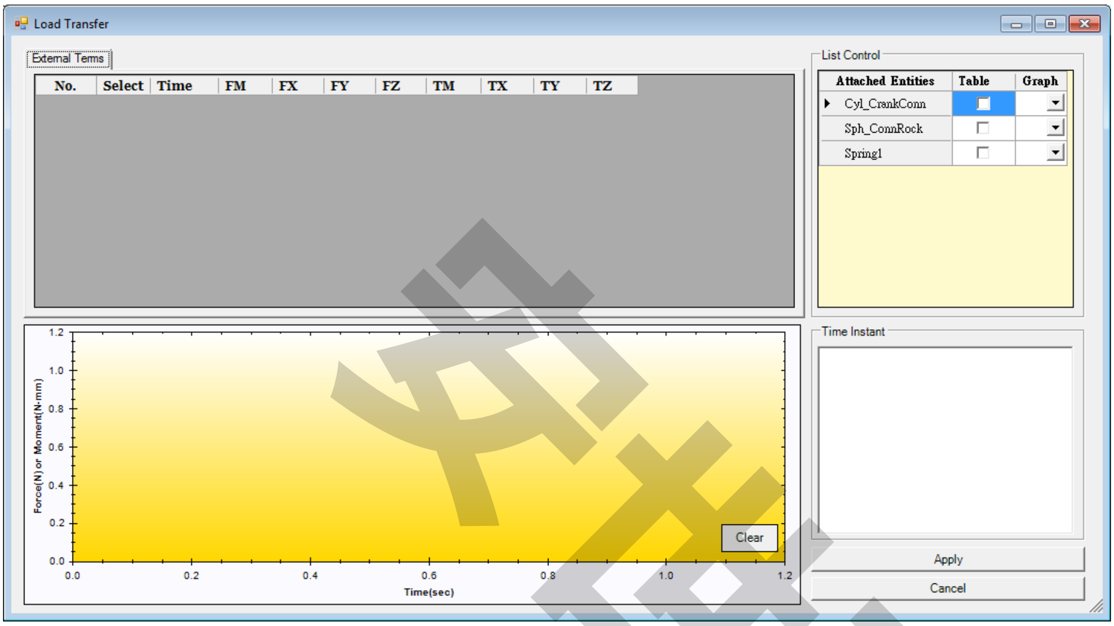


To select the time instants for the load transfer:

1. Click in the **Show Information Dialog** field and click on the **Apply** Button that appears.

The window below appears. You can see that the **List Control** section in the upper right contains a list of the model entities that apply loads on the Connector body:

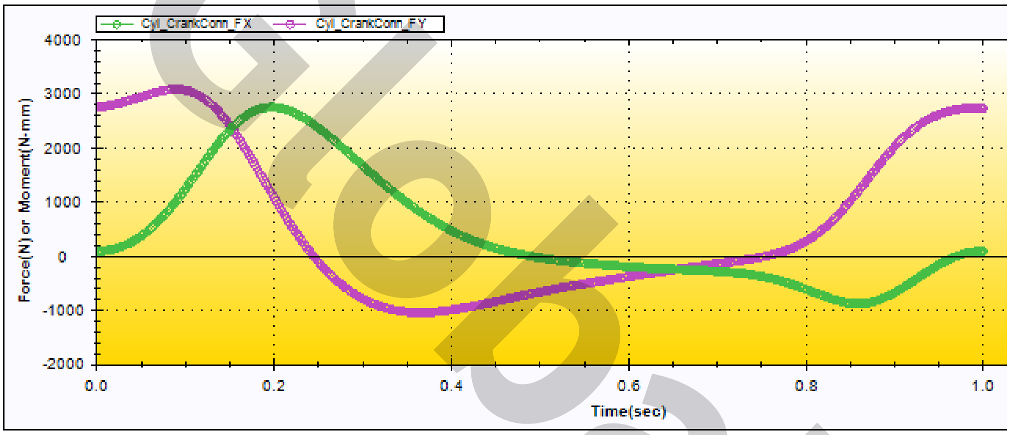
- **Cyl_CrankConn**, the cylindrical joint that attaches the Connector to the Crank.
- **Sph_ConnRock**, the spherical joint that attaches the Connector to the Rocker.
- **Spring1**, the spring force between the Connector and Ground.



