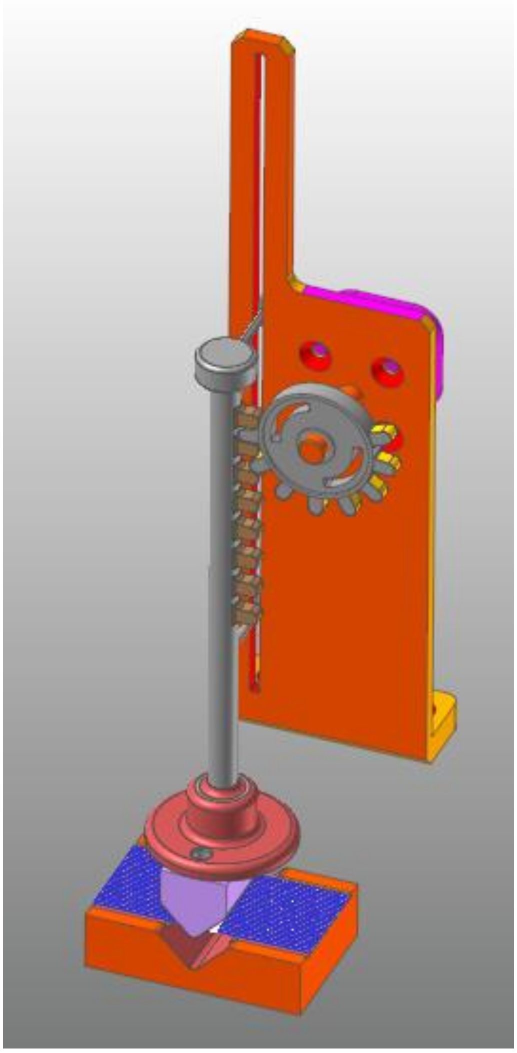
Plasticity Bending Machine Tutorial (FFlex)



Edition Note

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

Table of Contents

Chapter 1

Overview

A bending machine bends metal plates by holding them between its upper and lower frames and applying pressure to the plate. Different bending frames bend plates into different angles and shapes.

The shapes of the frames determine how a bending machine deforms the metal plate. The force applied by the bending machine creates a stress that exceeds the yield stress of the plate, so the plate is permanently deformed (plastic deformation). Therefore, to understand the plastic behavior of a metal plate, you must perform a plastic analysis rather than an elastic analysis.

To perform a plastic analysis, you must give a flexible body modeled in RecurDyn with plastic properties instead of elastic properties and simulate the behavior of the flexible body.

In this tutorial, you will learn how to assign plastic properties to a dynamic model composed of flexible bodies and perform a plastic analysis. You will also learn about the characteristics of a plastic analysis by comparing the results of the analysis to the results obtained using an elastic material.

Task Objective

This tutorial covers the following topics:

- Elastic analysis of the behavior of a flexible body
- Requirements for performing a plastic analysis
- How to apply plastic properties to an analysis model and the characteristics of plastic analysis
- How to analyze the results of a plastic analysis
- Differences between an elastic analysis and a plastic analysis

Prerequisites

This tutorial is intended for users who have completed the Basic and FFlex/RFlex tutorials provided with RecurDyn. If you have not completed these tutorials, then you should complete them before proceeding with this tutorial. In addition, this tutorial requires a basic understanding of dynamics and the finite element method.

Tasks

The following table outlines the tasks involved in this tutorial and their duration.

Procedure	Time (minutes)
Opening the Initial Model	5
Generating a FFlex Body	20
Performing an Elastic Analysis	10
Performing a Plastic Analysis	5
Analyzing and Reviewing the Results	10
Total	65

Estimated Time to Complete

This tutorial takes approximately 65 minutes to complete.

Chapter

2

Opening the Initial Model

Task Objective

Open the initial model, perform a simulation, and observe the behavior of the bending machine.

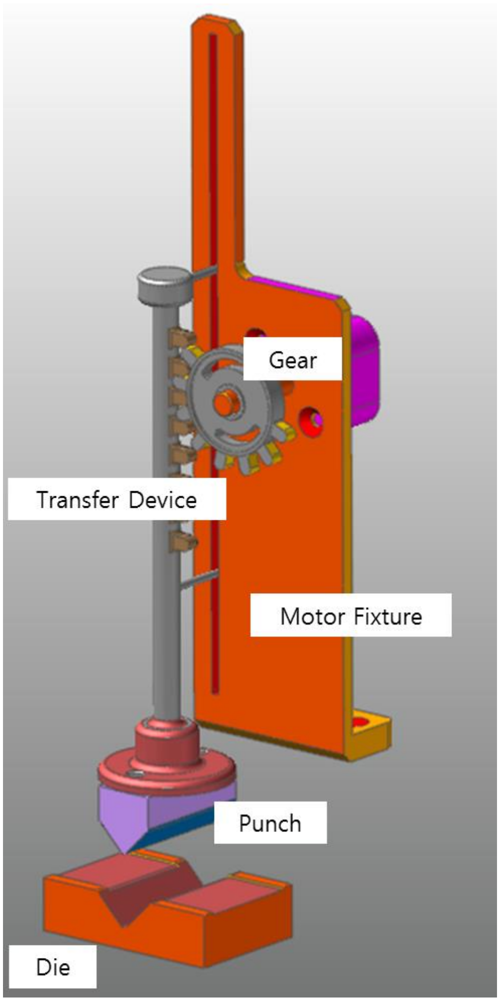
Estimated Time to Complete

5 minutes

Opening the RecurDyn Model

To run RecurDyn and open the initial model:

1. On the Desktop, double-click the **RecurDyn** icon to open RecurDyn.
2. When the **Start RecurDyn** dialog window appears, close it.
3. In the **File** menu, click **Open**.
4. Navigate to the **Plasticity** tutorial folder and select **Plasticity_Bending_Machine_Start.rdyn.** (File path: <Install Dir>\Help\Tutorial\Flexible\FFlex\Plasticity_Bending_Machine).
5. Click the **Open** button to open the model shown in the following figure.



The following explains the configuration of the model.

The figure on the right shows a type of bending machine. The motor turns the gear to raise the transfer device. When the transfer device reaches the final tooth on the gear, the transfer device drops to strike the metal plate (placed on the die) with the punch.

The figure on the right does not include the metal plate. Later in this tutorial, you will learn how to model a metal plate using a flexible body. To do so, you model a metal plate with plastic properties on top of the die. The energy of the falling punch and transfer device is applied to the metal plate through an impact to deform it.

When the punch hits it, the metal plate deforms in the shape of the die. The teeth on the gear then engages the teeth on the transfer device to raise the punch again.

The model used in this tutorial consists of a die, punch, transfer device, motor fixture, and gear. You will model the metal plate later in the tutorial.

To save the model:

1. In the **File** menu, click **Save As.**

   (You cannot perform the simulation if the model is in the tutorial path, so you must save the model in a different path.)

Simulating the Initial Suspension Model

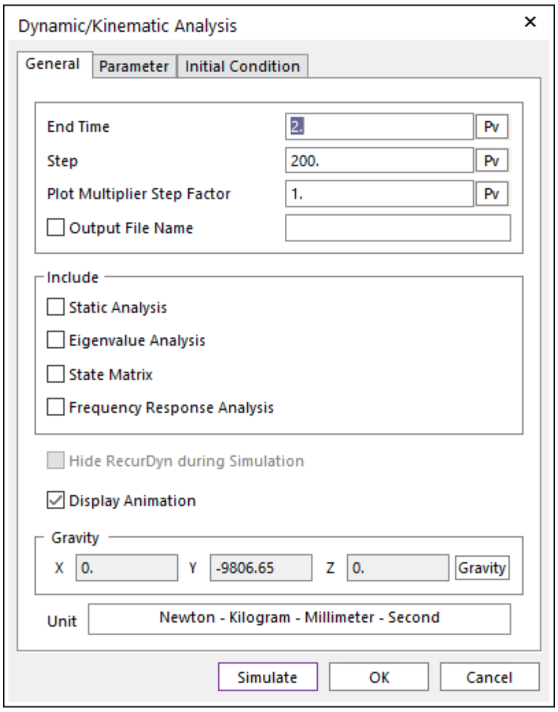
This section teaches you how to conduct an initial simulation using the model in order to understand its behavior.

To perform the initial simulation:

1. On the **Analysis** tab, in the **Simulation Type** group, click the **Dyn/Kin** icon.

   The Dynamic/Kinematic Analysis dialog window appears.

2. Set the **End Time** and **Step** fields as follows:
   - **End Time**: 2
   - **Step**: 200
   - **Plot Multiplier Step Factor**: 1
3. Click **Simulate**.



Viewing the Result

To view the results:

On the **Analysis** tab, in the **Animation Control** group, click the **Play** button.

In the work pane, the motor turns the gear to raise the transfer device. After the transfer device passes the final tooth on the gear, the punch falls and strikes the die. The gear then engages the transfer device again to raise it.

