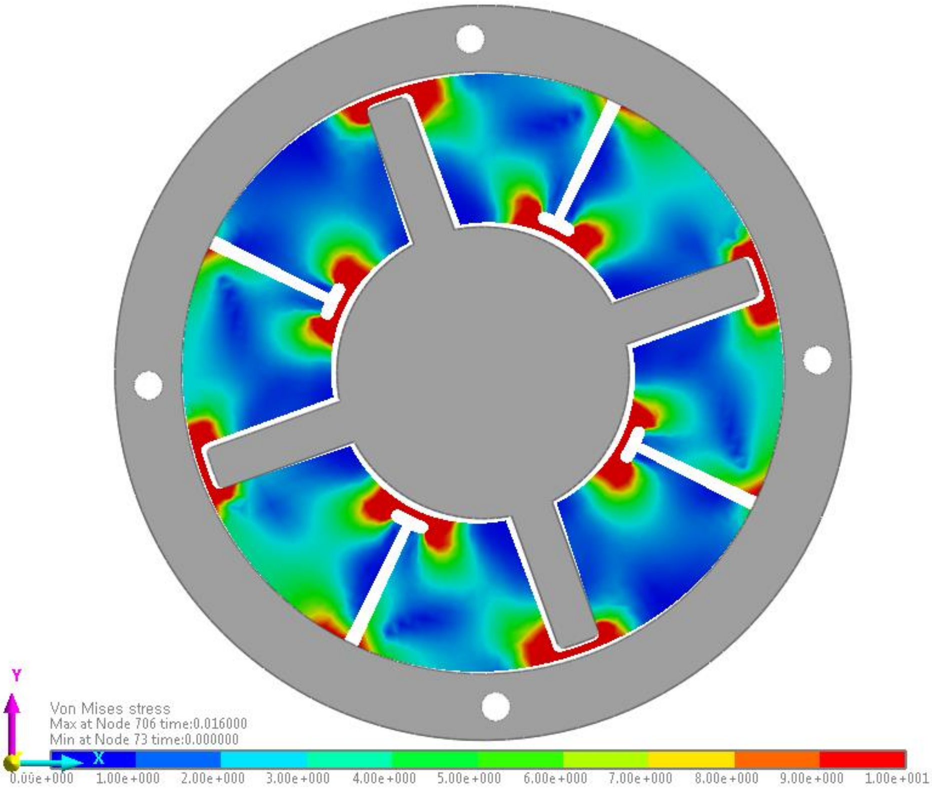
Compliant Clutch Tutorial (FFlex)



FunctionBay

Edition Note

This document describes the release information of **RecurDyn V9R4**.

Table of Contents

Chapter

1

Getting Started

Objective

The modeling and simulation of contact between flexible bodies is an important topic in multibody dynamics. The **RecurDyn FFlex toolkit** has powerful capabilities to define and simulate flexible bodies that have sliding or rolling contact with other bodies.

The example in this tutorial is a centrifugal clutch for a snowmobile, which relies on a compliant, or flexible, plate to operate. The benefits of using a compliant clutch plate, as opposed to having separate moving parts with springs, is that the part count is greatly reduced. This simplifies the manufacturing process and increases the reliability of the mechanism, as well.

Note that the model used in this tutorial is not based on any actual product or design concept but borrows from suggested applications for compliant mechanisms published by the Brigham Young University Compliant Mechanisms Research Center.

Audience

This tutorial is intended for new users of RecurDyn who have previously learned how to create geometry, joints, force entities, and 2D contacts. All new tasks are explained carefully.

Prerequisites

Users should first work through the 3D Crank-Slider, Engine with Propeller, and Pinball (2D contact) tutorials, or the equivalent. We assume that you have a basic knowledge of physics.

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 model geometry	10
Adding joints and forces	15
Defining surfaces and contacts	20
Creating a boundary condition	5
Simulating the model	15
Total:	**65**

Estimated Time to Complete

This tutorial takes approximately 65 minutes to complete. If you create all the patch sets yourself as explained in Appendix A, it will take approximately 25 additional minutes.

Chapter

2

Importing the Model Geometry

Task Objective

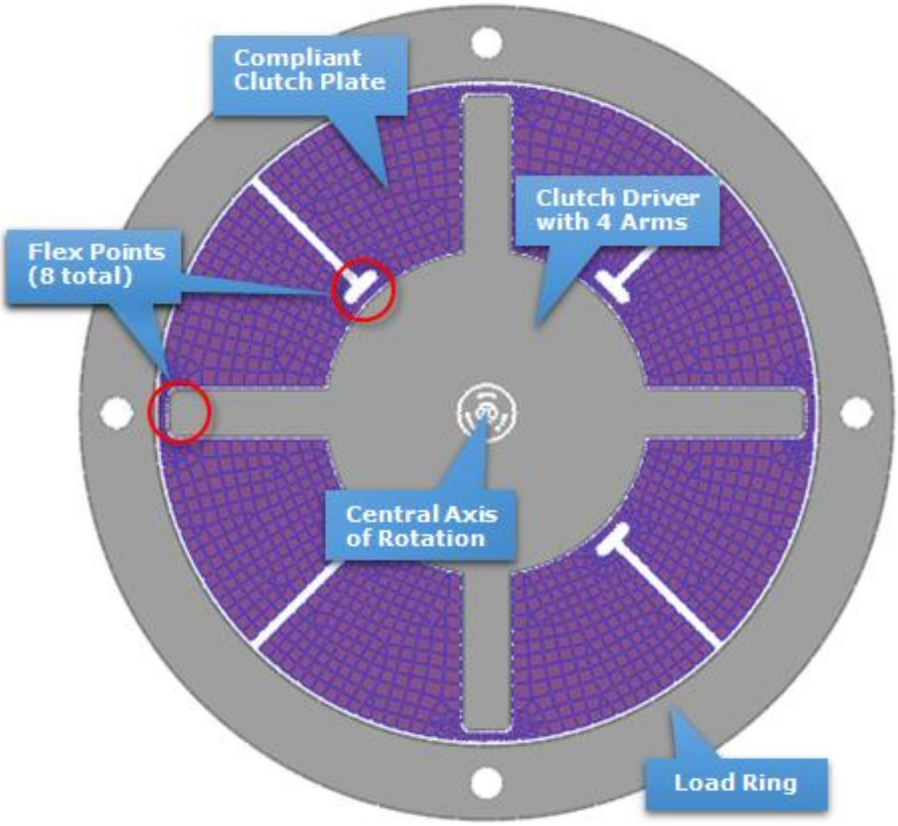
Learn how to import both rigid and flexible bodies to create the compliant clutch model.

Estimated Time to Complete

10 minutes

The Compliant Clutch Model

The model being used in this tutorial is a centrifugal clutch that relies on the flexibility of the clutch plate to operate. The following is a diagram of the clutch.



When power source rotates the clutch, driver is about the central axis, the arms push the compliant clutch plate into rotation, as well. As the rotational speed increases, centrifugal force pulls the clutch plate radially outwards, which flexes at eight flex points. As the plate moves outwards, it comes into contact with the outer ring to which the load is connected. Friction between the clutch plate and the load ring causes the load ring to begin to rotate, thereby transferring power from the clutch driver to the load.

In the RecurDyn model you will be creating, the compliant clutch plate will be treated as a flexible body, and a NASTRAN bulk data file will be imported for this part. The other bodies will be treated as rigid bodies, and geometry files will be imported for these parts.

Note:
To make the simulation more efficient for the tutorial, the model represents a 1 mm thick cross section of what the actual clutch would be. Driving and load torques have been scaled appropriately, and the same treatment can be applied to the simulation results you will obtain when interpreting them.

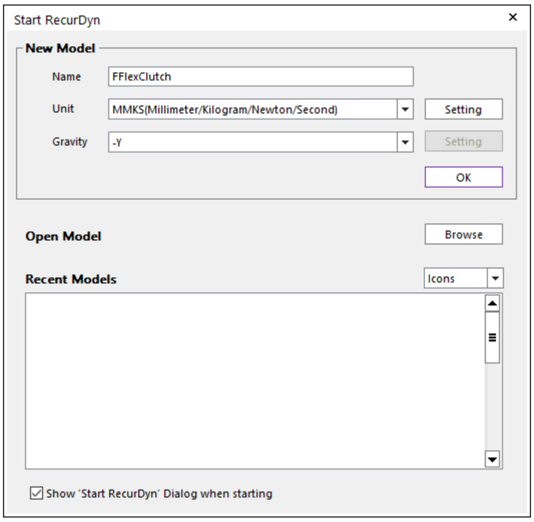
Starting RecurDyn

To start RecurDyn and create a new model:

1. On your Desktop, double-click the **RecurDyn** tool.

   RecurDyn starts and the New Model window appears.

2. Enter the name of the new model as **FFlexClutch**.
3. Click **OK**.



Importing the Compliant Clutch Plate Mesh Data

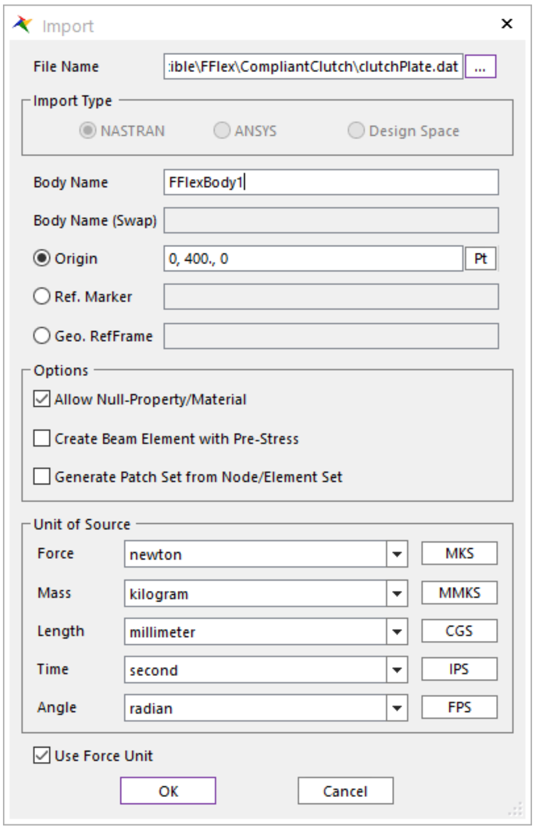
Now you will import the mesh data for the clutch plate into the model. In this case, the geometry was modeled in the CAD system in the correct locations. You do not need to adjust the geometry position.

To import a mesh data file:

1. From the **FFlex** group in the **Flexible** tab, click **Import**.
2. Select the point (0, 0, 0) as the origin by doing one of the following:
   - In the Input Window toolbar, enter 0, 0, 0.
   - In the Working Window, select the point.
3. Select the **ClutchPlate.dat**. (**The file location:** <Install Dir> \Help \Tutorial \Flexible \FFlex \CompliantClutch ask your instructor for the location of the directory if you cannot find it).
4. Click **Open**.

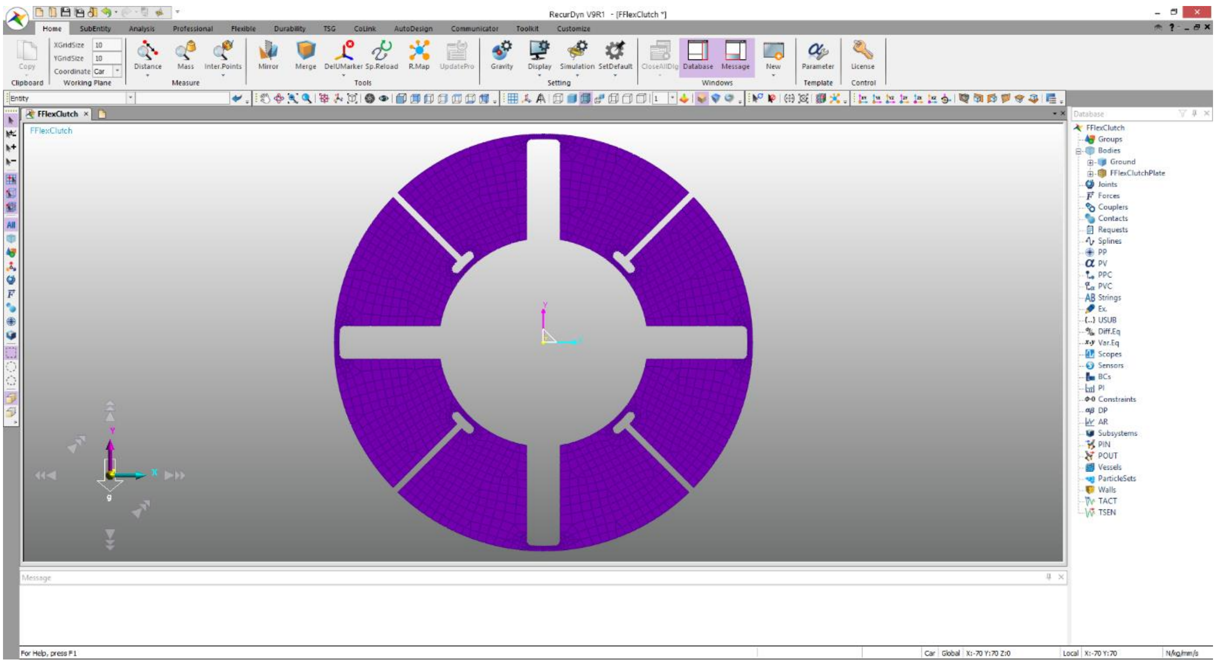
5. In the Import dialog box that appears, change **Body Name** to **FFlexClutchPlate**.
6. Click **OK**.