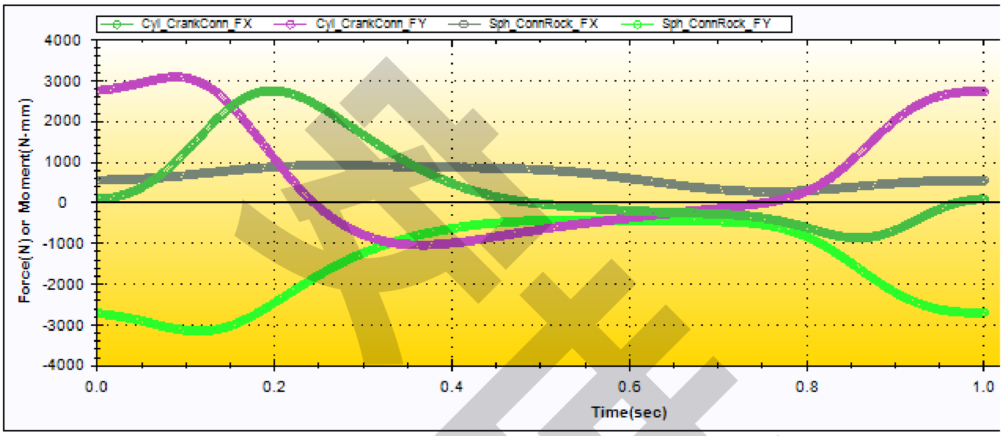
You can use the **Table** and **Graph** columns of the List Control section to display the forces for the 1200 output points that were saved in the earlier simulation. You will review the loads of interest for the Connector body and select a number of instants of time (time instants) with maximum forces that can influence the stresses in the Connector.

There are10 potential force components that can be considered: 1) four from the cylindrical joint, 2) three from the spherical joint, and 3) three from the spring. However, given the planar motion of the mechanism and the spherical joint on the Connector (which releases any potential torques acting on the body), it will be sufficient to examine the x and y translational forces of the joints and the spring, for a total of six components.

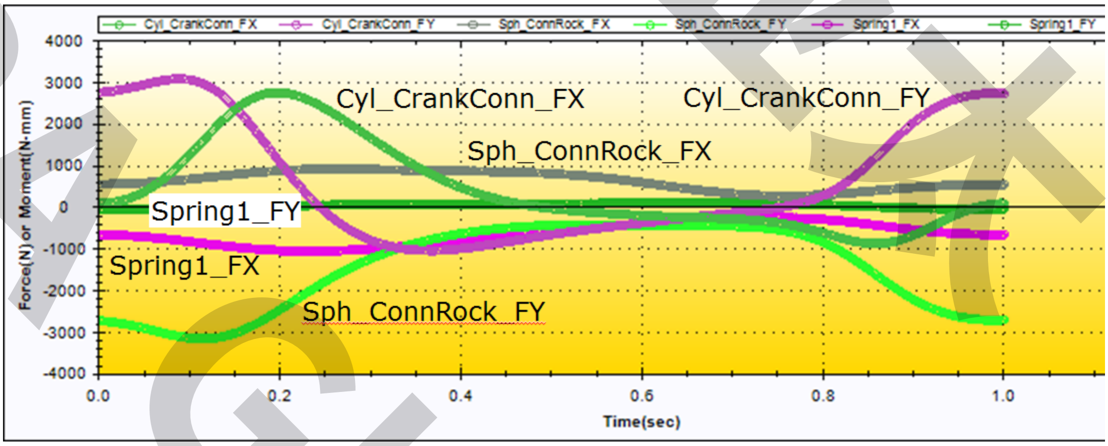
2. In the **List Control** section, go to the **Cyl_CrankConn** row. Pull down the Graph menu, select FX, and click on the joint name. Pull down the Graph menu, select FY, and click on the joint name. The Load Transfer dialog box should contain a plot that appear as follows (curves colors may be different):



1. In the **List Control** section, go to the **Sph_ConnRock** row. Pull down the Graph menu, select FX, and click on the joint name. Pull down the Graph menu, select FY, and click on the joint name. The Load Transfer dialog box should now contain a plot that appear as follows:



2. In the **List Control** section, go to the **Spring1** row. Pull down the Graph menu, select FX, and click on the joint name. Pull down the Graph menu, select FY, and click on the joint name. The Load Transfer dialog box should now contain a plot that appear as follows (labels added for clarity:



3. Now the overall set of curves should be reviewed in order to identify minimum and maximum time instants (output times) of interest. For purposes of this tutorial it has been decided in advance that we are going to select the most interesting four time instants. More time instants should be used with models that have a larger number of distinct load configurations. The approximate selected time instants are as follows:

   - 0.1 seconds – At this time Cyl_CrankConn_FY is at a maximum and Sph_ConnRock_FY is at a minimum.
   - 0.2 seconds - At this time Cyl_CrankConn_FX is at a maximum and both spring forces are at a minimum.
   - 0.4 seconds - At this time Cyl_CrankConn_FY is at a minimum.
   - 0.85 seconds - At this time Spring1_FX is near to its minimum magnitude and Cyl_CrankConn_FX is at a minimum.