Chapter

3

Generating a FFlex Body

As mentioned previously, the current RecurDyn model does not include the metal plate model needed to perform plastic analysis. Therefore, to create the flexible body for the metal plate, you must first create a rigid body using the box geometry provided in RecurDyn. Then, you can transform the rigid body into a flexible body using the Mesher. You must also define the contact points between the metal plate and die as well as the metal plate and punch.

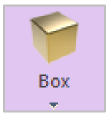
Task Objectives

This task teaches you how to replace the newly created rigid body with a flexible body using the Mesher provided in **RecurDyn/FFlex** (Full Flex).

Estimated Time to Complete

20 minutes

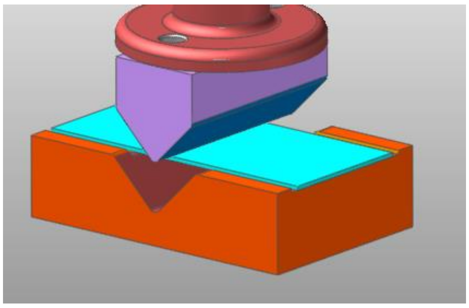
Creating the Box Geometry

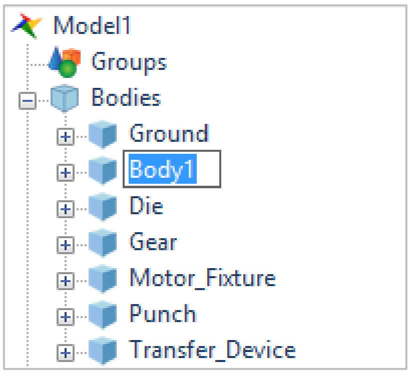


1. On the **Professional** tab, in the **Body** group, click the **Box** icon.
2. In the **Modeling Options** dropdown menu, select **Point, Point,** and then type the following values in the **Command Input** field.

   -75, 0, 35 (Press Enter after typing the values.)

   75, -2, -35 (Press Enter after typing the values.)
3. After you enter the values, the box geometry (shown below) appears. In the **Database** pane, change the name of **Body1** to **Plate.**

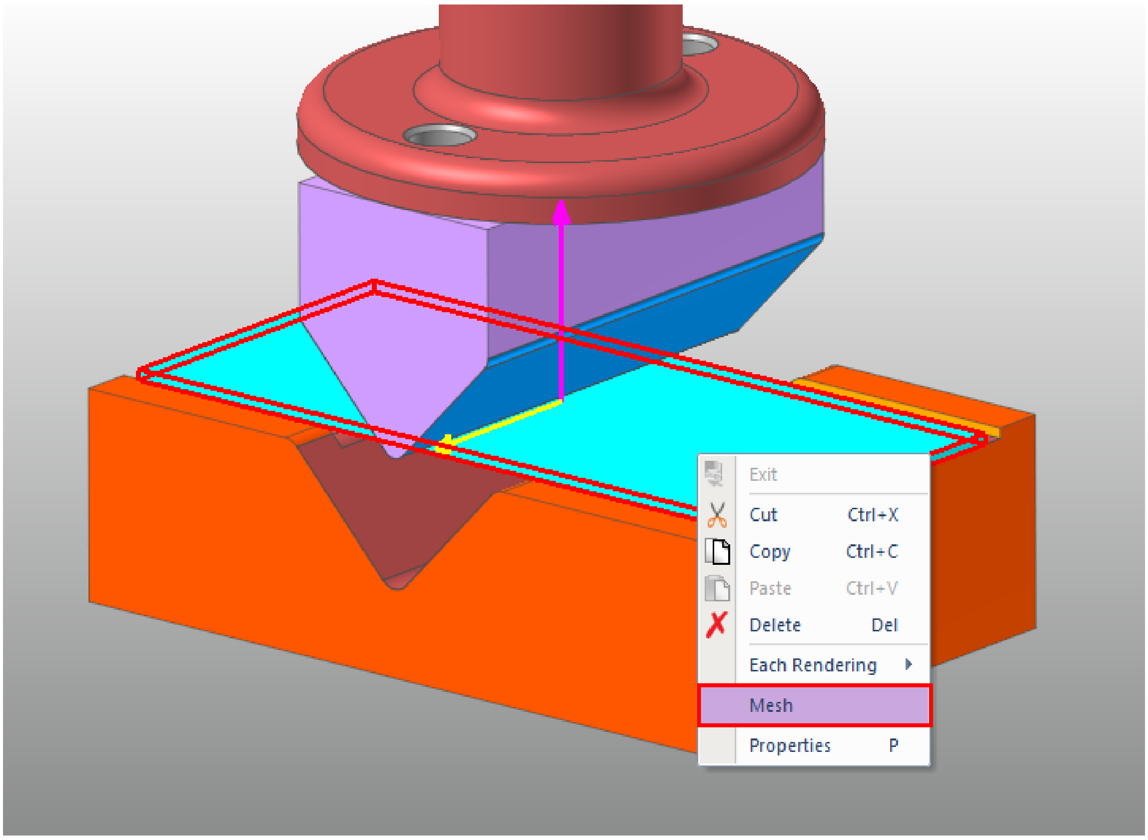


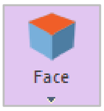


Creating the Box Mesh

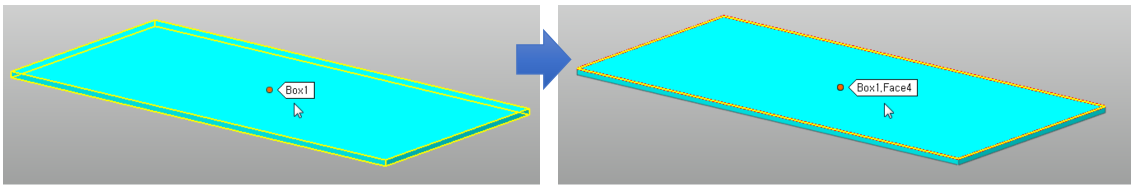
1. In **Assembly** mode, right-click the work pane to display the context menu and click **Mesh**.

   Mesh mode displays only the **plate body**, as shown below.

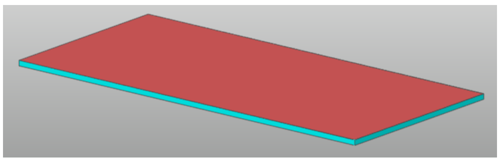


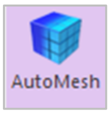


2. On the **Geometry** tab, in the **Surface** group, select **Face Surface**.
3. Select **Box1** to open the **FaceSurf Operation** dialog window. With the **FaceSurf Operation** dialog window open, select **Box1.Face4** on top of the box.