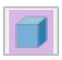
   The clutch plate appears in the Working window but is small.



7. Fit the geometry to the window by pressing **F**, and change the grid spacing, icon size, and marker size to **10**.

   The model appears as shown next.





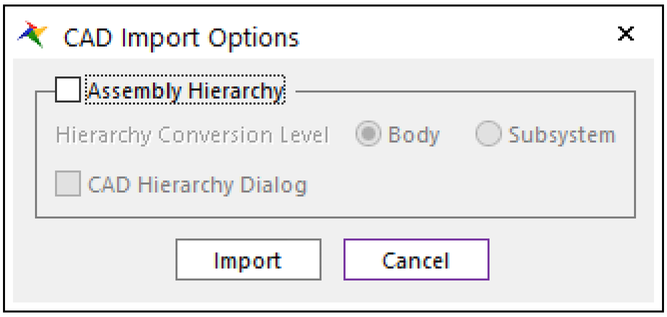
8. Change rendering mode to **Shade**.

Importing the Rigid Geometry

To import the driver:

1. From the **File** menu, click **Import**.
2. Set **Files of type** to **ParaSolid File (*.x_t,*.x_b ...)**.
3. In the **FFlex** tutorial folder, select the file: **clutchDriver.x_t**

   (**The file location:** <Install Dir> \Help \Tutorial \Flexible \FFlex \CompliantClutch, ask your instructor for the location of the directory if you cannot find it).
4. Click **Open**. The **CAD Import Options** window appears. Clear the **Assembly Hierarchy** checkbox and click the **Import** button.

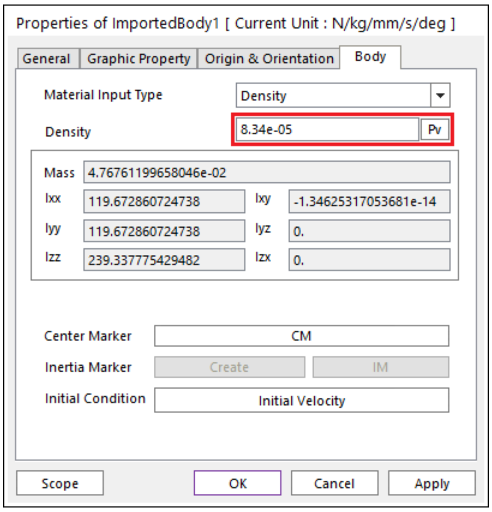


5. In the **Database window**, right-click **ImportedBody1**, and click **Property**.
6. On the **General** page, rename the body **ClutchDriver**.
7. Click **OK**.

To import the load ring:

1. Repeat steps 1 through 7 above, this time importing the file **clutchLoad.x_t**, and naming the body **ClutchLoad**. (**The file location:** <Install Dir> \Help \Tutorial \Flexible \FFlex \CompliantClutch, ask your instructor for the location of the directory if you cannot find it).
2. Click the **Body** tab.
3. In the **Density** text box, enter **8.34e-05**.

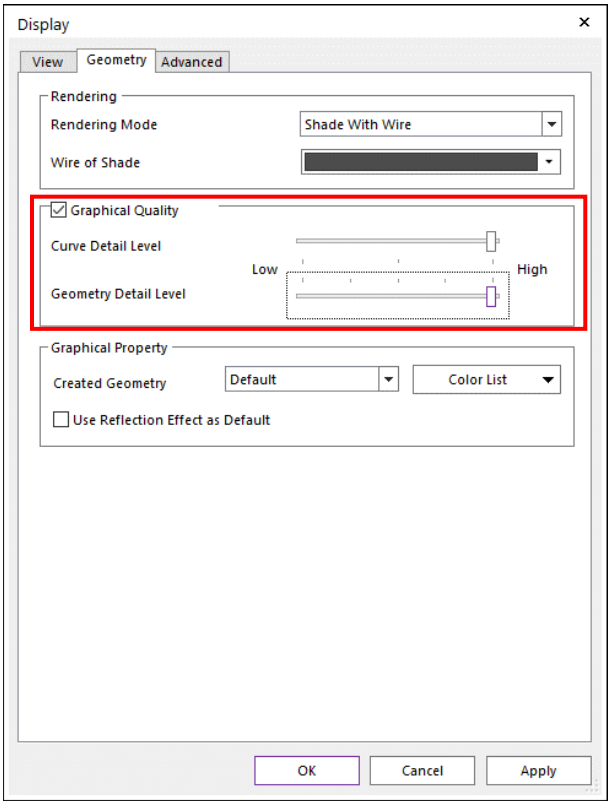
   This increases the rotational inertia of the load, simulating the weight of the snowmobile.



At this point, all the components of the clutch are present, but the rendering of the circular edges is poor. You will now improve the quality of the rendering, which will be important later when viewing the surface contact interactions in the animations.

To improve the rendering of the geometry:

1. From the **Setting** group in the **Home** tab, select the **Display Setting** tool.
2. Select the **Geometry** tab.
3. Move the sliders next to **Curve Detail Level** and **Geometry Detail Level** to **High**.
4. Click **OK**.



Save the model.

Chapter
3

Adding Joints and Forces

You will now add revolute joints, as well as driving and load torques, to the model.

Task Objective

Learn to:

- Create revolute joints.
- Create expressions, which will be used to create driving and load torques.

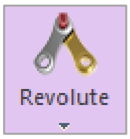
Estimated Time to Complete

15 minutes

Creating Revolute Joints

Now you will create two revolute joints, one for the driver and one for the load. The clutch plate will be constrained by the other geometry, so no joint must be added for that body.

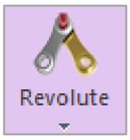
To create the driver joint:



1. From the **Joint** group in the **Professional** tab, click the **Revolute** tool.
2. Set the Creation Method toolbar to **Body, Body, Point**.
3. Click anywhere in the background of the **Working window** (not on any bodies) to select **Ground** as the first body.
4. Click the **ClutchDriver** body to select it as the second body.
5. In the **Input Window** toolbar, enter **0, 0, 0** as the joint location.
6. In the **Database window**, right-click **RevJoint1**, and click **Property**.
7. On the **General** page, rename the joint **Rev_driver**.
8. Click **OK**.

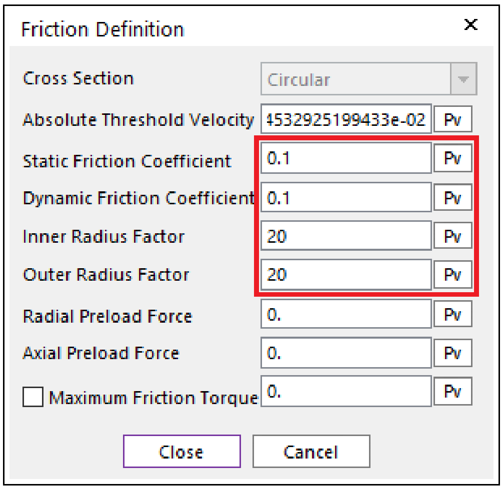
You will create the revolute joint for the clutch load in a similar way, but you will add friction.

To create the load joint:



1. Click the **Revolute** tool again. Ensure that the creation method is still **Body, Body, Point**.
2. Click anywhere in the background of the **Working window** to select **Ground** as the first body.
3. Click the **ClutchLoad** body to select it as the second body.
4. In the **Input Window** toolbar, enter **0, 0, 0** as the joint location.
5. In the **Database window**, right-click **RevJoint1**, and click **Property**.
6. On the **General** page, rename the joint **Rev_load**.
7. Click the **Joint** tab.
8. Click the checkbox next to **Include Friction** and click **Sliding**.

9. Make the following changes to the settings:
    - **Static Friction Coefficient**: 0.1
    - **Dynamic Friction Coefficient**: 0.1
    - **Inner Radius Factor**: 20
    - **Outer Radius Factor**: 20
10. Click **Close**.
11. Click **OK**.



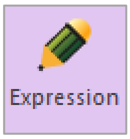
Creating the Torque Expressions

You will now create expressions for the driving and load torques:

- The driving torque will be a stepped transition from 0 to 10,000 N·mm over 0.01 seconds.
- The load torque will be in the opposite direction of the driving torque, and will vary with the load's rotational velocity, squared. This simulates the load that a snowmobile would experience due to wind resistance.

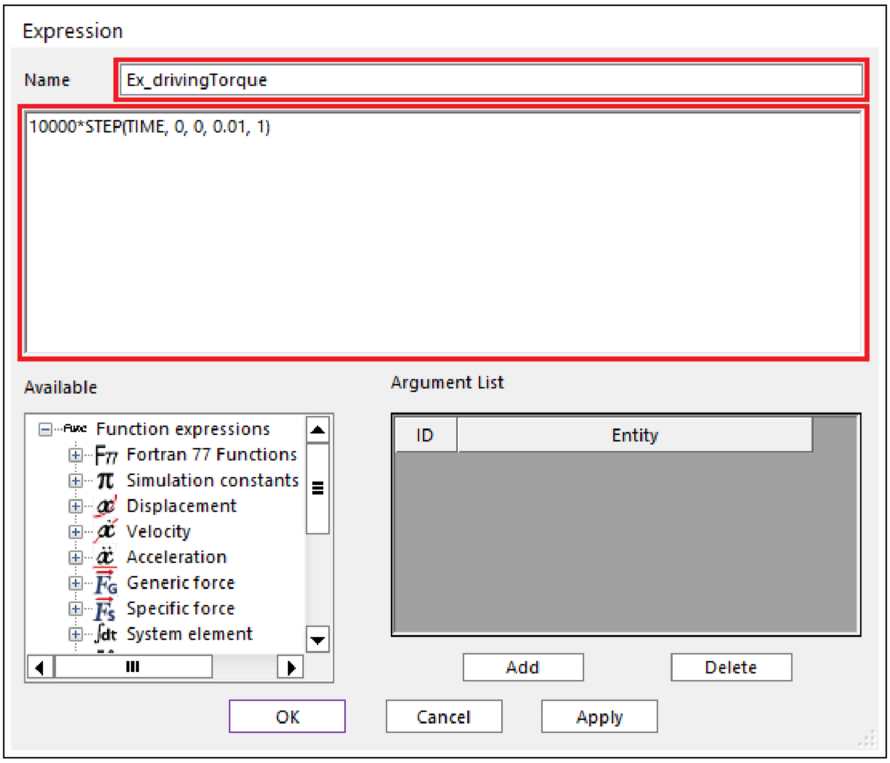
As stated earlier, the torques have been scaled down to match the model.

To create the driving torque expression:



1. From the **Expression** group in the **SubEntity** tab, click **Expression**
2. Click **Create**.

   Refer to the following diagram for the next several steps.

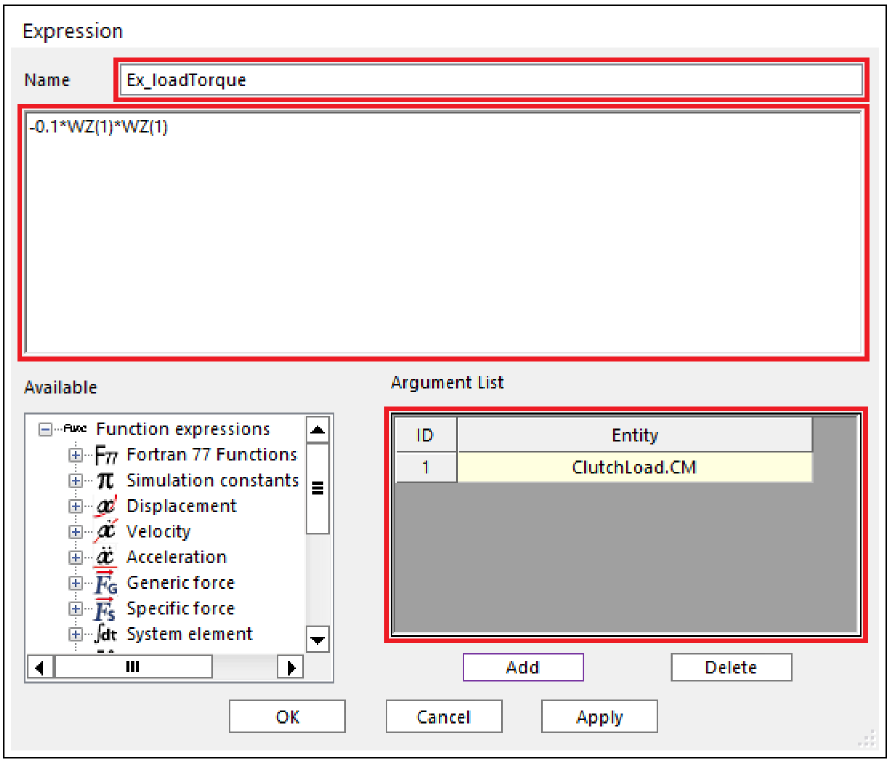


3. Change the name to **Ex_drivingTorque**.
4. For the expression, enter:
   - **10000*STEP(TIME, 0, 0, 0.01, 1)**
5. Click **OK**.