Now the table function under the Loads Acting on Body tab in the upper left of the Load Transfer dialog box will be used to pick the time instants.

4. First the time instant at approximately 0.1 seconds occurs when Cyl_CrankConn_FY is at a maximum. Do the following to select the time instant:

- Click the checkbox in the Table column of the Cyl_CrankConn row in the List Control area of the dialog box (upper right corner).
- Scroll the table in the vicinity of 0.1 seconds until you find the time of the maximum value of Cyl_CrankCon_FY, which is marked by red shading.
- Click in the **Select** box for that row, as shown in the figure.

Loads Acting on Body

No.	Select	Time	FM	FX	FY	FZ
44	☐	0.086	3230.361606	958.575282	3084.861348	0
▶ 45	☑	0.088	3243.631917	999.373657	3085.838671	0
46	☐	0.09	3256.600077	1040.920705	3085.762167	0

5. The time instant at approximately 0.2 seconds occurs when Cyl_CrankConn_FX at a maximum. Do the following to select the time instant:

- Scroll the table in the vicinity of 0.2 seconds until you find the time of the maximum value of Cyl_CrankCon_FX, which is marked by red shading.
- Click in the **Select** box for that row, as shown in the figure.

Loads Acting on Body

No.	Select	Time	FM	FX	FY	FZ
98	☐	0.194	3026.427732	2757.202618	1247.837545	0
▶ 99	☑	0.196	3005.053874	2758.559871	1191.929624	0
100	☐	0.198	2983.366234	2758.508084	1136.269001	0

6. The time instant at approximately 0.4 seconds occurs when Cyl_CrankConn_FY at a minimum. Do the following to select the time instant:

- Scroll the table in the vicinity of 0.4 seconds until you find the time of the minimum value of Cyl_CrankCon_FY, which is marked by blue shading.
- Click in the **Select** box for that row, as shown in the figure.

Loads Acting on Body

No.	Select	Time	FM	FX	FY	FZ
183	☐	0.364	1326.409948	829.903353	-1034.709512	0
▶ 184	☑	0.366	1312.613271	807.615133	-1034.751853	0
185	☐	0.368	1298.95175	785.630951	-1034.43688	0

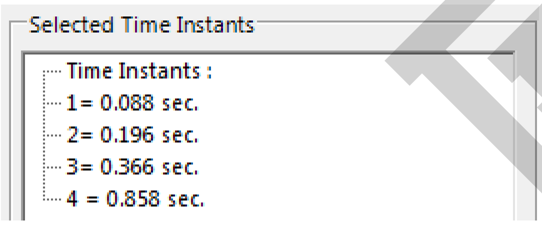
7. The time instant at approximately 0.85 seconds occurs when Cyl_CrankConn_FX at a minimum. Do the following to select the time instant:

- Scroll the table in the vicinity of 0.85 seconds until you find the time of the maximum value of Cyl_CrankConn_FX, which is marked by blue shading.
- Click in the **Select** box for that row, as shown in the figure.

Loads Acting on Body

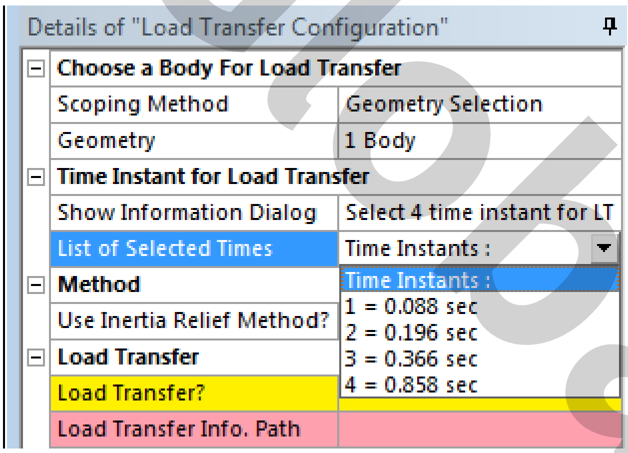
No.	Select	Time	FM	FX	FY	FZ
429	☐	0.856	1438.433711	-867.770742	1147.198971	0
430	☑	0.858	1471.730726	-868.517164	1188.136889	0
431	☐	0.86	1505.124402	-868.327623	1229.392779	0

- You may have noticed that as you have click on the Select box that the time instants were being added to the list in the **Time Instant** section of the dialog box. There are now four time instants in the list, as shown in the figure.