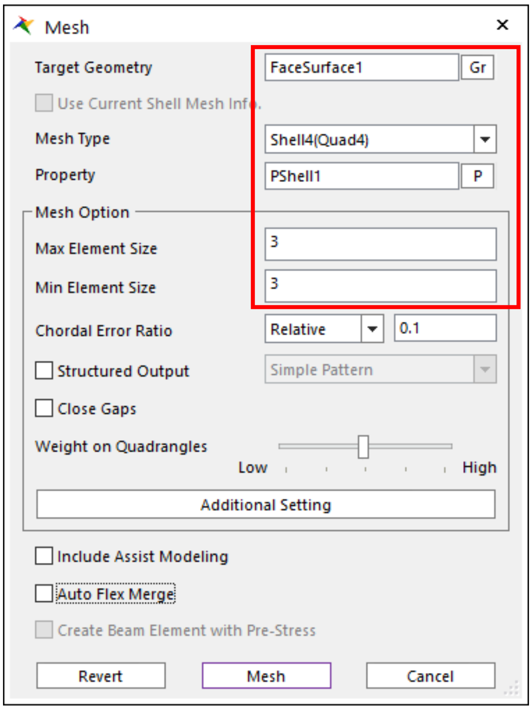


4. When **Face4** appears in the dialog window (shown below), click the OK button to create **FaceSurface1**.

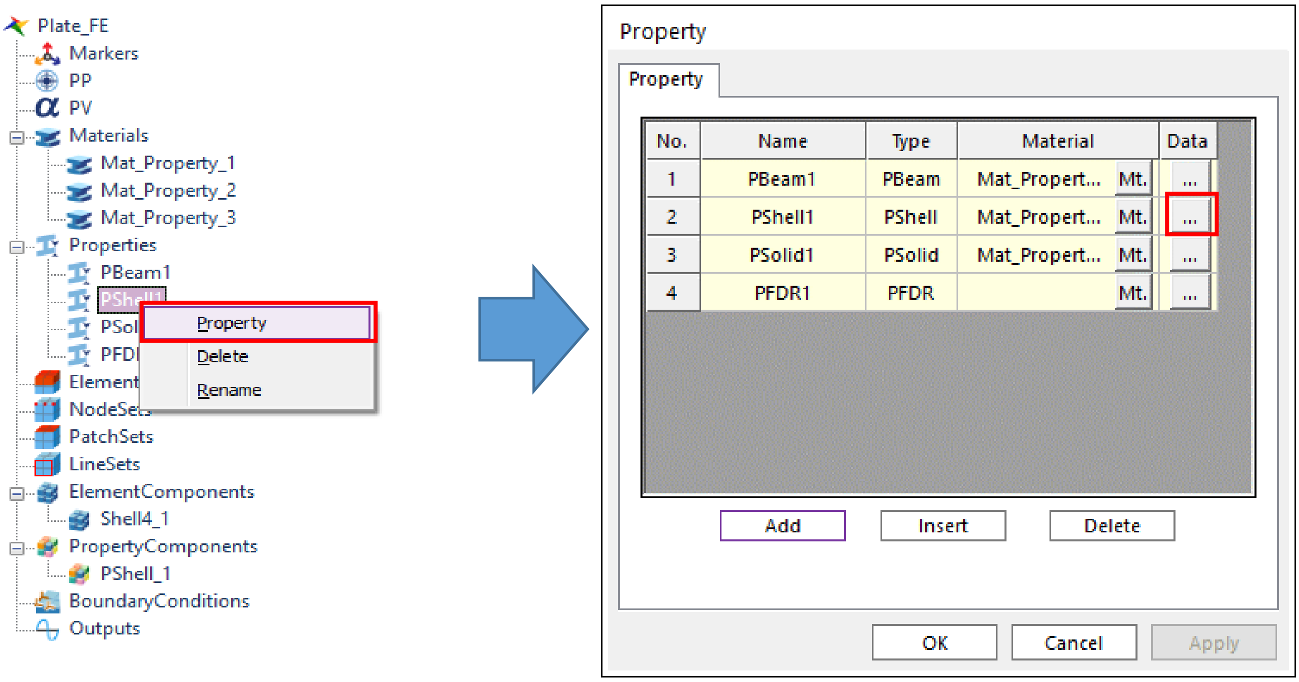




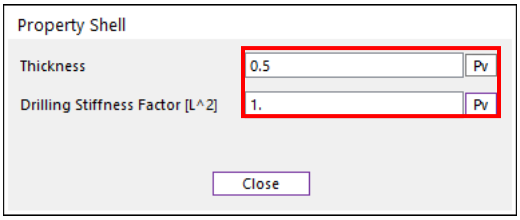
5. On the **Mesher** tab, click the **Mesh** icon to open the Mesh dialog window.
6. In the **Mesh dialog window**, perform the following:
   - In the **Target Geometry** option, Click the **Gr** button then select **FaceSurface1**.
   - In the **Mesh Type** dropdown menu, select **Shell4(Quad4)**.
   - In the Mesh Option pane, set both the Max Element Size and Min Element Size to 3.
   - Click the **Mesh** button.
   - Click the **Cancel** button.



7. In the **Database** pane, under the newly created **Plate_FE**, right-click **PShell1**, and then click **Property** in the context menu to open the **Property** dialog window.

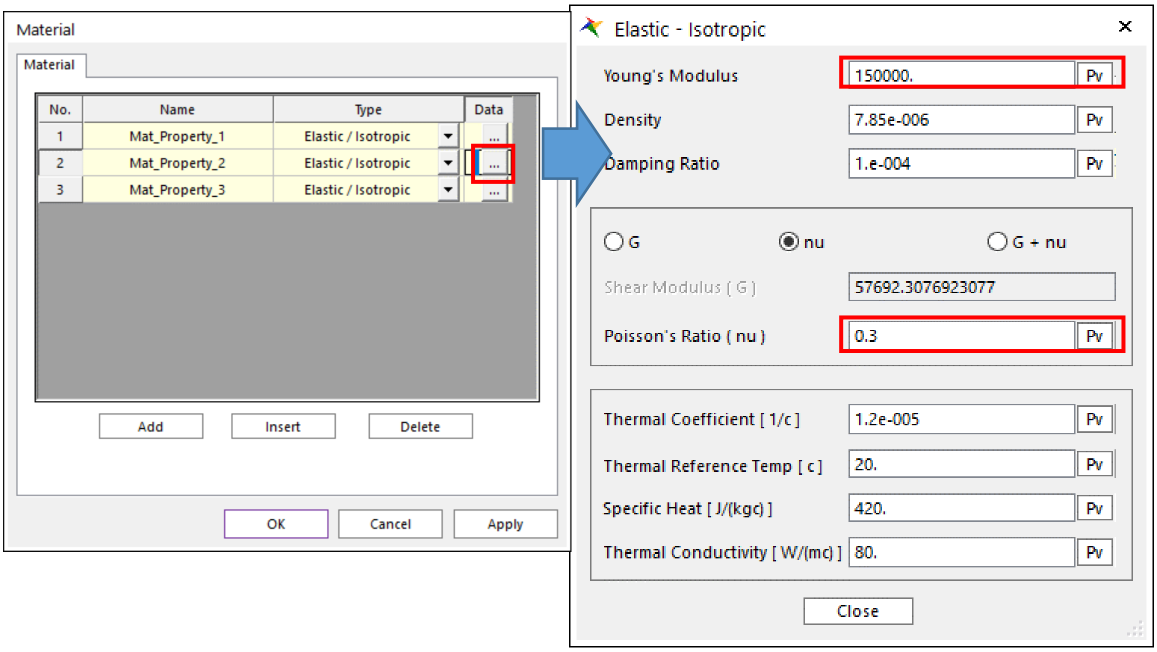


8. In the **Property** dialog window, click the **...** button to the right of No. 2 **PShell1**. When the **Property Shell** dialog window appears, change the **Thickness** from 10 to **0.5**, and then click the Close button.

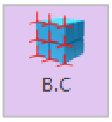


9. In the **Property** dialog window, click the **Mt.** button to the right of No. 2 PShell1. The **Material** dialog window appears (shown below).

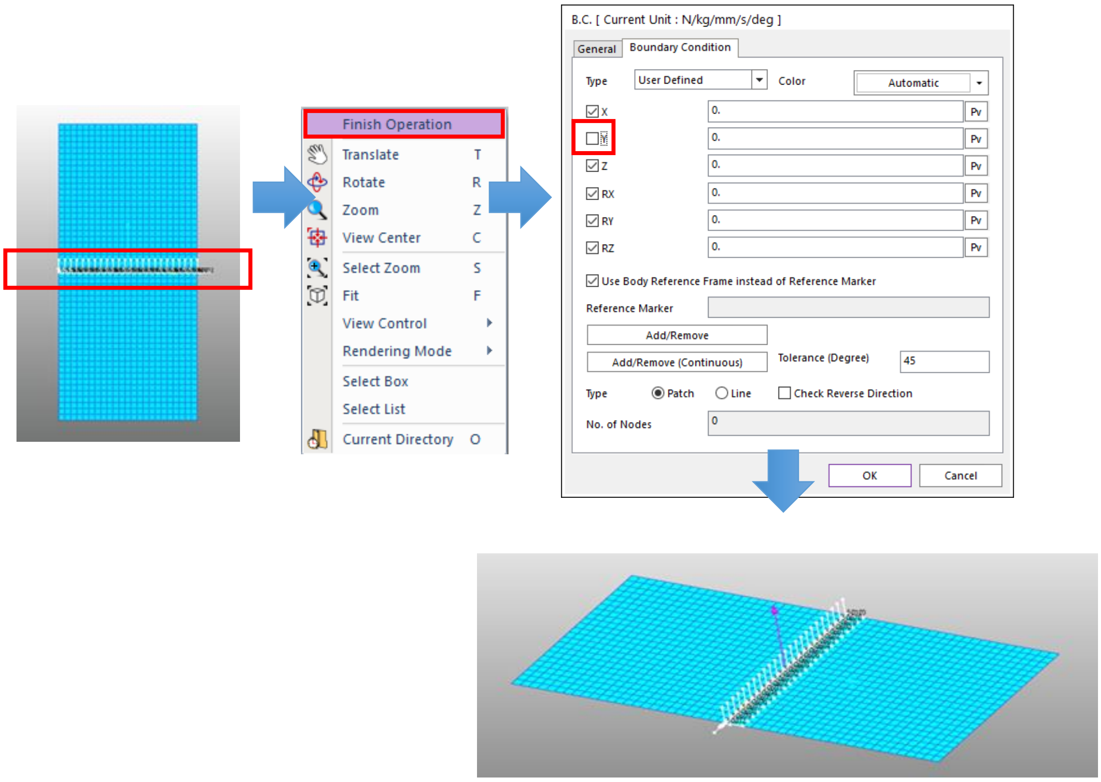
   - In the **Material** dialog window, click the **...** button to the right of No. 2, and then change the following settings:



- Change the **Young's Modulus** value from 200,000 to **150,000**.
- Select **nu**, and then change **Poisson's Ratio** to **0.3**. Then, click the Close button.
- Click the **OK** button twice to close all open dialog windows.