To create the load torque expression:

1. Click **Create** to create a new expression.

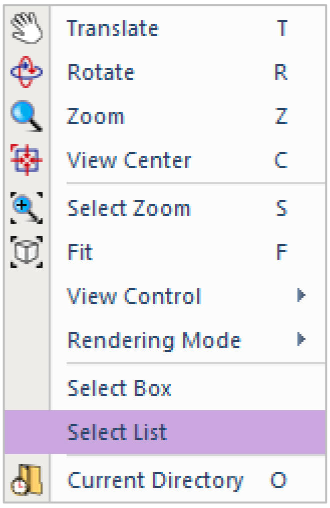
   Refer to the diagram below for the next several steps.



2. Name the expression **Ex_loadTorque**.
3. For the expression, enter:
   - **-0.1*WZ(1)*WZ(1)**
4. Under **Argument List**, click **Add**.
5. In the **Database window**, expand **Bodies** → **ClutchLoad** → **Markers** → **CM**.
6. Click and drag **CM** to the empty box under **Entity**, to enter it as the first argument.
7. Click **OK** twice.

Creating the Driving and Load Torques

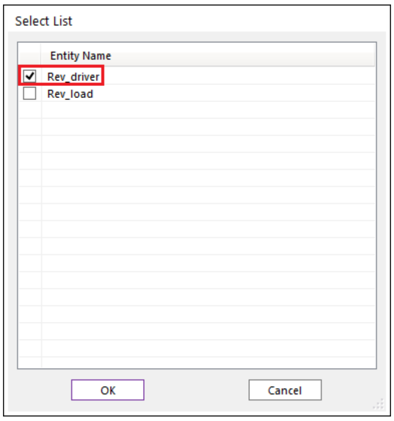
You will create driving and load torques to the clutch driver and load at the revolute joints and link them to the expressions you just created.



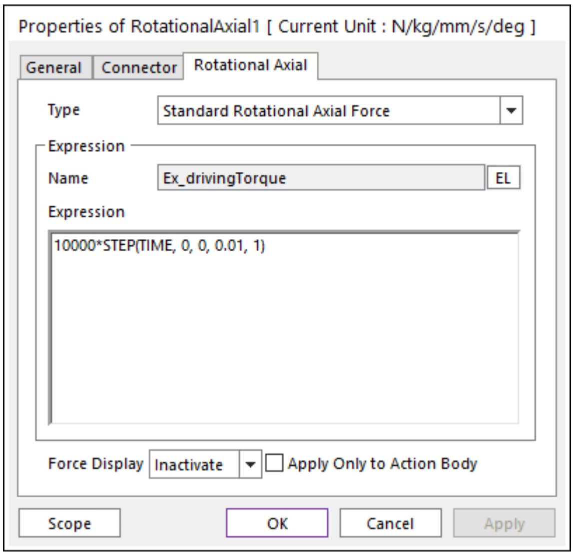
To create the driver torque:

1. From the **Force** group in the **Professional** tab, click the **Rotational Axial** tool.
2. Set the Creation Method to **Joint**.
3. In the **Working window**, right-click near the two revolute joints, and click **Select List**.

   A list of entities that are able to be selected at that screen location appears.
4. From the list, select **Rev_driver**.



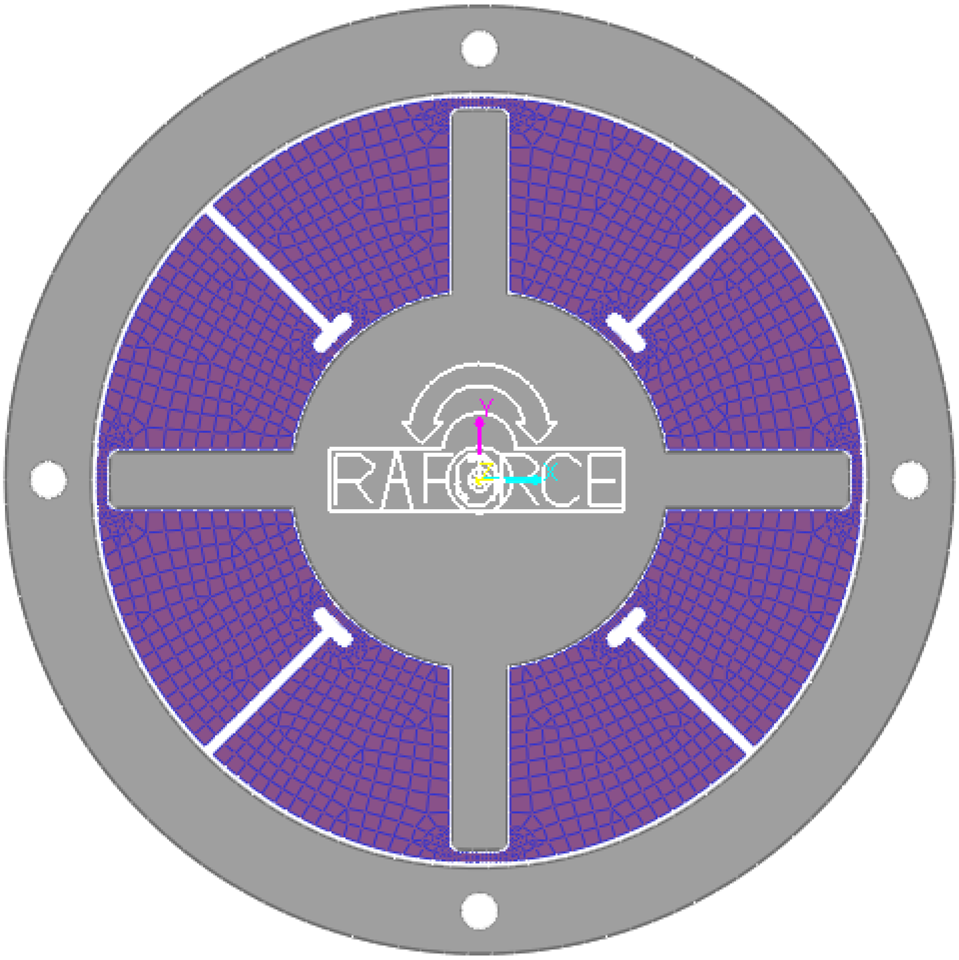
5. Click **OK**.



6. In the **Database window**, right-click **RotationalAxial1** (under **Forces**) and click **Property**.
7. Click **EL**.
8. From the Expression List window, select **Ex_drivingTorque**.
9. Click **OK**.

   The Properties dialog should now appear as shown at right.
10. On the **General** page, rename the force **RotAx_driver**.
11. Click **OK**.

After creating the driving torque, the model appears as shown below:



To create the load torque:

Repeat the steps above, this time:

- Selecting Rev_load as the joint.
- Selecting Ex_loadTorque as the expression.
- Renaming the force RotAx_load.

Save the model.

Chapter
4

Defining Surfaces and Contacts

In this chapter, you will create the necessary entities for a flexible surface to rigid surface contact.

Task Objective

You will:

- Create a contact surface on a flexible body.
- Define a contact between the flexible surface and a rigid surface.

Estimated Time to Complete

20 minutes

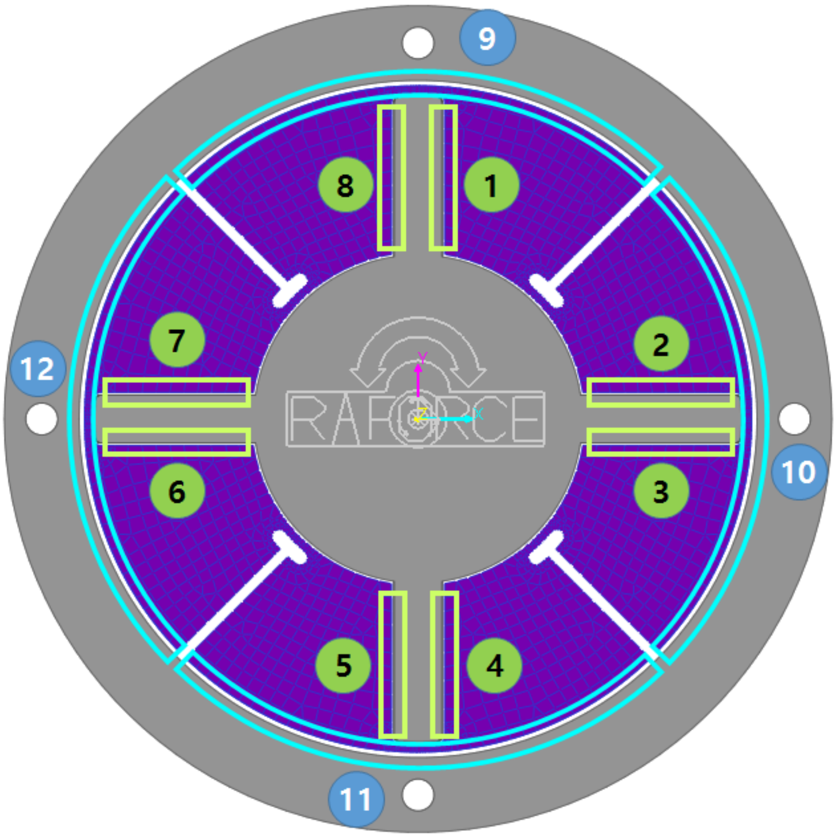
Contacts in the Model

You will now create 12 contacts in the model, each representing the interaction between two surfaces. In this model, some contacts will be between surfaces on the driver and the plate, and others between the plate and the load.

- If the contact surface is on the plate, it is defined by selecting a set of element faces from the finite element mesh. This set of element faces is referred to as a Patch Set.
- If the contact surface is on the driver, it is defined as a face surface.

The figure below shows the 12 regions where the contacts will be created, and the numbering scheme that will be used in the rest of the tutorial.

Contact Layout Diagram



Note: Not all of the possible contact regions in the clutch have been selected to be modeled. This is because when the driver rotates the clutch plate, centrifugal force will pull the plate elements outwards, towards the clutch load. With this in mind, you can reduce the number of contacts and, therefore, reduce the simulation time.

Creating the Patch Sets

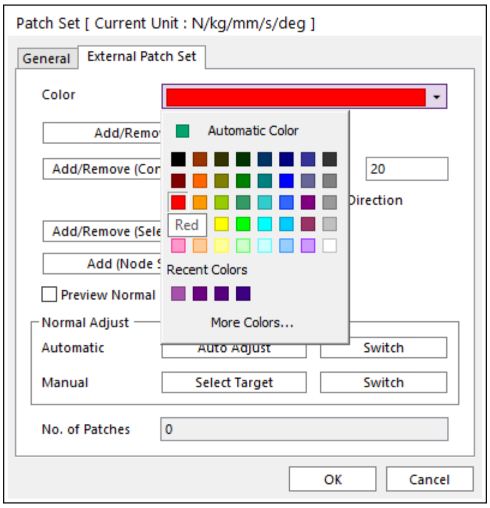
As explained before, if the surface is on the plate, you need to create a patch set before defining the contacts. There are nine patch sets that you need to create. To make it easy for you, after creating the first patch set, you can open an intermediate model that contains the patch sets already defined. If you would like more experience creating the patch sets, follow the instructions in *Appendix A, Creating the Remaining Patch Sets*, on page 47.

To create a patch set on a flexible body:

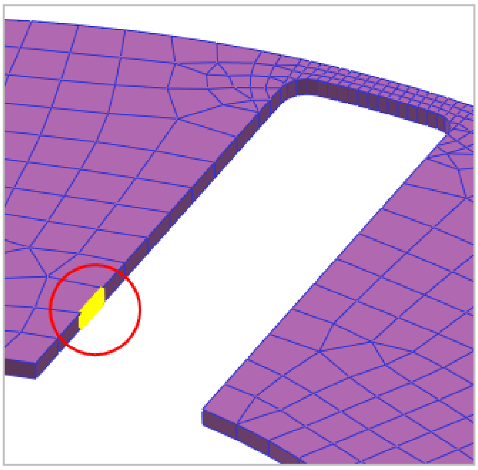
1. In the **Database window**, right-click **FFlexClutchPlate**, and click **Edit**.

   RecurDyn is now in body-editing mode for the clutch plate.