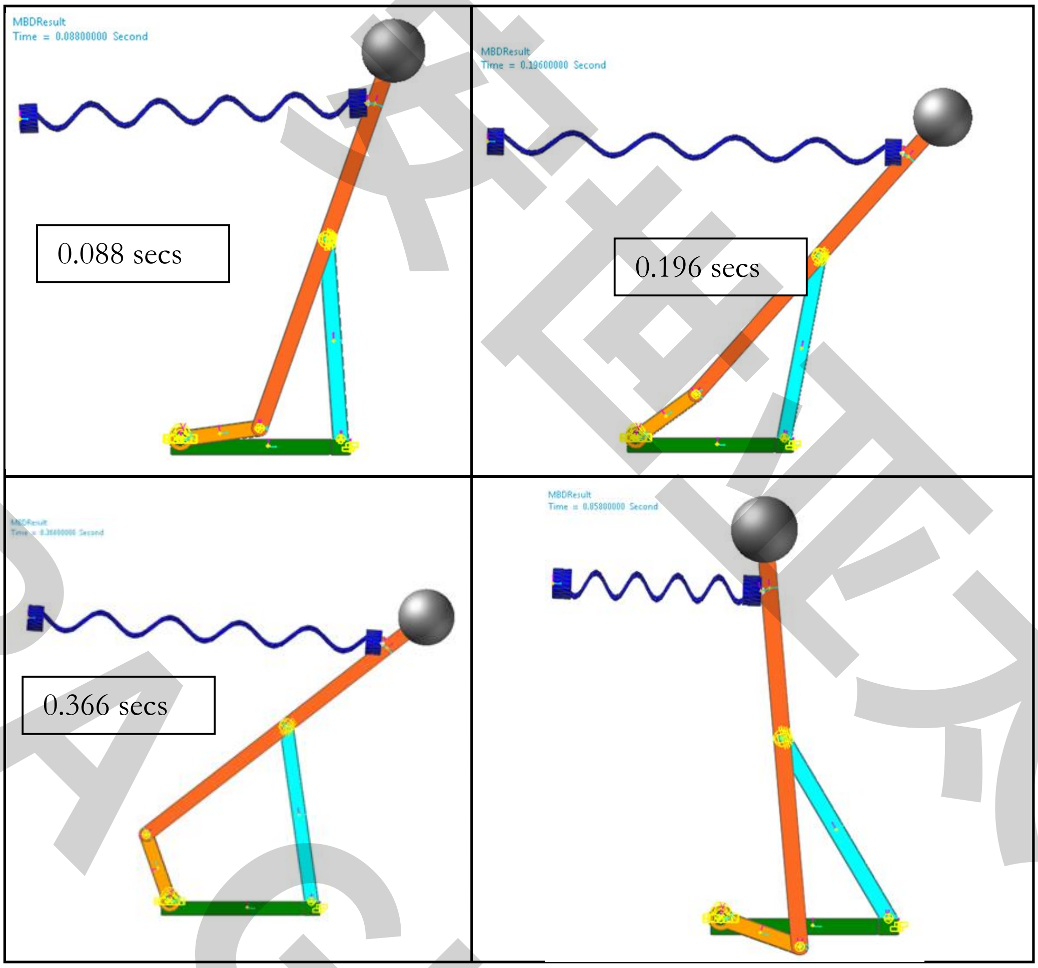


Tip: Deciding which output time instant to select.
In this tutorial we have been looking at individual force components that act on the body of interest and taking note of times of maximum and minimum magnitude. There is no guarantee that this approach will result in the selection of output times associated with maximum stresses in the body. Each organization should investigate the best process for selecting output time instances of interest. Please let us know if you develop an approach to time instant selection that you think should be incorporated into MBD for ANSYS.

8. Now that all of the time instances have been identified you can click on the **Apply** button. All of the time instant values are shown in the Details window for the Load Transfer Configuration.



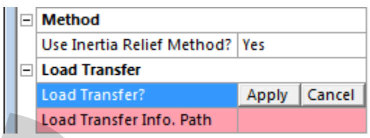
The configurations of the mechanism at the selected times instants are shown in the figure. A review of the MBD output data suggest why maximum loads occurred at these times. Configurations at 0.088 and 0.196 seconds occurred at a similar Crank position where the Connector was undergoing a large vertical acceleration. The configuration at 0.366 seconds occurred at a low horizontal acceleration. The configurations at 0.858 seconds occurred at a time of minimum spring length.



It can be useful to review the body motions as well as the joint and external forces in order to understand the corresponding stresses for each load case.