10. In the **FFlex Edit** group, click the **B.C** icon. When the B.C. Current Unit dialog window appears, click the **Add/Remove** button.
    - Press **Shift** and **Z** at the same time to change the working plane to the **ZX** plane. Then, click and drag the mouse to select the nodes located at the center of the plane.



- Once you select the nodes on which to apply the B.C., right-click the screen and select **Finish Operation from the context menu**. (For your information, the ID of the leftmost node is 50103.)
- When the B.C. Property dialog window appears again, clear the **Y** checkbox, as shown below. Click the **OK** button to see the new **boundary conditions** created on the screen.

11. Use one of the following methods to return to a higher menu:
    - Right-click the working pane to display the context menu, and then click **Exit**.
    - On the **Mesher** tab, in the **Exit** group, click the **Exit icon**.

Tip: Setting the boundary conditions for the FFlex Body as such removes the constraint on the Y axis (the direction in which the punch travels) but maintains the constraint for the remaining DOFs in the center of the plate, where deformation is not expected to occur, in order to increase the speed of analysis.

Chapter

4

Performing Elastic Analysis

Task Objectives

In this chapter, you will learn how to conduct dynamic modeling and analysis on a FFlex Body and check the results of the analysis.

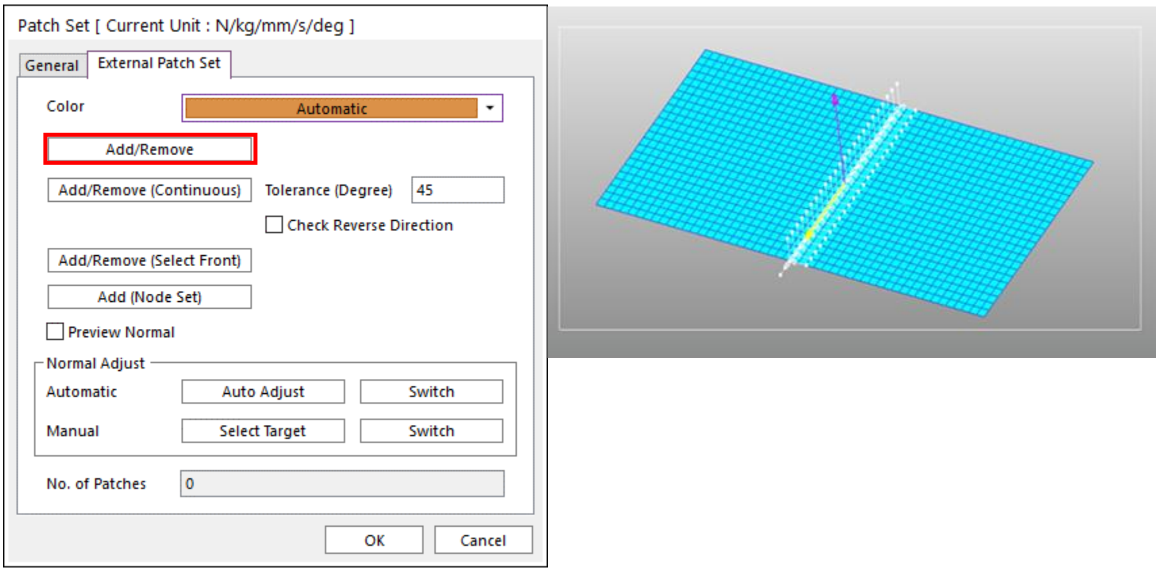
Estimated Time to Complete

10 minutes

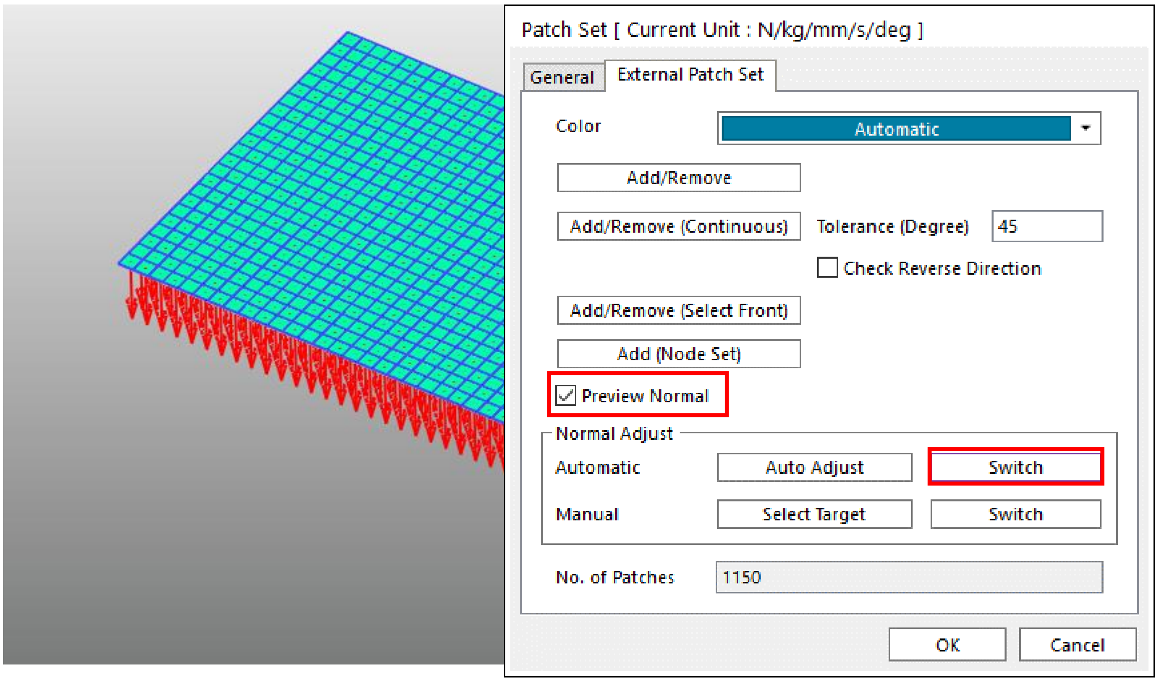
Performing Dynamic Modeling

To create a patch set:

1. Double-click the **Plate_FE** body to enter **Body Edit** mode.
2. On the **FFlex Edit** tab, in the **Set** group, click the **Patch Set** icon.
3. When the Patch Set dialog window appears, click the **Add/Remove** button. Then, in the work pane, click and drag the mouse to select the whole body.



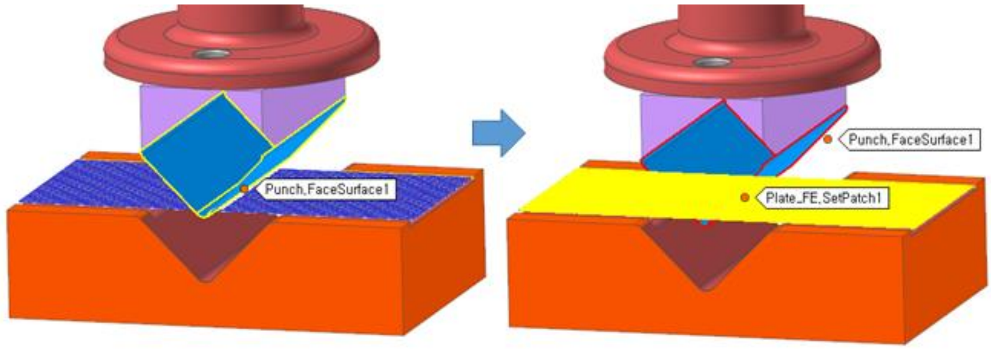
4. Right-click the work pane to display the context menu, and then click **Finish Operation**.
5. In the Patch Set dialog window, click the **OK** button.
   - Confirm that **SetPatch1** appears in the **Database** pane.
6. Click the **Patch Set** icon again and repeat steps 3 and 4 to create another patch set. In the **Patch Set** dialog window, select the **Preview Normal** checkbox, as shown below. Then, in the Automatic row of the Normal Adjust pane, click the **Switch** button. (This process creates two patch sets to define the contacts on both sides of the shell.)



7. In the **Patch Set** dialog window, click the **OK** button.
   - Confirm that **SetPatch2** appears in the **Database** pane.
8. On the **FFlex Edit** tab, in the **Exit** group, click the **Exit icon** to return to a higher menu.