2. From the **Set** group in the **FFlex Edit** tab, click **Patch**.
3. Select **red**, as shown at right.
4. Change **Tolerance (Degree)** to **20**.
5. Click **Add/Remove (Continuous)**.

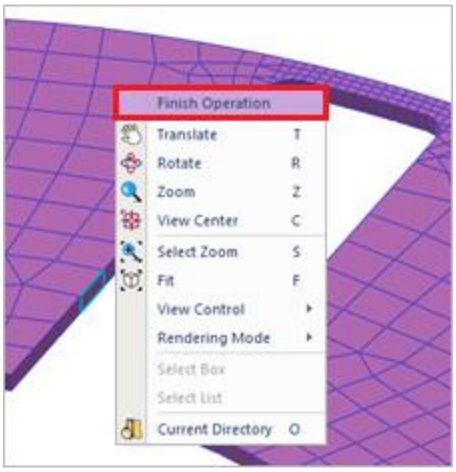


6. Zoom in carefully to the **FFlexClutchPlate** geometry as shown in the figure at right and select the element as shown at right, highlighted in light yellow.

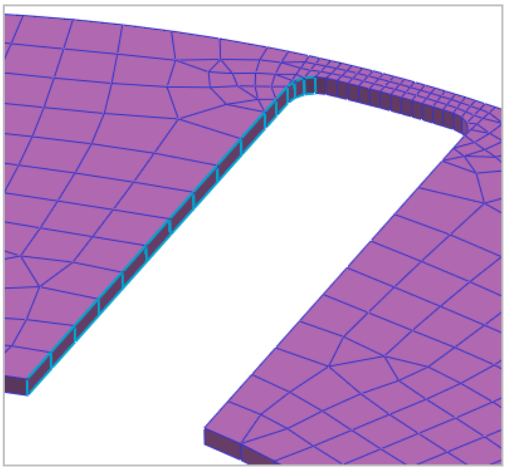


7. In the **Working window**, right-click, and click **Finish Operation**.

8. Click **Add/Remove** in the patch set dialog.

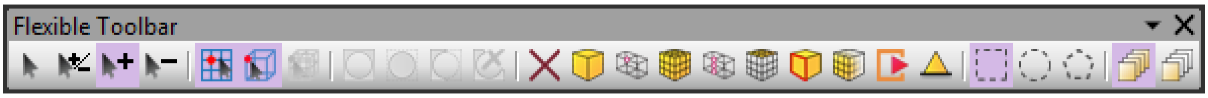


9. Holding down the **CTRL** key, select the element faces of the first contact surface, as shown at right, highlighted in sky blue and right-click, and click **Finish Operation**.



Tip: How to select faces easily:
To deselect elements which you mistakenly selected, continue to hold the **CTRL** key and click again on the same element. It will toggle between being selected and deselected.
In the current selection mode, if you forget to hold the CTRL key down and select another element, all the previously selected elements will be deselected and only the element you just clicked on will be selected.

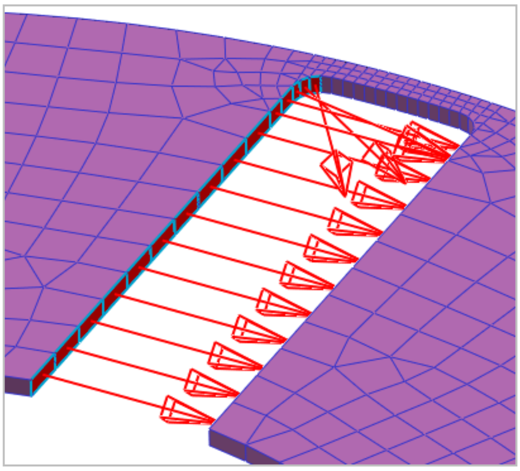
- Select **Add** from the **Flexible Toolbar**, as shown below.



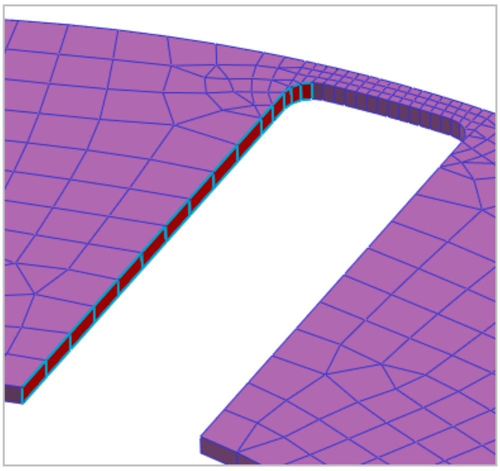
- Also note that there is a Remove option. Selecting this will change the selection behavior so that elements are only deselected when clicked on.

---

10. In the patch set dialog, click the checkbox next to **Preview Normal**.
11. Ensure that all the normals are pointing out from the surface, as shown in the figure on the right. If not, click **Auto Adjust** (to the right of **Automatic**) to switch all of the normals to be consistently oriented in the same direction. If that direction is not the desired direction, click Switch to **switch** the normals.



12. Click **OK**.
13. The patch set should now appear in red as shown at right.



At this point, you can either:

- Open the intermediate model, **FFlexClutch_Intermediate.rdyn**, with the remaining patch sets already defined and continue with the next section, **Creating the Face Surfaces**. (The file location: <Install Dir> \Help \Tutorial \Flexible \FFlex\CompliantClutch)
- (OR, Follow the instructions in Appendix A, Creating the Remaining Patch Sets, on page 46, to define the remaining patch sets yourself to gain experience.)

Save the model.

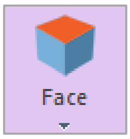
- If you use **FFlexClutch_Intermediate.rdyn** model, Save as other path because you cannot simulation this model in install path.

Creating the Face Surfaces

To create a face surface on a rigid body:

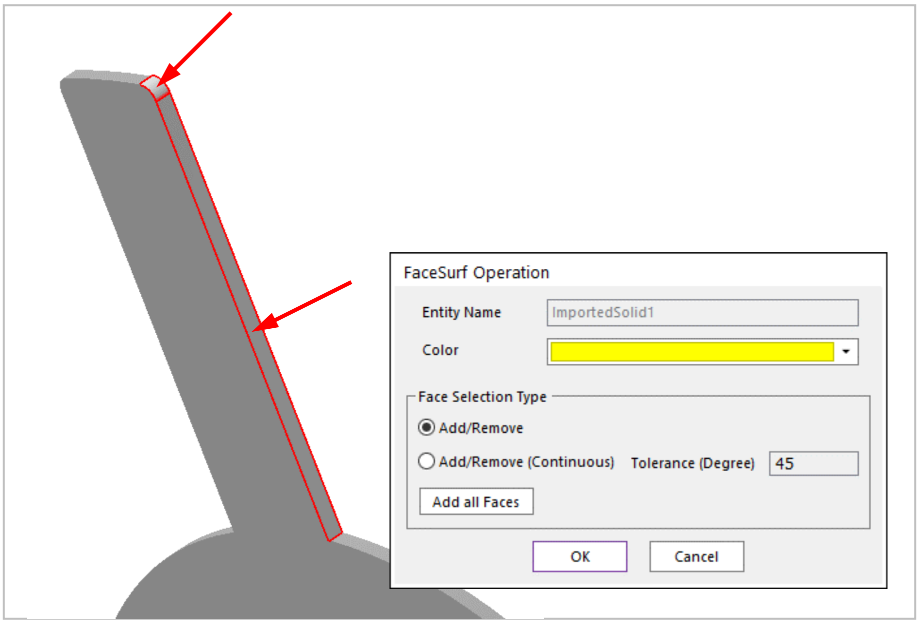
1. In the **Database window**, right-click **ClutchDriver**, and click **Edit**.

   You are now in the body edit mode for the clutch driver.



2. From the **Surface** group in the **Geometry** tab, click **Face Surface**.
3. Set the Creation Method toolbar to **Solid(Sheet), MultiFace**.
4. Select the **ClutchDriver** geometry on the Working window.

   The **FaceSurf** Operation dialog box appears.
5. On the top arm, holding down the **CTRL** key and select the two faces on the right side, as shown below.



6. Change the color to **Yellow**.
7. Click **OK**.

To create the remaining face surfaces:

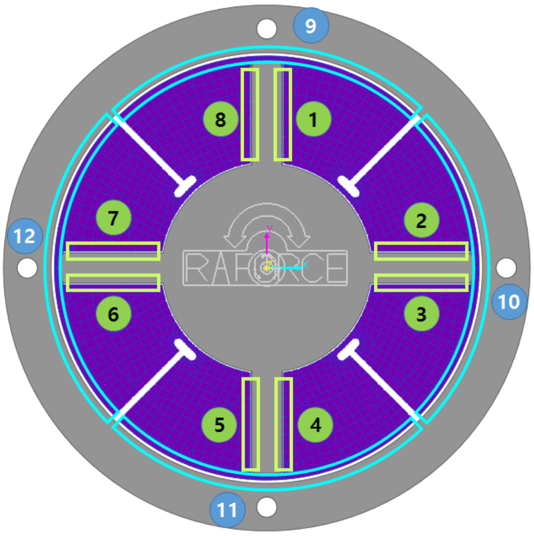
- Repeat the steps above, proceeding around the clutch driver, until all eight face surfaces have been created.

- Click **Exit**.

Creating the Contacts

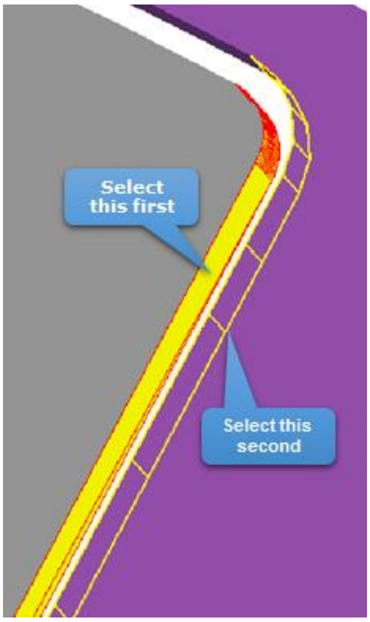
Now that you have defined the necessary contact surfaces on the clutch plate and driver, you can create the contacts. (refer to the figure below shows the 12 regions where the contacts will be created)

Contact Layout Diagram



To create a driver arm contact:

In the **1** region, you can create a contact.



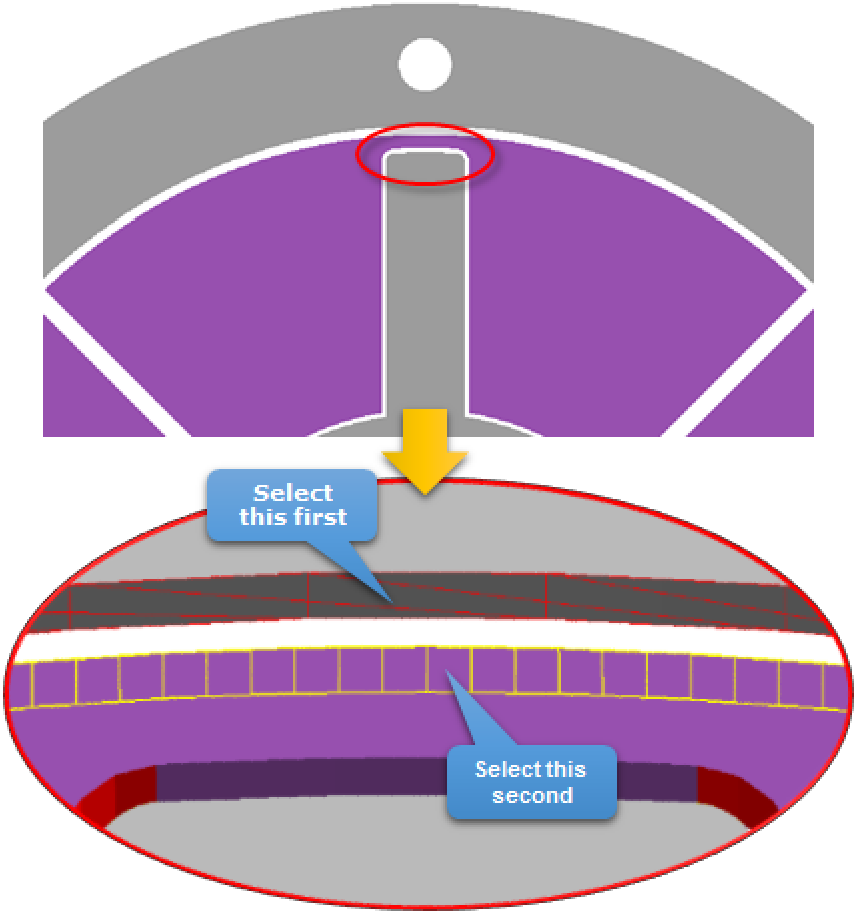
1. From the **FFlex** group in the **Flexible** tab, click **FSurf.ToSurf.Contact**.
2. Set the Creation Method toolbar to **Surface, PatchSet**.
3. Select the first face surface that you had created on the clutch driver, as shown at right.
4. Select the first patch set that you had created on the **FFlexClutchPlate**, as shown at right.