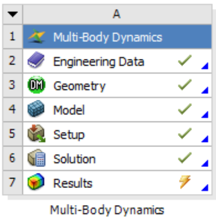
To execute the load transfer:

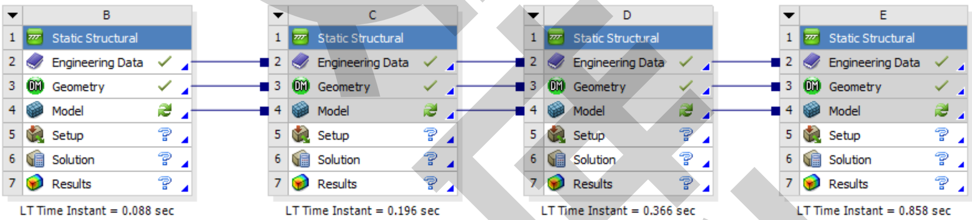
1. The next item in the Details window of the Load Transfer is the Method. The default option is use the inertia relief method. The default value of Yes is good and should not be changed.



2. Click in the **Load Transfer?** field and click the **Apply** button. Wait 5-20 seconds for the load transfer to complete.

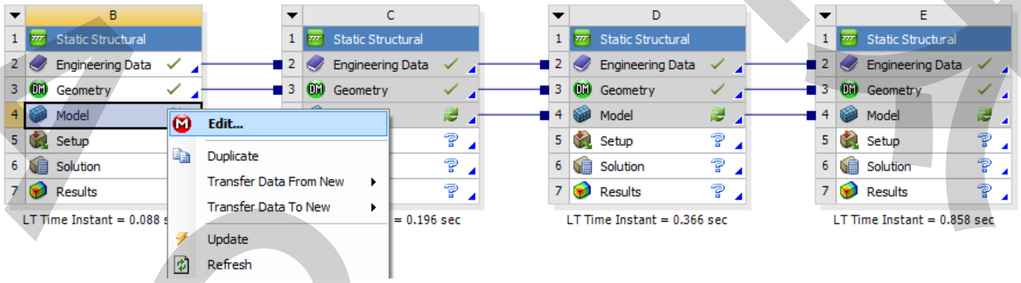
3. Close the Mechanical window. In the Workbench window you can see that a new linear statics project has been created for each of the four time instants that were designated.



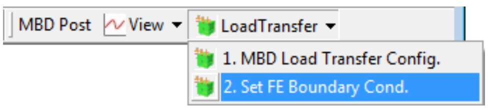


4. In the Workbench window, select the **Save** icon in the toolbar.

5. Click the right mouse button on the **Model** line of the first Static Structural block (for time = 0.088 seconds) and select the **Edit** option in order to open the model in Mechanical.



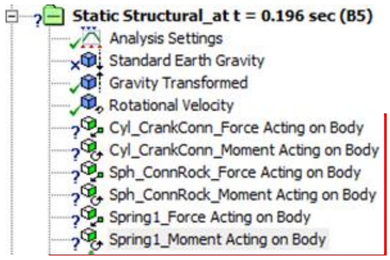
6. In the **MBD Post** toolbar, pull down on the **LoadTransfer** menu and select the **2. Set FE Boundary Cond.** option.



This command will run for 5-20 seconds. The components of the MBD model other than the Connector geometry will be suppressed. You may need to select the Connector geometry item in order for the graphics window to refresh and show that body.

Associating the Loads with Geometry

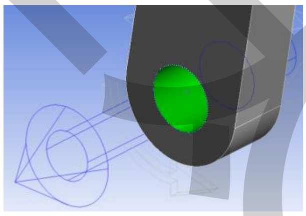
To associate the transferred loads with the appropriate geometry:



Six new entities related to the transferred loads have been added under the first Static Structural group in the Outline window as shown. The six items correspond to the forces and moments of the cylindrical joint, spherical joint and spring.

1. Since a spherical joint and a spring do not have reaction moments we will delete their moment entries by:
   - Placing the cursor over the **Sph_ConnRock_Moment Acting on Body** entity name,
   - Clicking on the right mouse button, and
   - Selecting the delete option, and
   - Repeating the above steps for **Spring1_Moment Acting on Body**.

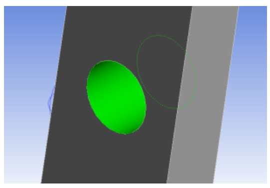
2. Each of the remaining four loads needs to be associated with geometry on the body. Both the Force and the Moment entities of the cylindrical joint will be associated with the hole at the bottom of the shaft. To do this:

   - Select the **Cyl_CrankConn_Force Acting on Body** entity
   - In the Details window click in the Geometry field
   - Select the surface of the hole at the shaft bottom.
   - Click the **Apply** button.

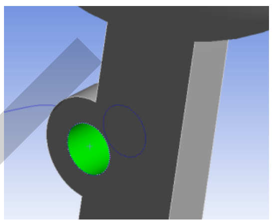
   Repeat the same process with the **Cyl_CrankConn_Moment Acting on Body** entity.