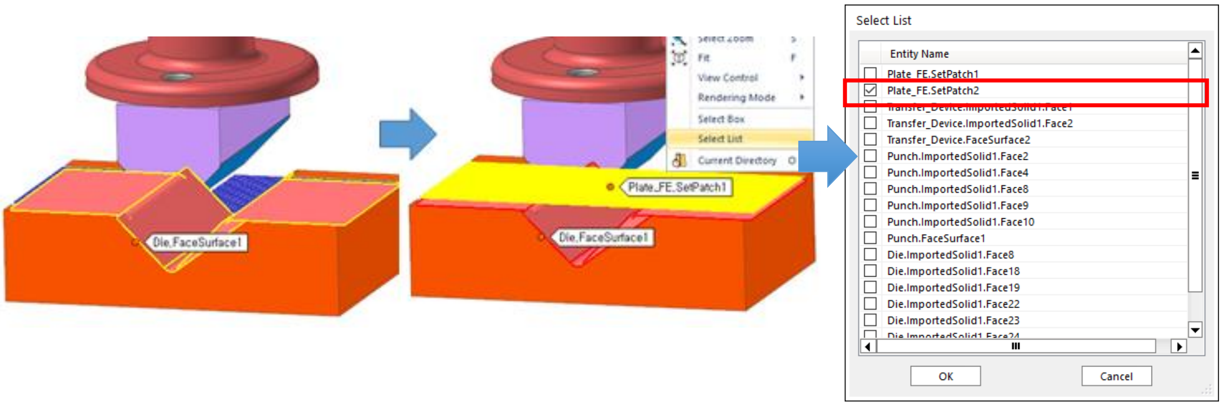
To create a contact:

1. On the Professional tab, in the Contact group, click the Geo Surface Contact icon. Then, in the Modeling Option dropdown menu, select **Surface(PatchSet)**, **Surface(PatchSet)**, as shown below.

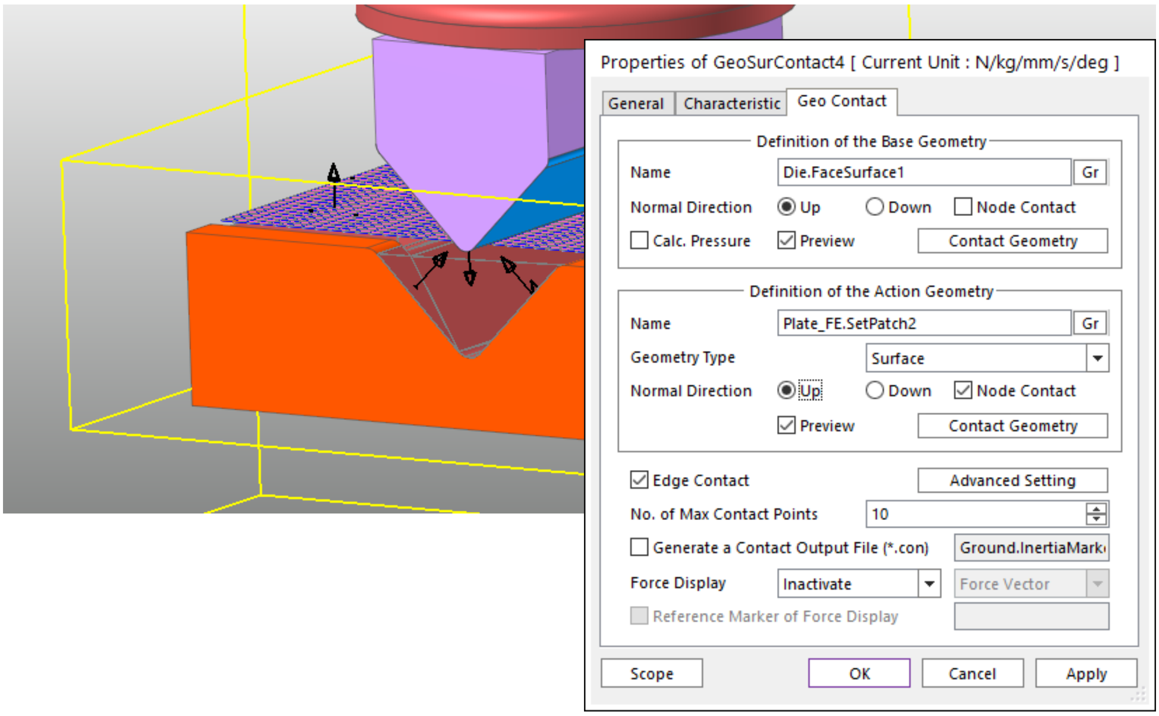


2. Select **Punch.FaceSurface1**, which will be designated as the base body for the geo surface contact, and **Plate_FE.SetPatch1**, which will be designated as the action body. Confirm that **GeoSurContact3** is created in the Database pane.
3. Click the Geo Surface Contact icon again. Then, select the areas that will become the base and action bodies for the geo surface contact, as shown below, to create **GeoSurfaceContact4**.
   - For the base body, select **Die.FaceSurface1**.

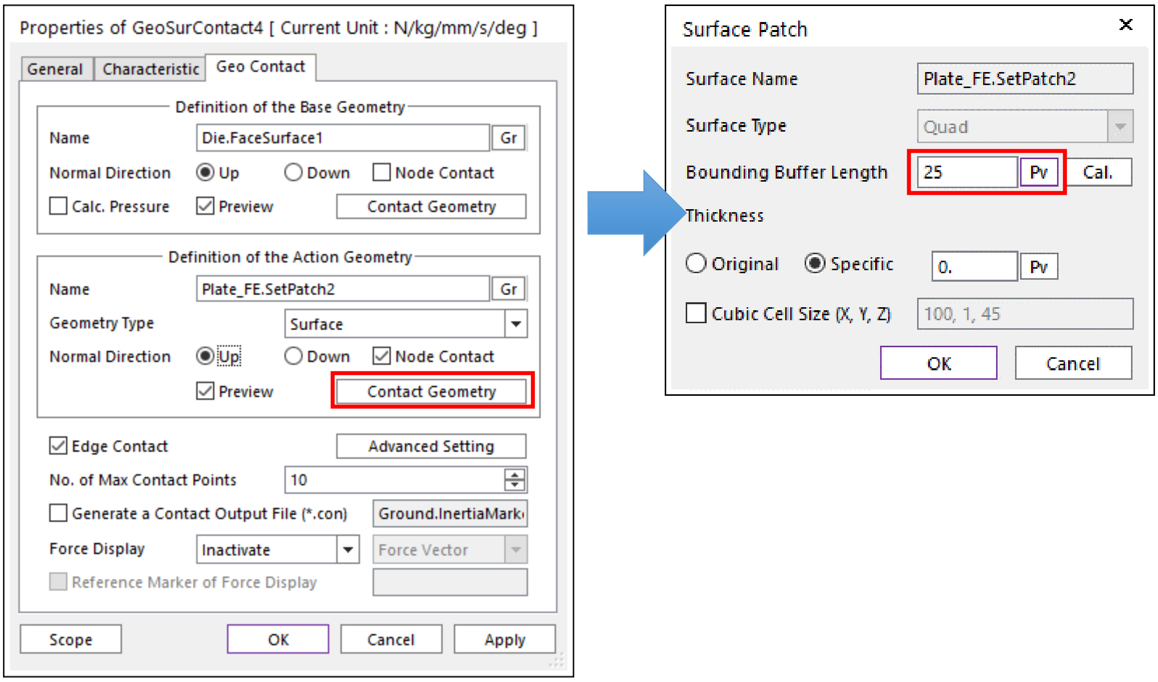
- For the action body, it is difficult to select **Plate_FE.SetPatch2** in the work pane. So, right-click the screen to display the context menu, and then click Select List. When the Select List dialog window appears, select the **Plate_FE.SetPatch2** checkbox.



4. To see if the geo surface contacts are set properly, right-click **GeoSurfaceContact4** to open the Property dialog window.



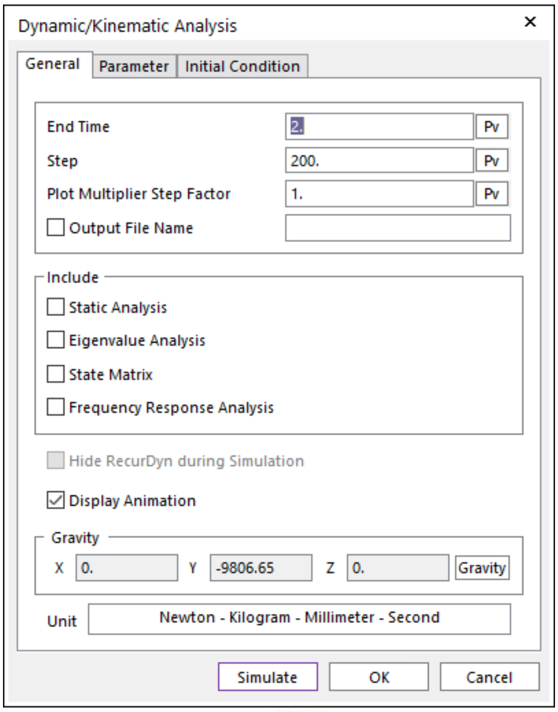
5. In the Definition of the Action Geometry pane, click the **Contact Geometry** button. When the Surface Patch dialog window appears, change the **Bounding Buffer Length** to **25.**



6. Perform the same process for **GeoSurContact3**, but leave the Normal Direction set to Up. Only change the **Bounding Buffer Length** for **Plate_FE.SetPatch1** to **25**.