To create the remaining driver arm contacts:

Repeat the steps above for the remaining driver-to-plate contacts (**2-8** regions), proceeding clockwise around the clutch driver.

To create a plate-to-load contact:

1. Repeat the same steps for creating a contact (**9** region), but this time for the uppermost plate-to-load contact. Select the inner cylindrical surface of the load, as shown below. The edges of this surface are highlighted in light red when selecting it

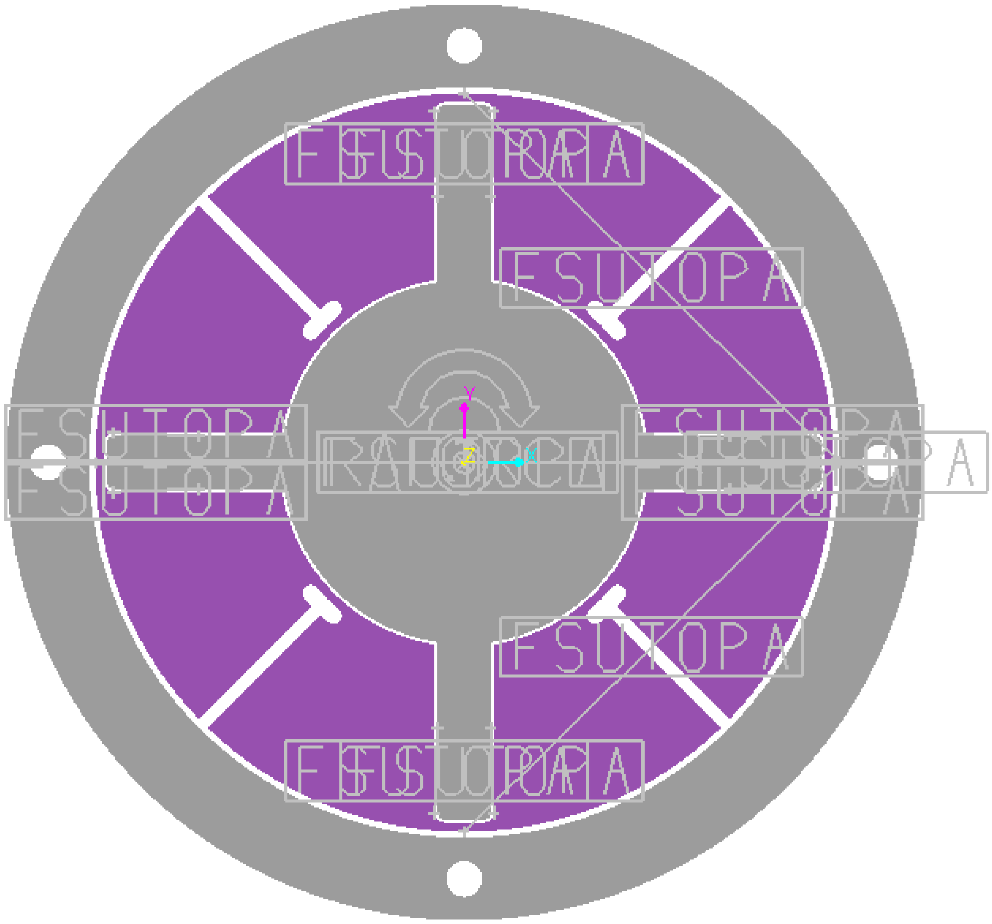


r

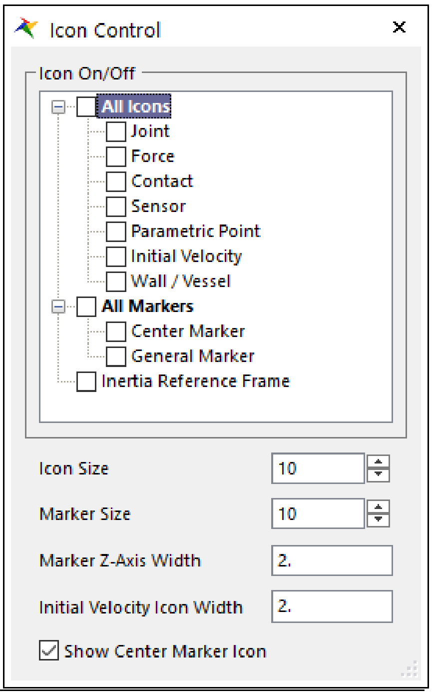
2. Select the uppermost patch set. As with the other patch sets.

To create the remaining plate-to-load contacts:

- Repeat the steps above for the remaining the plate-to-load contacts (10-12 regions), proceeding clockwise around the clutch driver.
- The model appears as shown next.



The icons for the contact clutter the display of the model and make upcoming steps more difficult. Because of this, you will turn off the display of the contacts, as well as the revolute joints and rotational axial forces.



To clean up the display of the model:

1. From the **Render Toolbar**, click **Icon Control**.
2. Clear the selection of the check boxes.
3. Close the **Icon Control** dialog box (use the **x** in the upper right corner).

Tuning the Contacts

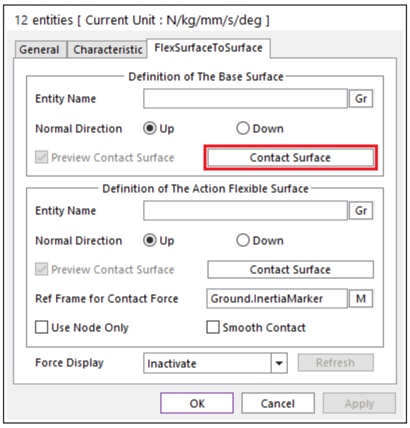
Now that the contacts have been created, you will tune them. You will make certain changes to all the contacts, first, and then add friction to the plate-to-load contacts.

The friction characteristics of the plate-to-load contacts are important because the clutch relies on this friction to transfer power from the driver to the load. To keep the simulation time short, you will use a friction coefficient that is higher than the normal range.

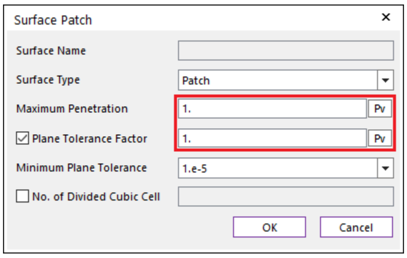
For the driver arm contacts, you are less concerned with the friction. For these contacts, you will neglect friction to shorten the simulation time.

To tune the contacts:

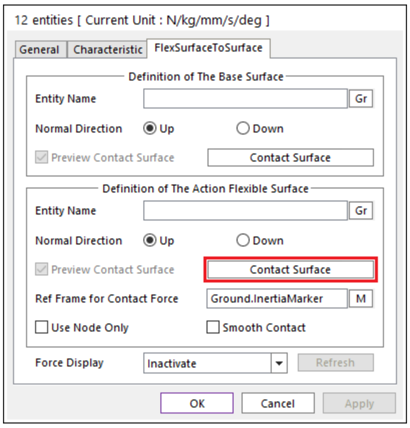
1. In the **Database window**, use the **Shift** key again to select all twelve contacts at once (**FSurfaceToSurface1 – FSurfaceToSurface12**).
2. Right-click the selected contacts and click **Property**.
3. On the **FlexSurfaceToSurface** page, click the first **Contact Surface** button, for the **Base Surface**, as shown in the figure on the right.



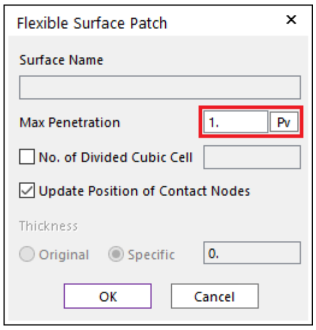
4. In the Surface Patch dialog box, make the following changes:
   - **Max Penetration**: 1
   - **Plane Tolerance Factor**: 1
5. Click **OK**.



6. Click the second **Contact Surface** button, for the **Action Flexible Surface**.

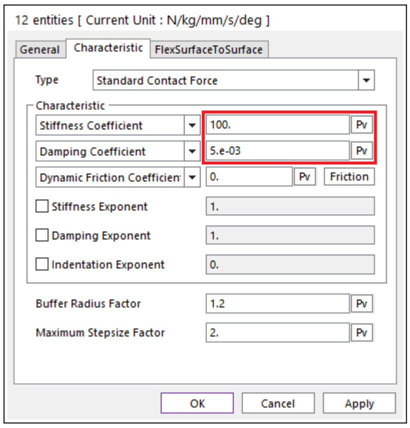


7. Change **Max Penetration** to **1**.
8. Click **OK**.



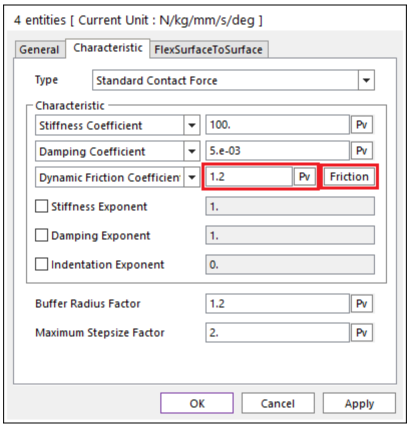
9. Select the **Characteristic** tab.

10. Make the following changes:
    - **Spring Coefficient**: 100
    - **Damping Coefficient**: 5.e-03
11. Click **OK**.

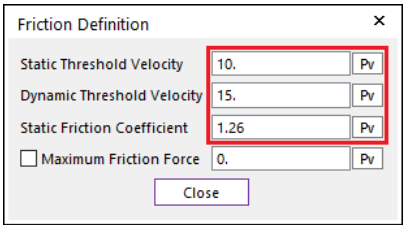


To add friction to the plate-to-load contacts:

1. In the **Database window**, use the **Shift** key again to select the last four contacts (**FSurfaceToSurface9 – FSurfaceToSurface12**).
2. Right-click the selected contacts and click **Property**.
3. Select the **Characteristic** tab.
4. Make the following change:
   - **Dynamic Friction Coefficient**: 1.2
5. Click **Friction**.



6. Make the following changes:
   - **Static Threshold Velocity**: 10
   - **Dynamic Threshold Velocity**: 15
   - **Static Friction Coefficient**: 1.26
7. Click **Close**.
8. Click **OK**.



Save the model.

Chapter
5

Creating a Boundary Condition

In this chapter, you will create a boundary condition for all the finite element nodes of the clutch plate. The boundary condition will limit the motion of these nodes in the Z-direction to zero. This will simulate the actual clutch, in which a structure would be in place to prevent the clutch plate from falling out of the clutch. In addition, it will reduce simulation time by reducing a degree of freedom for all elements.

Task Objective

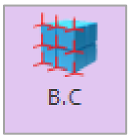
Learn to set up a boundary condition for all the elements in the clutch plate.

Estimated Time to Complete

5 minutes

Creating a Boundary Condition

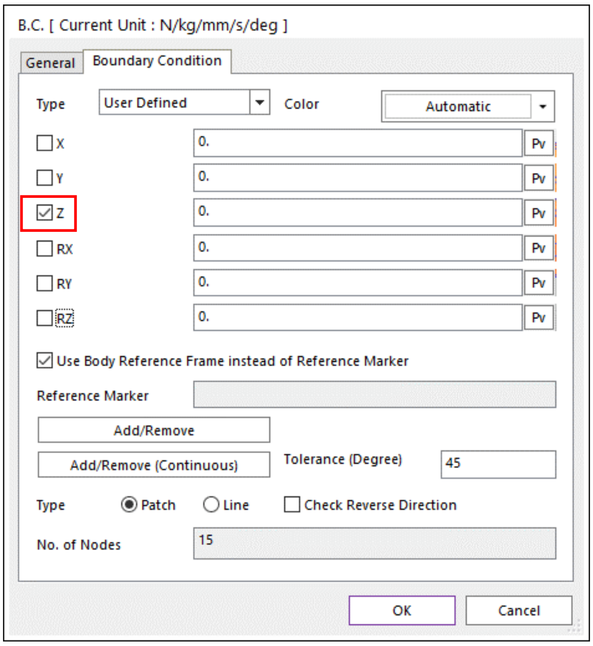
To set up a boundary condition:

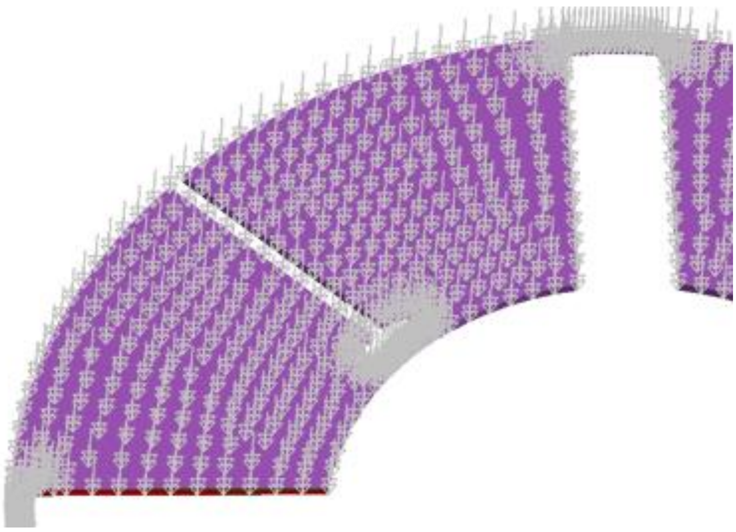


1. In the **Database window**, right-click **FFlexClutchPlate**, and click **Edit**.
2. From the **FFlex Edit** group in the **FFlex Edit** tab, click **B.C**.
3. Clear the check boxes next to the axes of motion, except for **Z**, as shown on the right.
4. Click **Add/Remove**.
5. In the Working window, select all the nodes by clicking and dragging a select box around the entire clutch plate.
6. In the **Working window**, right-click, and click **Finish Operation**.
7. Click **OK**.