3. The forces of the spherical joint need to be associated with the hole at the midpoint of the shaft. To do this:

   - Select the **Sph_ConnRock_Force Acting on Body** entity
   - In the Details window click in the Geometry field
   - Select the surface of the hole at the shaft midpoint.
   - Click the **Apply** button.

4. The forces of the spring need to be associated with the hole near the top of the shaft. To do this:

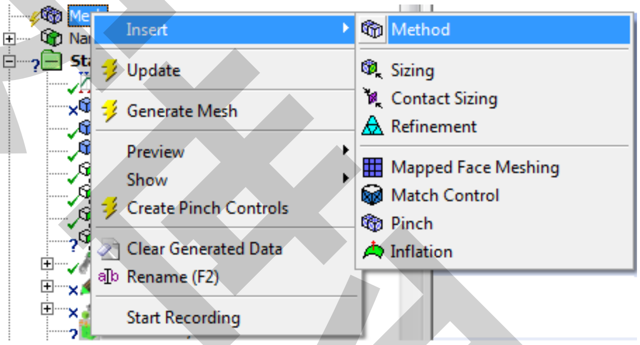
   - Select the **Spring1_Force Acting on Body** entity
   - In the Details window click in the Geometry field
   - Select the surface of the hole near the top of the shaft.
   - Click the **Apply** button.

Setting Up and Defining the Mesh

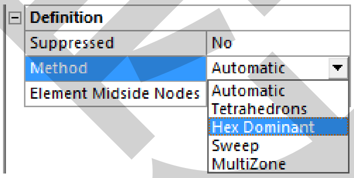
To define the mesh:

When meshing thick-walled solid geometry it is good practice to use hexahedral elements as much as possible. Another good practice is to define more elements near areas of stress concentration, which in this model are in the vicinity of the applied loads. The mesh that will be defined will follow these good practices.

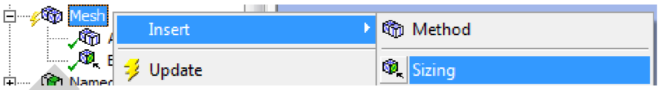
1. Set up the mesh to use hexahedral elements as much as possible by:
   - Placing the cursor over the Mesh name in the Outline window and clicking on the right mouse button
   - Selecting the **Insert** option.
   - Selecting **Method**.
   - In the Detail window click in the **Geometry** field and select the G_Connector geometry.



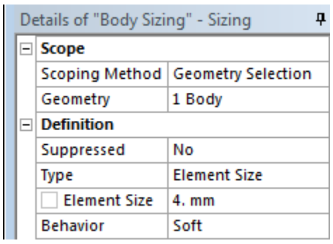
   - Pull down the menu in the Method field and select the **Hex Dominant** option.



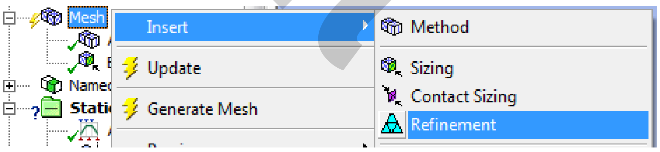
2. Control the element sizing in the mesh by:
   - Placing the cursor over the Mesh name in the Outline window and clicking on the right mouse button.
   - Selecting the **Insert** option.
   - Selecting **Sizing**.



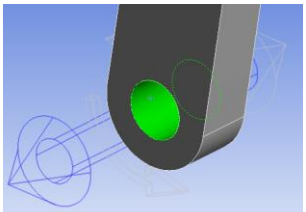
   - In the Detail window click in the **Geometry** field and select the G_Connector geometry.
   - Click in the **Element Size** field and set the value to 4 (4 mm).



3. Control the refinement in the mesh at the cylindrical joint hole by:
   - Placing the cursor over the Mesh name in the Outline window and clicking on the right mouse button.
   - Selecting the **Insert** option.
   - Selecting **Refinement**.



- In the Detail window click in the **Geometry** field and select the face of the hole at the bottom of the shaft.

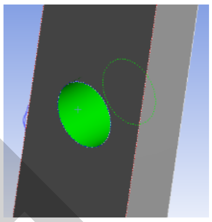


- Click in the **Refinement** field and set the value to 1.

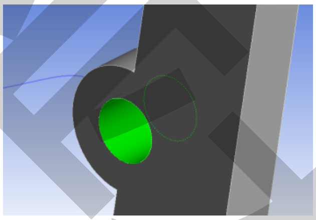
Details of "Refinement" - Refinement

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Refinement	1

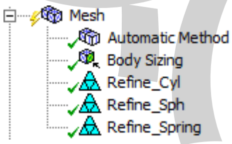
4. Control the refinement in the mesh at the spherical joint hole by following the same steps as given above.
   - In the Detail window click in the **Geometry** field and select the face of the hole at the midpoint of the shaft.