Performing Elastic Analysis

In this procedure, you will learn how to perform a simulation on the FFlex body created earlier and the contacts defined for elastic analysis.



To run the simulation:

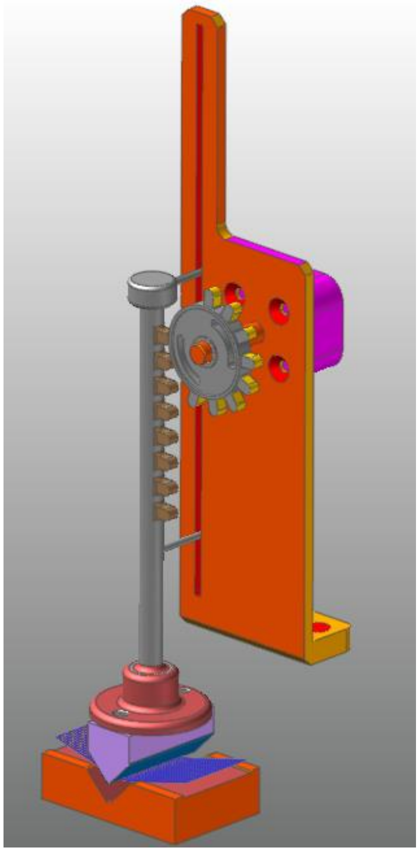
1. On the **Analysis** tab, in the **Simulation Type** group, click the **Dyn/Kin** icon.

   The Dynamic/Kinematic Analysis dialog window appears.

2. Set the **End Time** and **Step** fields as follows:
   - **End Time**: 2
   - **Step**: 200
   - **Plot Multiplier Step Factor**: 1
3. Click **Simulate**.

To view the results:

On the **Analysis** tab, in the **Animation Control** group, click the **Play** button.



Unlike the transfer device in previous simulation, in which the punch came into direct contact with the die, the transfer device in this simulation behaves differently because the contact occurs between the Plate_FE flexible body and the punch and die.

The Plate_FE flexible body acts as a spring when the punch falls on it, and the transfer device rebound upwards. This phenomenon does not occur in real life. The reason why this phenomenon occurs is because the Plate_FE flexible body has only elasticity, not plasticity. Therefore, even though a large amount of deformation occurs, the flexible body return to its original state, springing the transfer device upwards.

Due to this spring effect, the transfer device is raised without the help of the gear and must remain aloft until the final gear tooth.

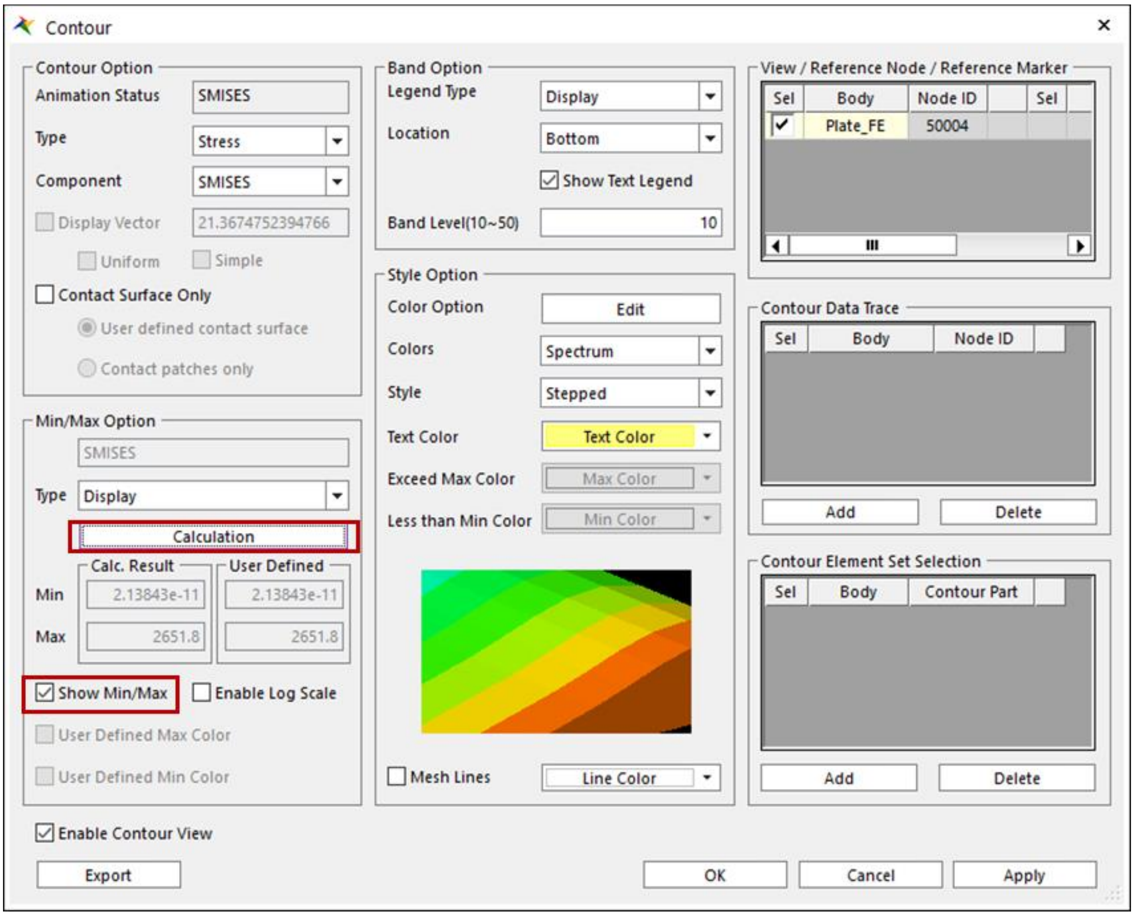
Therefore, to realistically simulate the deformation of a metal plate and the operation of a transfer device, you must add plasticity to the flexible body and perform plastic analysis rather than elastic analysis.

To check the contour results:

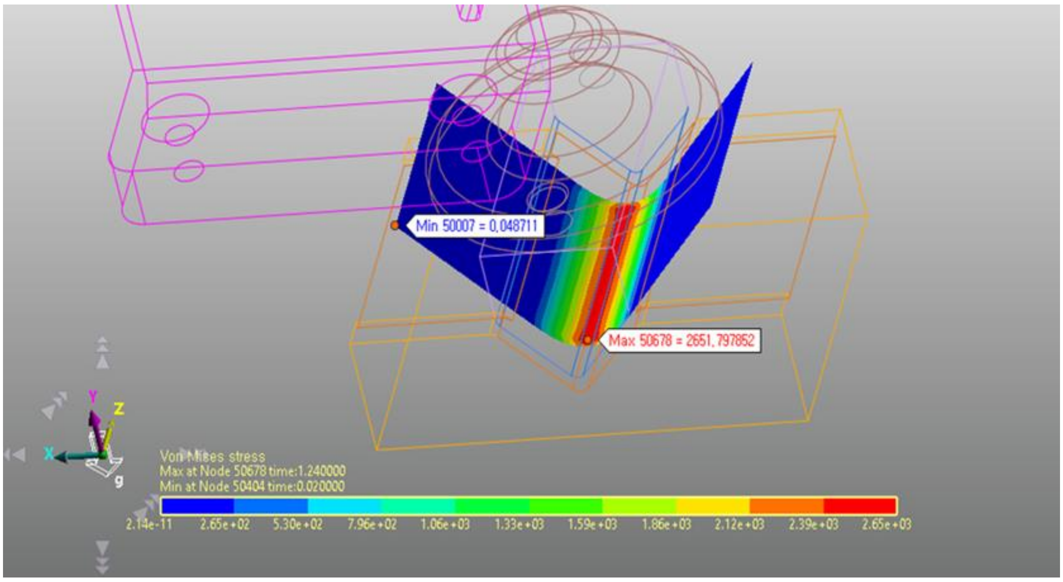
1. On the **Flexible** tab, in the **FFlex** group, click the **Contour icon**.

   The **Contour** dialog window appears.

2. In the **Contour dialog window**, perform the following:
   - Click the **Calculation** button in the middle of the dialog window.
   - Select the **Show Min/Max** checkbox.
   - Click the **OK** button to see the results.