   The clutch plate now appears as shown below, with arrows pointing in the negative Z-direction.

8. Click **Exit**.





Save the model.

Chapter
6

Simulating the Model

In this chapter, you will run a simulation, enable contoured viewing of internal stresses, and play the simulation.

Task Objective

You will learn to:

- Set up and run a simulation.
- Enable the viewing of internal stresses.
- Plot and interpret the simulation results.

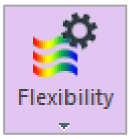
Estimated Time to Complete

15 minutes

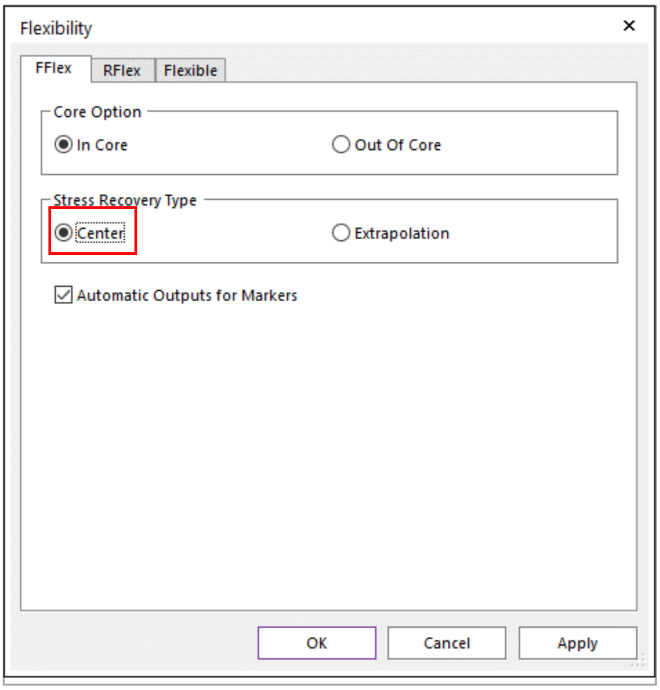
Setting Stress Recovery to Center

Before running the simulation, you will change the Stress Recovery setting for the FFlex body. This setting should be made in most models for the contour plots to be interpreted correctly.

To set the Stress Recovery to Center:



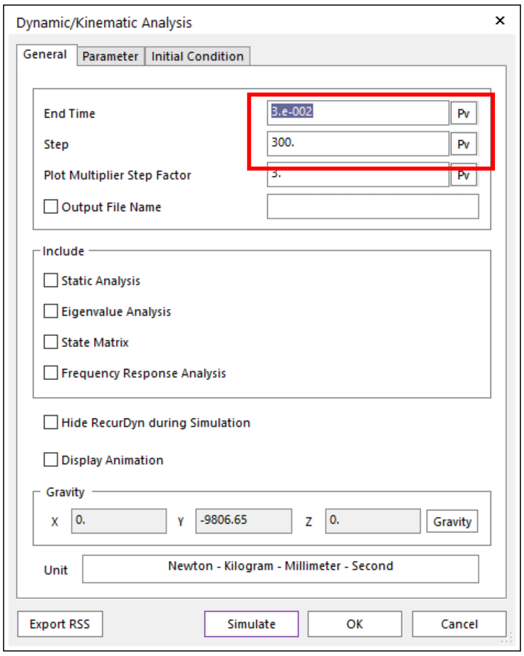
1. From the **Setting** group in the **Home** tab, click **Flexibility Setting**.
2. Select the **FFlex** tab.
3. Under **Stress Recovery Type**, click **Center**.
4. Click **OK**.



Setting up and Running the Simulation

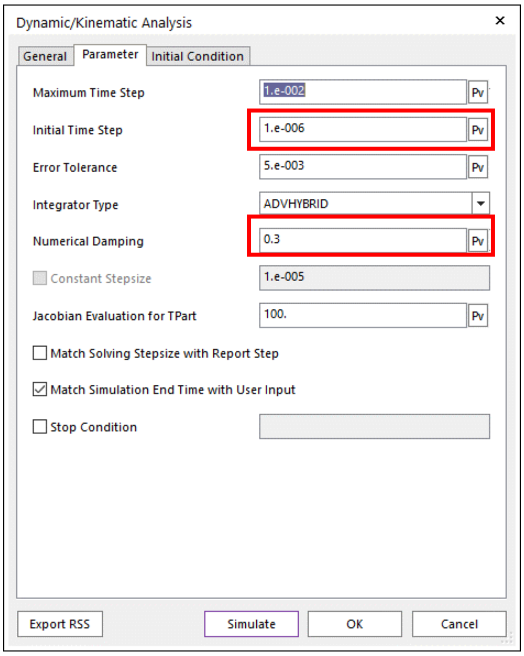
To set up and run the simulation:

1. From the **Simulation Type** group in the **Analysis** tab, click **Dyn/Kin**.
2. On the **General** tab, make the following changes:
   - **End Time**: 3.e-02
   - **Step**: 300
   - **Plot Multiplier Step Factor**: 3



3. On the **Parameter** tab, make the following change:
   - **Initial Time Step**: 1.e-06
   - **Numerical Damping**: 0.3
4. Click **Simulate**.

The simulation runs for 1 to 5 minutes.



Enabling Contoured View of Stresses in the Clutch Plate

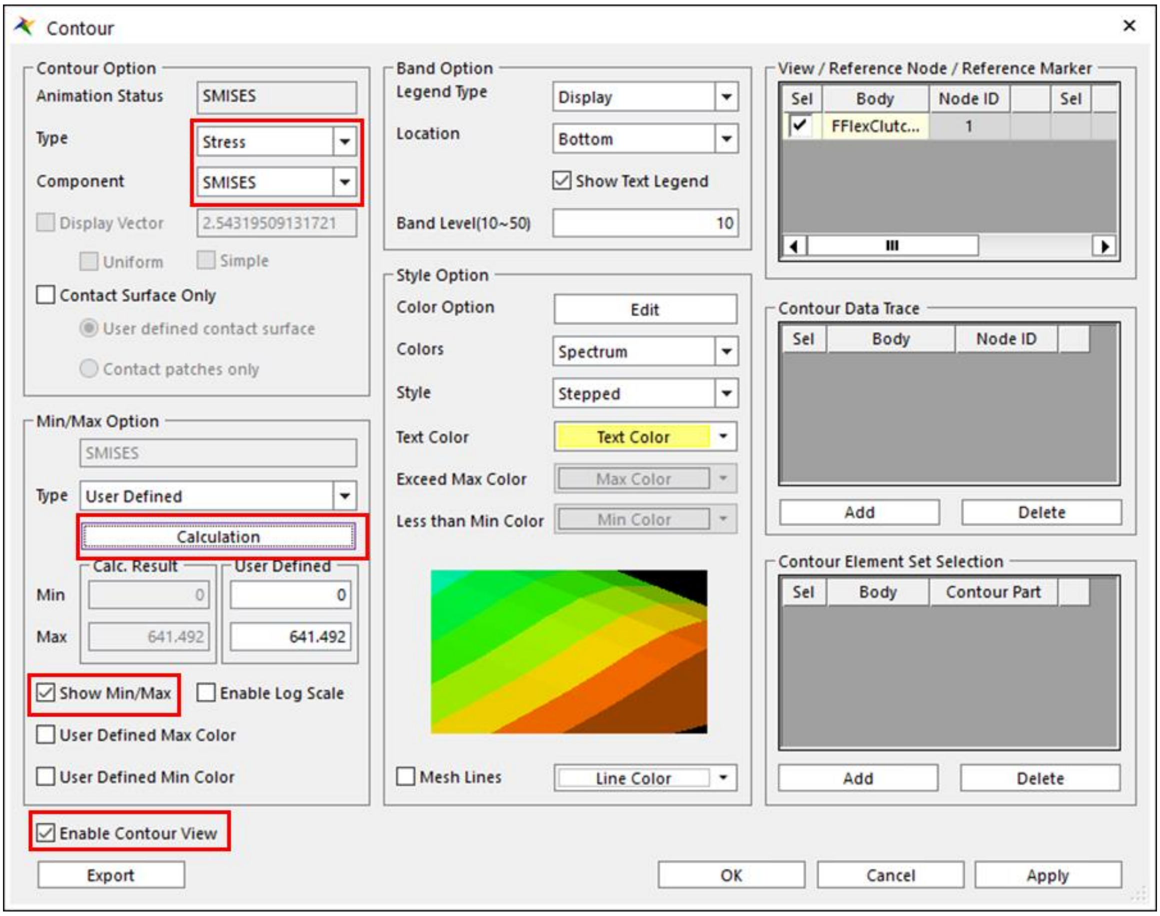
To visualize the internal stresses in the clutch plate, enable the contour display.

To enable the contour display:

1. From the **FFlex** group in the **Flexible** tab, click **Contour**.
2. Near the bottom of the dialog box, as shown in the figure on the right, select **Enable Contour View**.
3. In the **Contour Option** section, set **Type** to **Stress**.
4. At the bottom of the list, click **SMISES**.
5. Click **Calculation**.

   This determines the minimum and maximum stress that occurs throughout the simulation.
6. Click the checkbox next to **Show Min/Max**.
7. Click **OK**.

   Now that you have run the simulation and enabled contour display, you can play the animation.



Viewing the Animation Results

To play the animation:

- From the **Animation Control** group in the **Analysis** tab, click **Play/Pause**.

The animation shows the driver begin to spin and then contact the plate. As the plate and driver increase in rotational speed, the plate expands outwards and makes contact with the load. At the end of 0.03 second, the load rotates with the plate, but not at the same speed.

When the animation finishes:

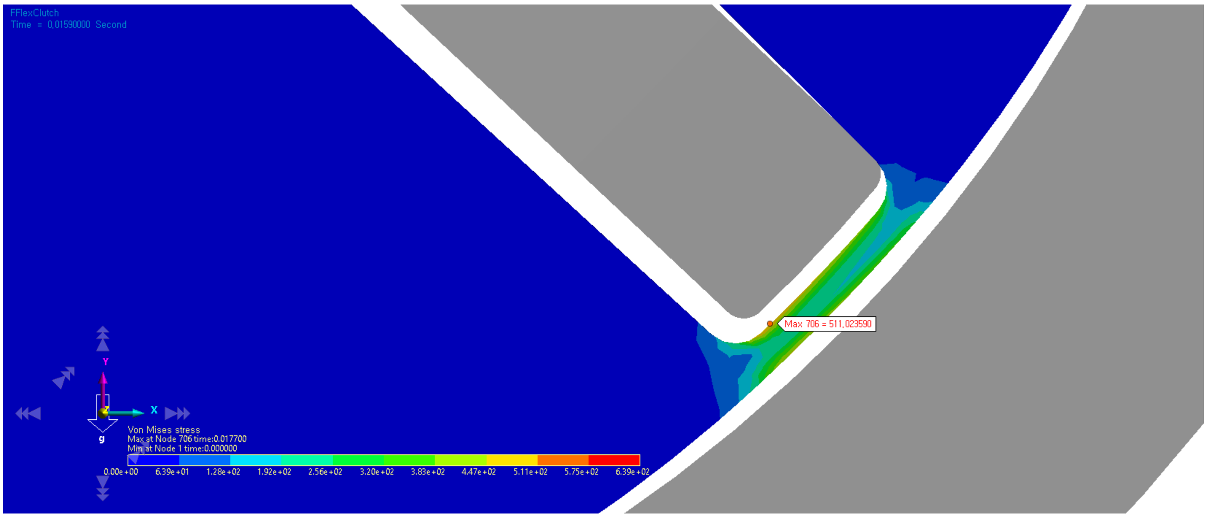
1. Examine where the areas of highest stress are.

   As expected, they are at the flex points.

   RecurDyn should also indicate which of the flex points has the highest stress.

2. Zoom into this part of the clutch plate, and view in Shade mode.

   It should appear as shown below.