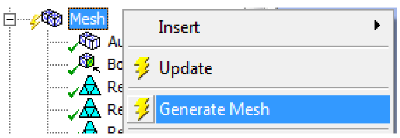
5. Control the refinement in the mesh at the spring attachment point by following the same steps as given above.
   - In the Detail window click in the **Geometry** field and select the face of the hole near the top of the shaft.



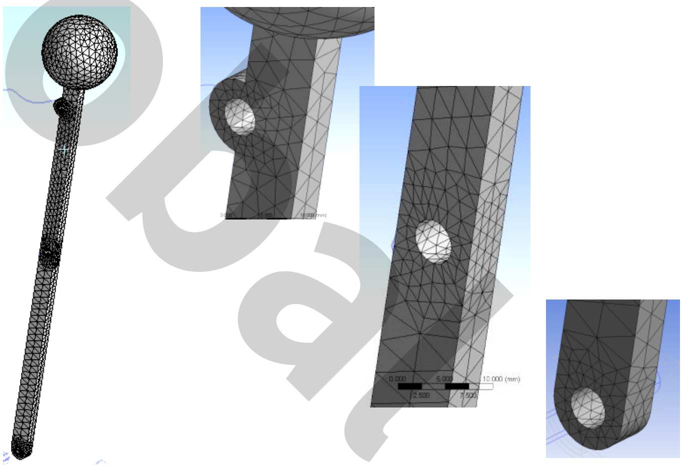
6. Rename the three refinement entities as **Refine_Cyl**, **Refine_Sph**, and **Refine_Spring**. The Outline window should appear as shown.



7. Create the mesh as follows:
   - Place the cursor over the Mesh and click on the right mouse button.
   - Selecting the **Generate Mesh** option.



The mesh will appear on the geometry as shown when the Mesh item is selected. The mesh refinement near the holes can be seen.

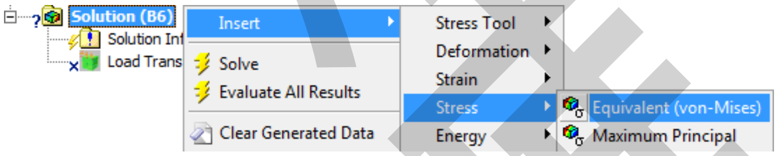


Running the Analysis

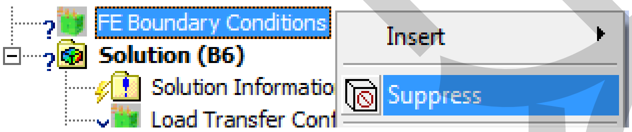
To run the structural analysis and retrieve stresses:

It is often a good practice to examine von-Mises stresses when taking the first look at the stresses in a component under 3D loading. You will request the display of von-Mises stress and perform a solve of the component.

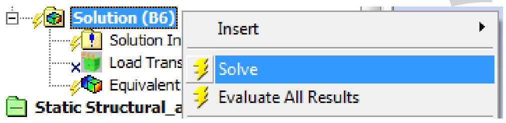
1. Request **Equivalent (von-Mises)** stresses by:
   - Placing the cursor over the **Solution** item in the Outline window and clicking on the right mouse button,
   - Selecting **Insert**, Selecting **Stress**, and Selecting **Equivalent (von-Mises)**



2. Since we have externally defined loads and we are using the Inertial Relief Method there are no FE Boundary Conditions to use. We need to suppress the FE Boundary Conditions entity by:
   - Placing the cursor over the **FE Boundary Conditions** item in the **Static Structural_at t=0.088 sec (B5)** group, clicking on the right mouse button, and
   - Selecting **Suppress**.



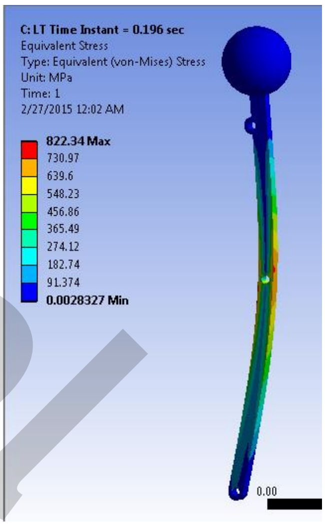
3. The analysis is now ready to run. Invoke the solve by:
   - Placing the cursor over the **Solution** item under the **Static Structural_at t=0.088 sec (B5)** group, clicking on the right mouse button, and
   - Selecting the **Solve** item



The results will be ready in approximately 10 seconds.

To display the results:

1. Display the **Equivalent (von-Mises)** stresses by:
   - Clicking on the **Equivalent Stress** item in the Solution Group. The deformed body with stress contours will be displayed as shown in the figure.