3. On the **Analysis** tab, in the **Animation Control** group, click the **Play** button.
   - To see the contour result for **Plate_FE**, as shown below, click the **Wireframe** icon in the **Toolbar** and run the animation.
   - At **1.24 seconds**, you can see that the Maximum Von-Mises Stress is approximately **2652 MPa.**


Min 50007 = 0,048711
Max 50678 = 2651,797852
Y
Z
X
g
Von Mises stress
Max at Node 50678 time:1.240000
Min at Node 50404 time:0.020000
2.14e-11
2.65e+02
5.30e+02
7.96e+02
1.06e+03
1.33e+03
1.59e+03
1.86e+03
2.12e+03
2.39e+03
2.65e+03

Chapter

5

Performing Plastic Analysis

Task Objectives

In this chapter, you will learn how to apply plastic properties to an FFlex body and check the results after performing plastic analysis.

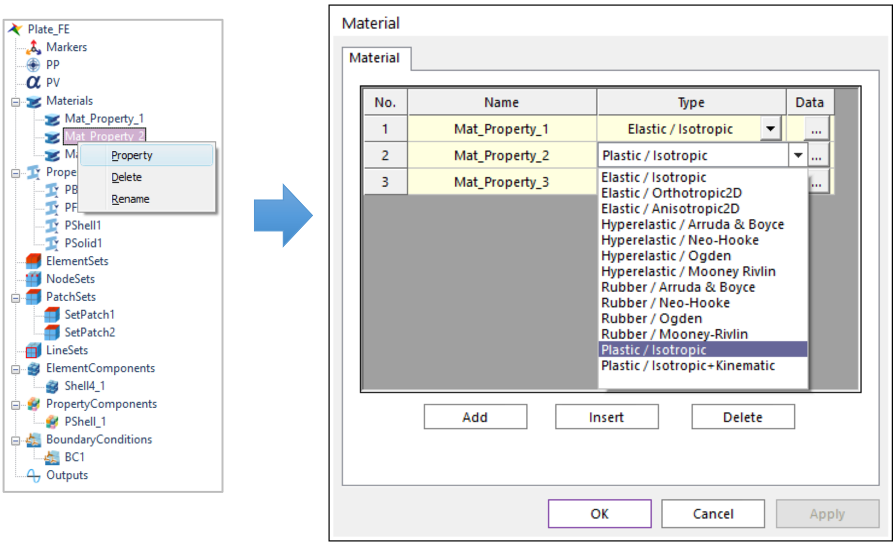
Estimated Time to Complete

20 minutes

Performing Plastic Modeling (1)

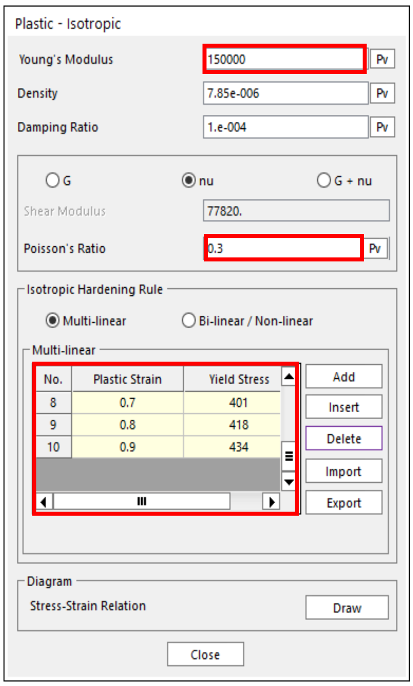
To create a plastic material:

1. Double-click the **Plate_FE** body to enter **Body Edit** mode.
2. In the Database pane, right-click Plate_FE, and then click Edit in the context
3. menu. Then, double-click Mat_Property_2 to open the **Material Property** dialog window, as shown below.

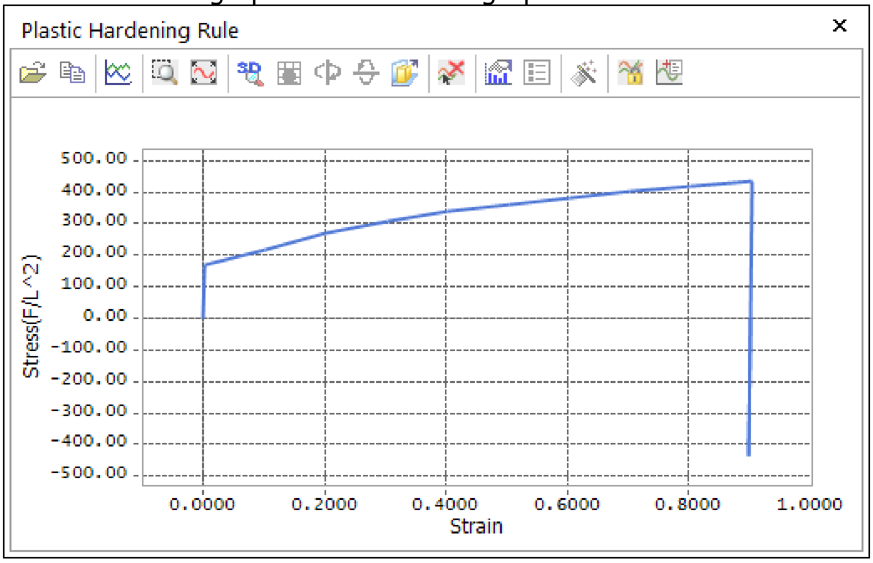


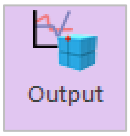
4. In the **Material** dialog window, select **Plastic / Isotropic** in the dropdown menu to the right of **Mat_Property_2** (No. 2). Then, click the **...** button to display the **Plastic – Isotropic** dialog window.
5. In the **Plastic – Isotropic** dialog window, perform the following:
   - Change the **Young's Modulus** value from 200,000 to **150,000**.
   - Select the **Nu** radio button and change **Poisson's Ratio** to **0.3**.
   - In the **Multi-linear** pane, click the **Add** button to add a new row. Then, type in the **Plastic Strain** and **Yield Stress** values shown in the following table.

No.	Plastic Strain	Yield Stress
1	0	168
2	0.1	219
3	0.2	272
4	0.3	308
5	0.4	337
6	0.5	361
7	0.6	382
8	0.7	401
9	0.8	418
10	0.9	434



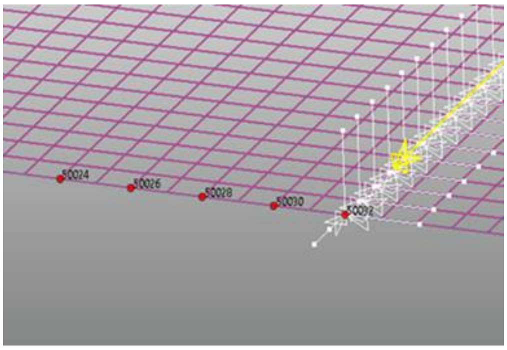
6. To ensure that the values entered in the previous step are correct, in the **Diagram** pane, click the **Draw** button to the right of the **Stress-Strain Relation** menu and ensure that the graph resembles the graph shown below.

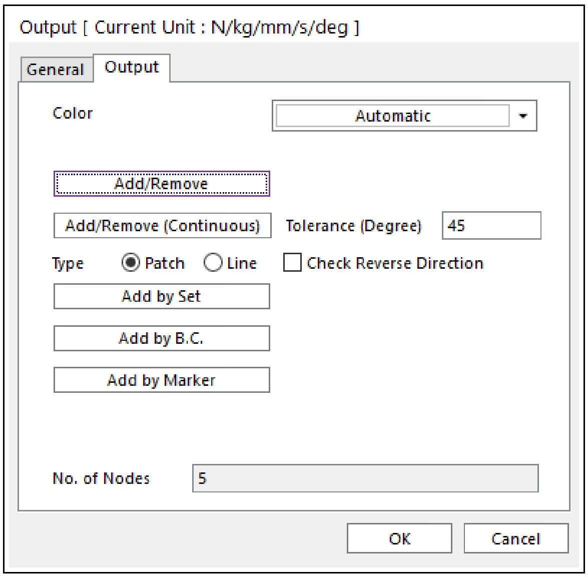




7. In the **FFlex Edit** group, click the **Output** icon. When the Output Current Unit dialog window appears, click the **Add/Remove** button.
   - Hold down the **Shift** key and click the five nodes (50032, 50030, 50028, 50026, and 50024), as shown below.
   - After selecting the nodes, right-click the work pane to display the context menu, and then click **Finish Operation**.