RecurDyn reports the node number and stress at the maximum stress point. At the middle of the simulation, the maximum stress is **511.023 N/mm2**, occurring at **Node 706**.

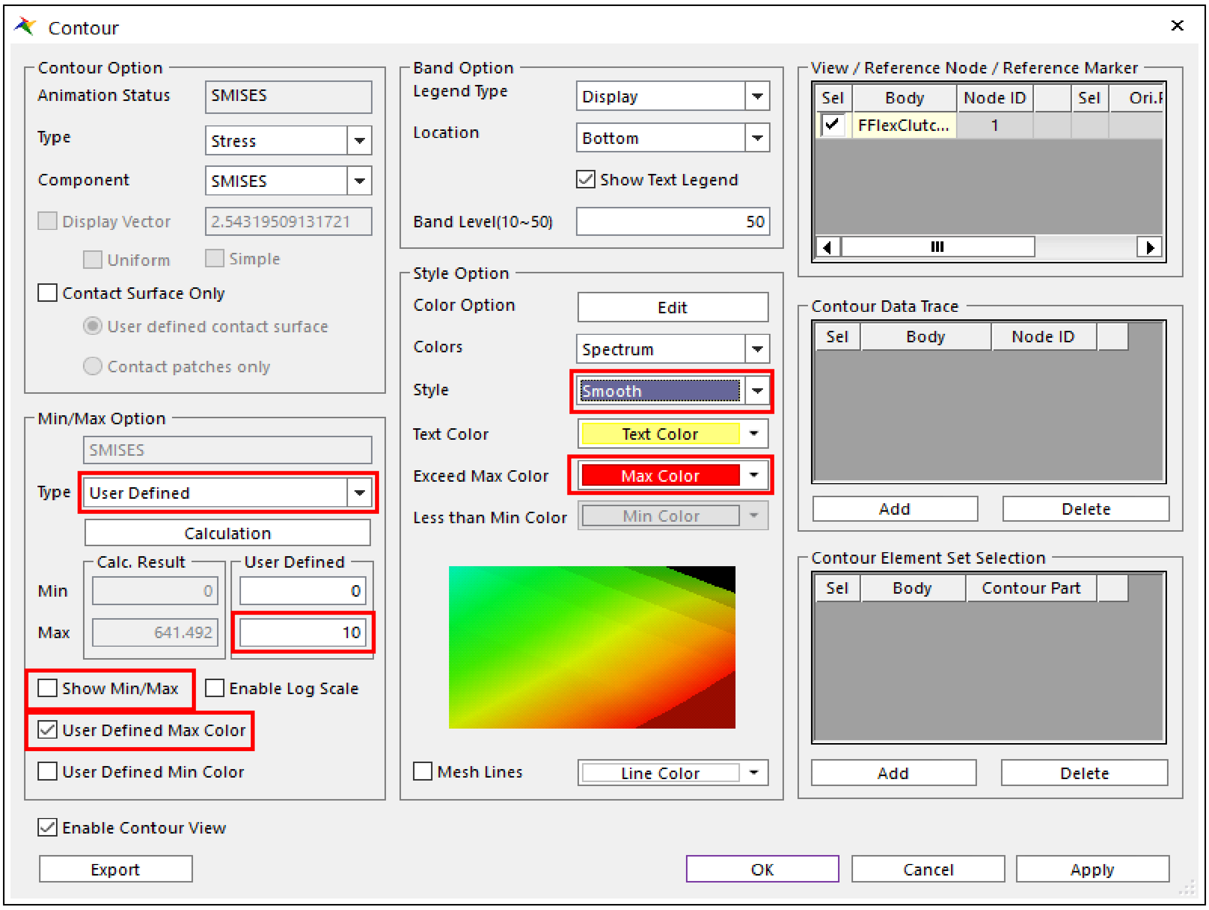
Did you notice anything else about the results that you might not have expected? Even though the structure of the clutch is symmetrical, the deformation of the plate at the end of the simulation is asymmetrical. To examine why this occurred, adjust the scale and appearance of the contour plot as follows.

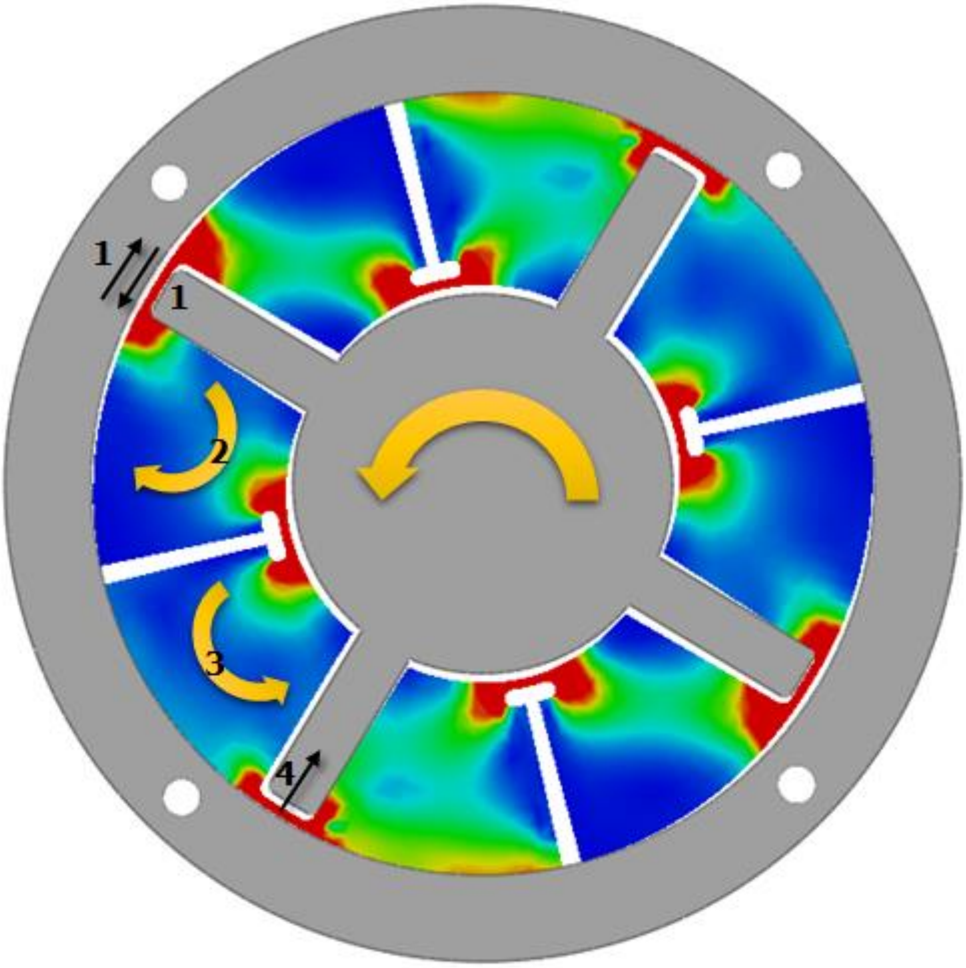
To adjust the contour display:

1. From the **FFlex** group in the **Flexible** tab, click **Contour**.
2. Under **Min/Max Option**, change **Type** to **User Defined**.
3. Set the **Max** to **10**.
4. Clear **Show Min/Max** option.
5. Under **Style Option**, set **Style** to **Smooth**.
6. Check **User Defined Max Color** option.
7. Change **Exceed Max Color** to **red**
8. Click **OK**.
9. Play the animation again and observe the results.

With the given friction settings, this particular design shows instability, with a tendency to shift to an asymmetrical mode.

At some point during the animation, you see the results shown next.





One explanation for the asymmetrical behavior could be as follows.

Forces (1) on the left plate segment from the clutch driver and load friction cause a clockwise torque (2) on the plate segment. The torque is transferred to the adjacent segment below, where it becomes a counterclockwise torque (3). The torque in turn pulls (4) the lower-left flex points away from the load, therefore, creating the asymmetry. The same chain of events occurs on the opposite side of the plate, as well.

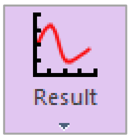
You can confirm then from looking at the contour plots that the load path described above follows the areas of highest stress and ends where the stress begins to fade.

Plotting the Simulation Results

You will now use RecurDyn's plotting capability to visualize more results. We assume that you have a basic knowledge of how to plot data in RecurDyn. If you need to review these methods, refer to the 3D Crank-Slider tutorial.

First, you will compare plots of rotational velocities and torques to the animation, in real time.

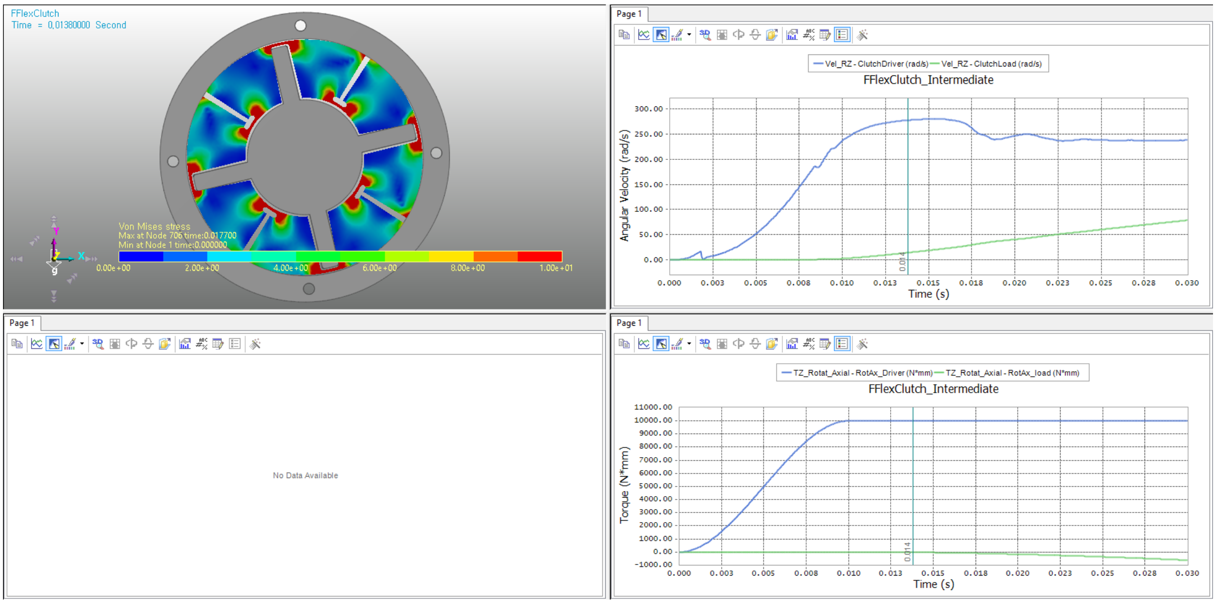
To compare rotational velocity and torque plots to the animation:



1. From the **Plot** group in the **Analysis** tab, click **Plot Result**.

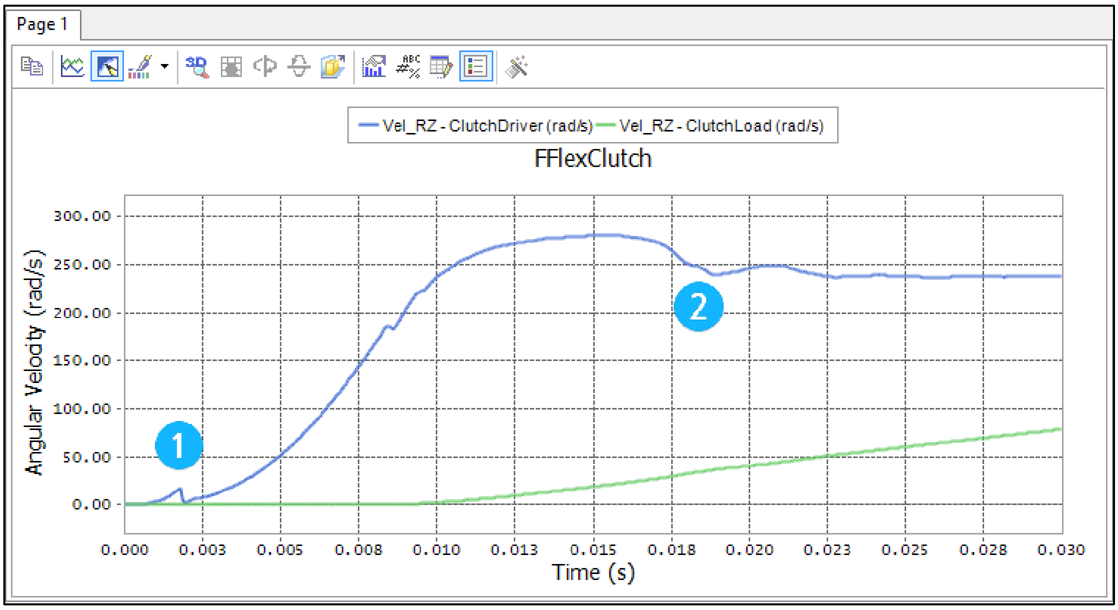
2. From the **Windows** group in the **Home** tab, click **Show All Windows**.
3. In the upper left window, click **Load Animation** from the **Animation** group in the **Tool** tab.
4. In the upper right window, plot the rotational velocities of the clutch driver and load:
   - **FFlexClutch → Bodies → ClutchDriver → Vel_RZ**
   - **FFlexClutch → Bodies → ClutchLoad → Vel_RZ**
5. In the lower right window, plot the torques applied to the clutch driver and load:
   - **FFlexClutch → Force → Rotational Single (Axial) Force → RotAx_driver → TZ_Rotat_Axial**
   - **FFlexClutch → Force → Rotational Single (Axial) Force → RotAx_load → TZ_Rotat_Axial**

   Your Plot windows appear as shown below.