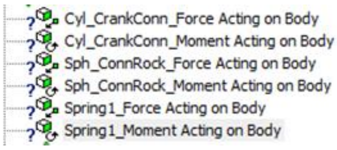


Repeating the Analysis

To repeat the analysis for the other three load cases:

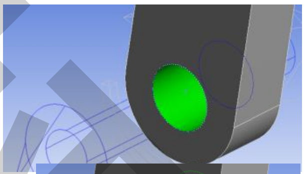
The following operations need to be done for each of the other three load cases:

- Repeat the operations on the six load entities
- Request von-Mises stresses
- Suppress the FE Boundary Conditions entity.
- Run the Analysis

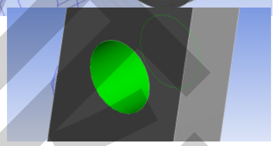


1. Delete moment load entries **Sph_ConnRock_Moment Acting on Body** and **Spring1_Moment Acting on Body**.

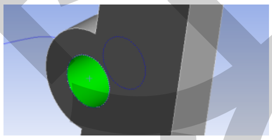
2. Associate the following four load entities:
   - Associate **Cyl_CrankConn_Force Acting on Body** and **Cyl_CrankConn_Moment Acting on Body** with the surface of the hole at the shaft bottom.



   - Associate **Sph_ConnRock_Force Acting on Body** with the surface of the hole at the shaft midpoint.



   - Associate **Spring1_Force Acting on Body** with the surface of the hole near the top of the shaft.



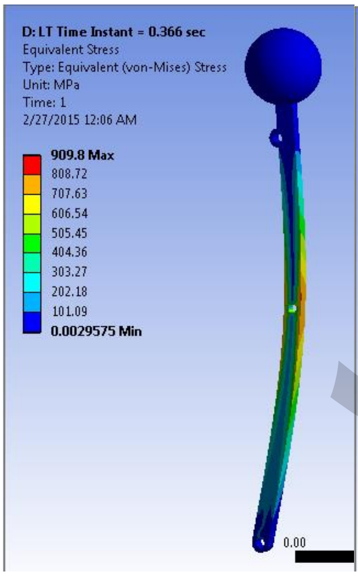
3. Request **Equivalent (von-Mises)** stresses by:
   - Placing the cursor over the **Solution** item in the Outline window and clicking on the right mouse button, and
   - Selecting **Equivalent (von-Mises)**

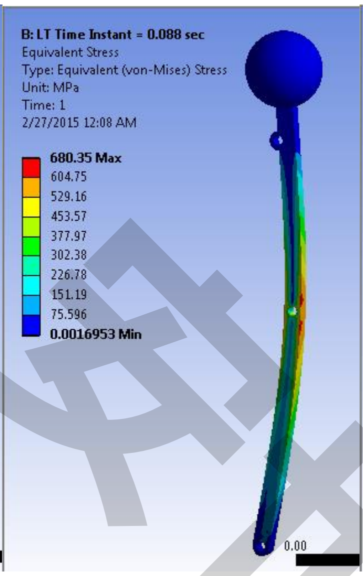
4. Suppress the FE Boundary Conditions entity by:
   - Placing the cursor over the **FE Boundary Conditions**, clicking on the right mouse button, and
   - Selecting **Suppress**.

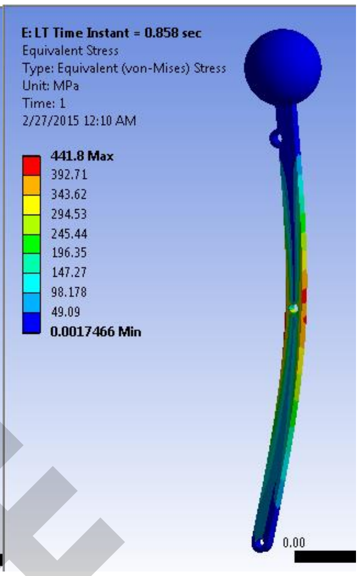
5. Run the Analysis by:
   - Placing the cursor over the **Solution** item, clicking on the right mouse button, and
   - Selecting the **Solve** item

To display the results:

1. Display the **Equivalent (von-Mises)** stresses for each of the additional cases by:
   - Clicking on the **Equivalent Stress** item in the Solution Group. The results for the three cases will be displayed as shown in the figure.







The results for all four load cases are similar, with similar deformed shapes and stress distributions. This is due to the spring load that is pulling continually to the left. However the max loads on the scale are 822 MPa, 910 MPa, 680 MPa, and 442 MPa, which reflects the cycling of the load during the work cycle. These stresses can be used to perform durability calculations.

Thanks for participating in this tutorial!