6. Play the animation.

Looking at the plot below, it is apparent that the two velocities will probably converge sometime in the future. What significant events occur at points 1 and 2?



Answers: (1) The clutch driver makes contact with the clutch plate, which initially starts in a floating position, and (2) the clutch plate assumes an asymmetric shape.

You will now plot the contact forces experienced by the driver arm and plate-to-load contacts. Recall that contacts 1 through 8 are the driver arm contacts, and contacts 9 through 12 are the plate-to-load contacts.

To plot the contact forces:

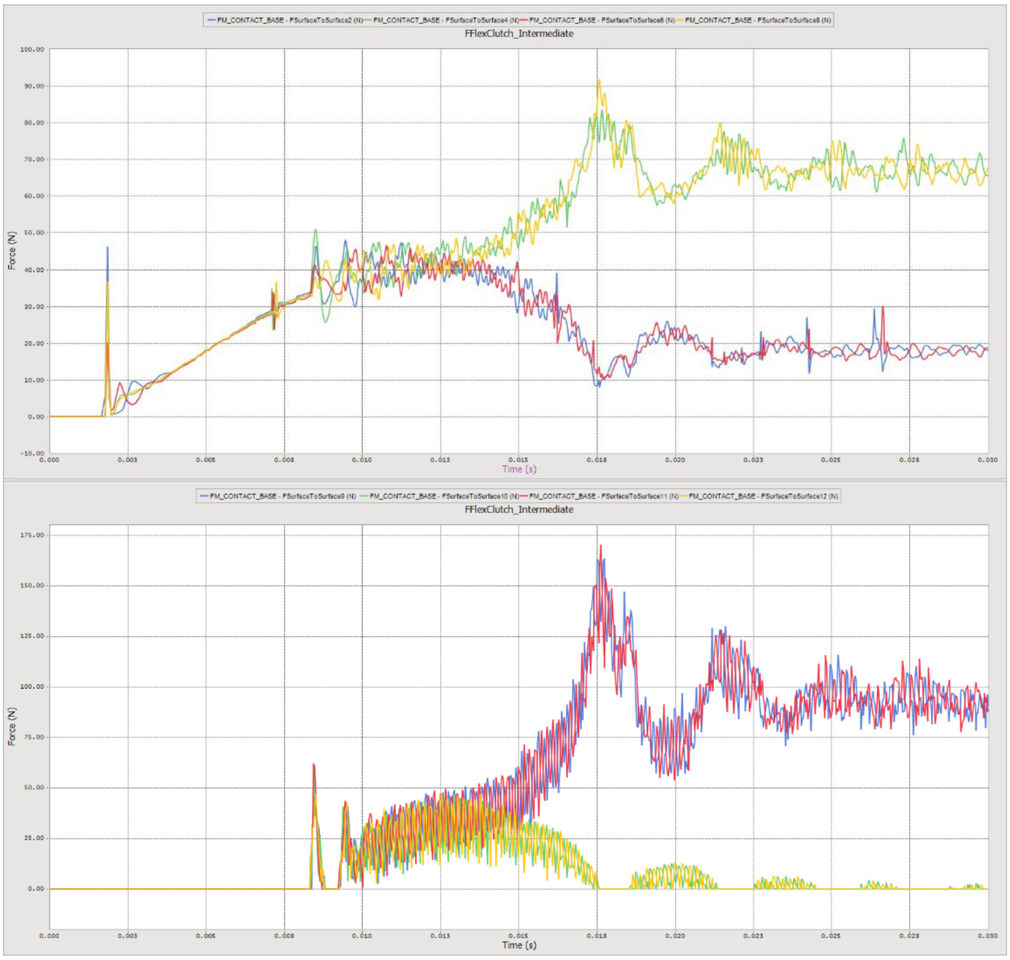
1. In a new Plot window, plot the four driver arm contact forces that experience contact during the simulation.

   These will be for contacts 2, 4, 6, and 8 (**FFlexClutch → Contact → Flexible surface → FSurfaceToSurface2 → FM_CONTACT_BASE**, and so on).

2. In another Plot window, plot the four-clutch plate-to-load contact forces.

   These will be for contacts 9 through 12 (**FFlexClutch → Contact → Flexible surface … → FSurfaceToSurface9 → FM_CONTACT_BASE**, and so on).

   Your plots appear as shown next.



These plots are interesting but can be made easier to interpret by reducing the noise in the data. You will now apply a low-pass filter to the data, using a cut-off frequency of 600 Hz.

Note:
The FM_CONTACT_BASE plot option takes the value of the total force experienced by the contact. This is why the units in the plots above are force, not force/area.

To apply a filter to the contact force data:

1. Select the **Filter** of the **Analysis** group in the **Home** tab.
2. Use the figure below for the next several steps.
3. Change the **Type** to **Low Pass** (click on the field to enable the dropdown button and make the selection from the dropdown list).
4. Set **Cutoff (Hz)** to **600**.
5. Click **Execute**.
6. Set **Curve** to **1: FM_CONTACT_BASE – FsurfaceToSurface2 (N)**, again by activating the dropdown list in the field.
7. Click **Execute**.
8. Repeat the two steps above for the three remaining curves of interest.
9. Click **Close**.

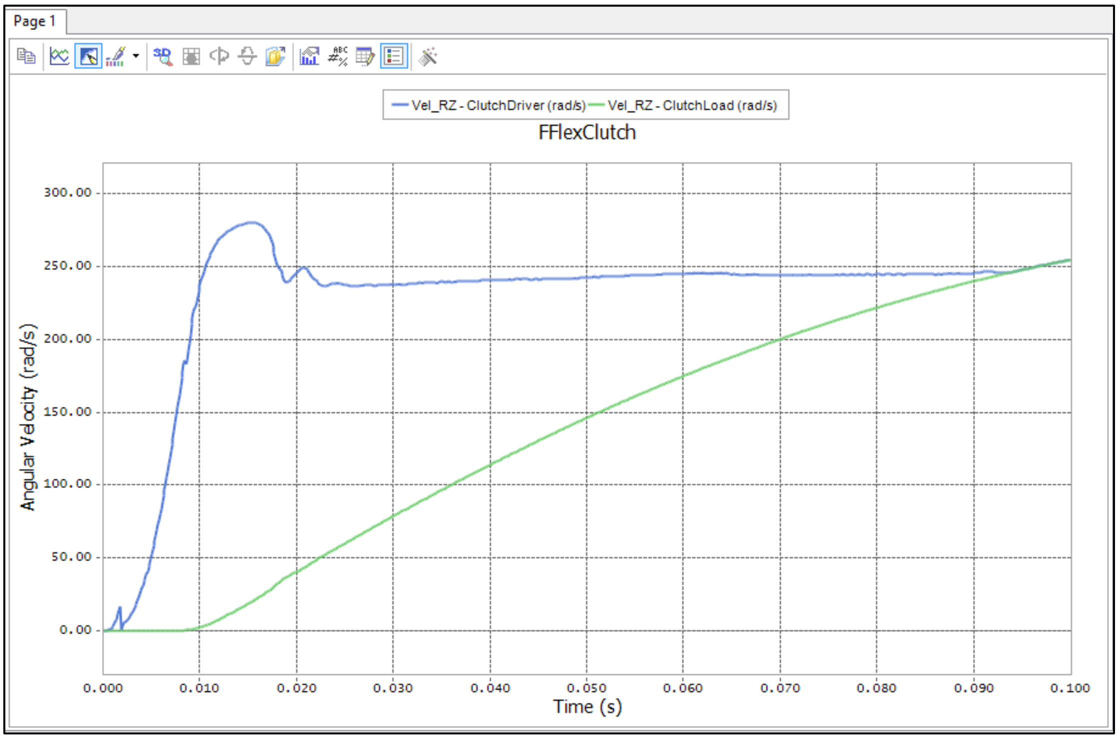
   When finished applying the filters, note that the filter causes a small-time lag.
10. To clean up the plot, delete the unfiltered plot lines and change the new plot colors as desired to make each line more distinct.
11. Repeat the steps above for the other contact force plot.

Your plots should now appear as shown next.

The plots are now much easier to interpret. Looking at the driver arm contacts; all four contacts experience similar forces until contact is made between the plate and load. At this point, contacts 2 and 6 oscillate opposite contacts 4 and 8, with contact 4 and 8 ultimately maintaining higher contact forces. This makes sense because contacts 4 and 8 are on the driver arms whose surrounding plate segments are those that end up making more contact with the load, and, therefore, transfer more of the torque. Interestingly, the divergence of contacts 2/6 from 4/8 occurs at ~0.010 sec., some time before the clutch plate assumes an asymmetrical shape (~0.016 sec.).

Looking at the clutch plate-to-load contacts, contacts 9 and 11 are similar, and 10 and 12 are similar. The 9/11 group and the 10/12 group initially oscillate in the same direction, but as time progresses, the oscillation becomes opposite, and the 9/11 group assumes the higher contact forces. These contacts are on the top and bottom of the clutch plate in its starting position and are the ones that end up having more contact with the load.

Finally, you might wonder how long it takes the clutch to engage. If you were to run the simulation longer, you would obtain the results below, which show that the clutch engages at 0.095 seconds.



Appendix

A

Creating the Remaining Patch Sets

This appendix continues the section, *Creating the Patch Sets*, on page 20, explaining how to create the remaining eight patch sets required for the surface contacts. It provides you with more experience creating patch sets.

Task Objective

You will practice creating patch sets.

Estimated Time to Complete

25 minutes

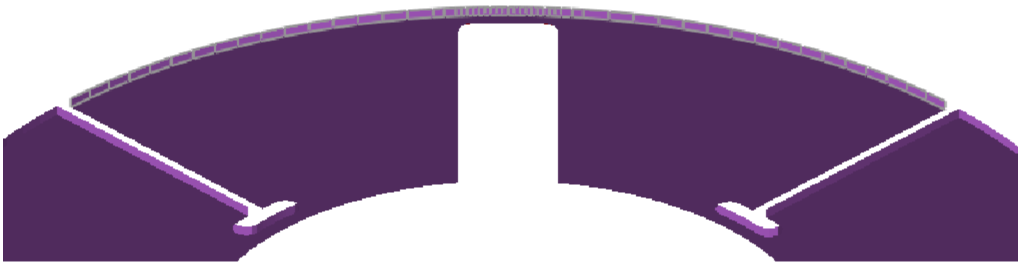
Creating the Remaining Patch Sets

To create the remaining patch sets:

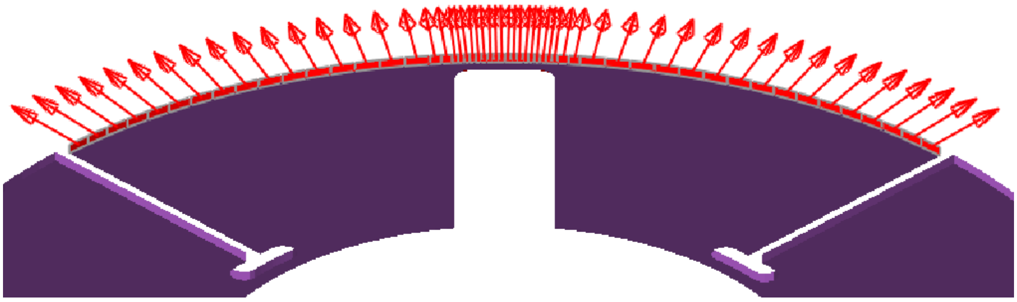
1. Using the same method (**Add/Remove(Continuous)**) used to create the first patch set, select the surfaces the clutch plate as indicated in the contact layout diagram at the beginning of this appendix.

   As you proceed, RecurDyn automatically names the patch sets **SetPatch2**, **SetPatch3,…**, **SetPatch8**.

2. Repeat the same steps for the outer surfaces of the clutch plate, starting with the upper surface and going clockwise around the clutch plate, as indicated in the contact layout diagram. For each surface, select the elements as shown below, highlighted in light gray:



   The normals should all point outwards towards the clutch load, as shown below:



3. Click the **Exit** button to return to the root model.
4. Continue with the section, *Creating the Face Surfaces*, on page 23.

Thanks for participating in this tutorial!