8. In the **Outputs** dialog window, click the **OK** button.





9. On the **FFlex Edit** tab, in the **Exit** group, click the **Exit** icon to return to a higher menu.

10. In the **File** menu, click **Save As**. Then, save the model as **Plasticity_Bending_Machine_Isotropic.rdyn**.

Performing Plastic Analysis (1)

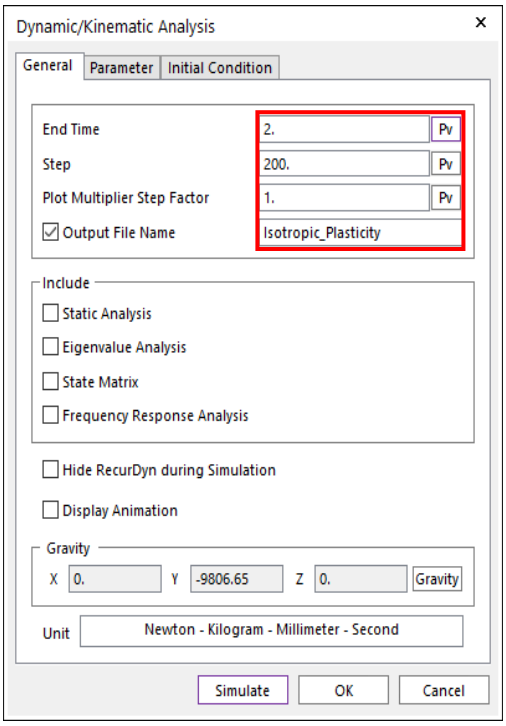
To see the results of the plastic analysis for the plastic material, you must run the simulation.

To run the simulation:

1. On the **Analysis** tab, in the **Simulation Type** group, click the **Dyn/Kin** icon.

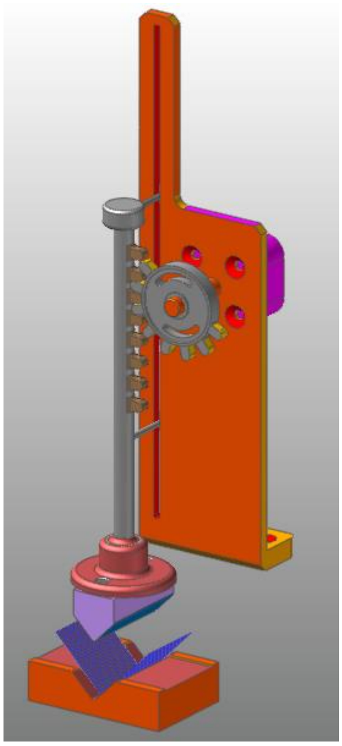
   The **Dynamic/Kinematic** Analysis dialog window appears.

2. Set the **End Time** and **Step** fields as follows:
   - **End Time**: 2
   - **Step**: 200
   - **Plot Multiplier Step Factor**: 1
   - **Output File Name**: Isotropic_Plasticity
3. Click **Simulate**.



To view the results:

On the **Analysis** tab, in the **Animation Control** group, click the **Play** button.

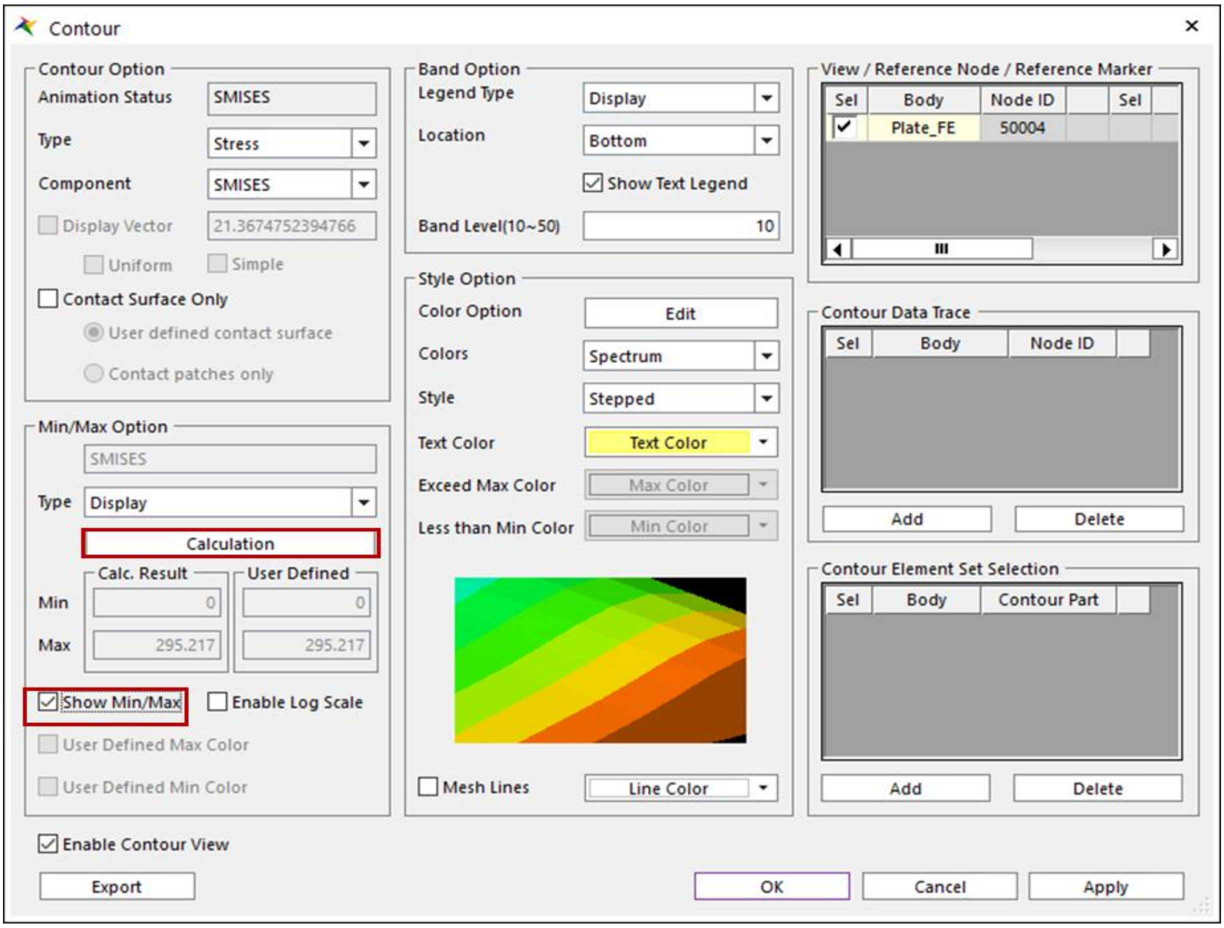


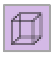
Unlike the results of the elastic analysis, the metal plate in the plastic analysis remains in the V-shape of the die after the punch strikes.

In addition, the spring effect from the elasticity has disappeared, so the transfer device does not rebound. Because of this, the transfer device is raised again when the gear engages the teeth on the transfer device again, and the process is repeated.

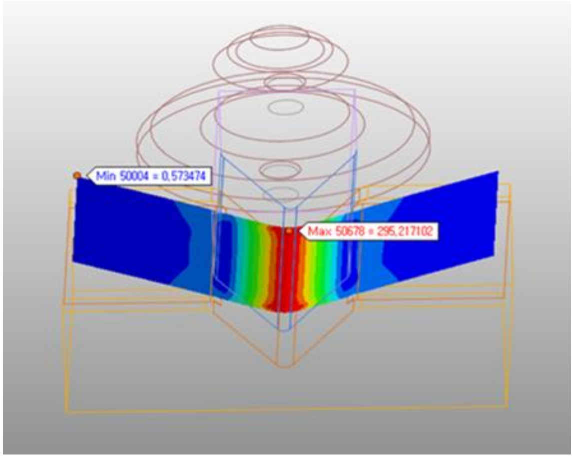
To check the contour results:

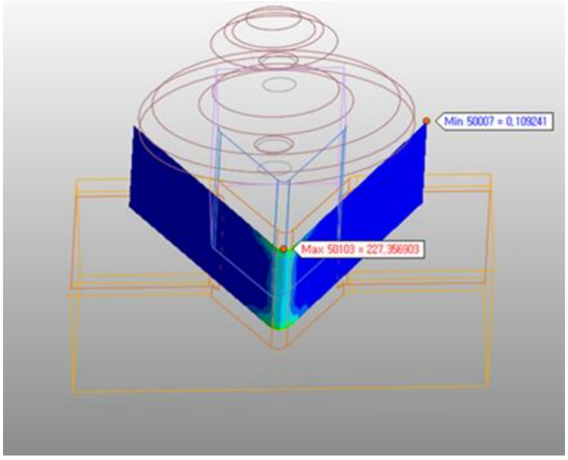
1. On the **Flexible** tab, in the **FFlex** group, click the **Contour icon** to display the Contour dialog window.
2. In the **Contour dialog window**, perform the following:
   - Click the **Calculation** button in the middle of the dialog window.
   - Select the **Show Min/Max** checkbox.
   - Click the **OK** button to see the results.
3. On the **Analysis** tab, in the **Animation Control** group, click the **Play** button.





- To see the contour result for **Plate_FE**, as shown below, click the **Wireframe** icon in the **Toolbar** and run the animation.
- You can see that, at **1.23 seconds**, the Maximum Von-Mises Stress is approximately **295.22 MPa**. After **1.52 seconds**, approximately **227.36 MPa** of residual stress remains.

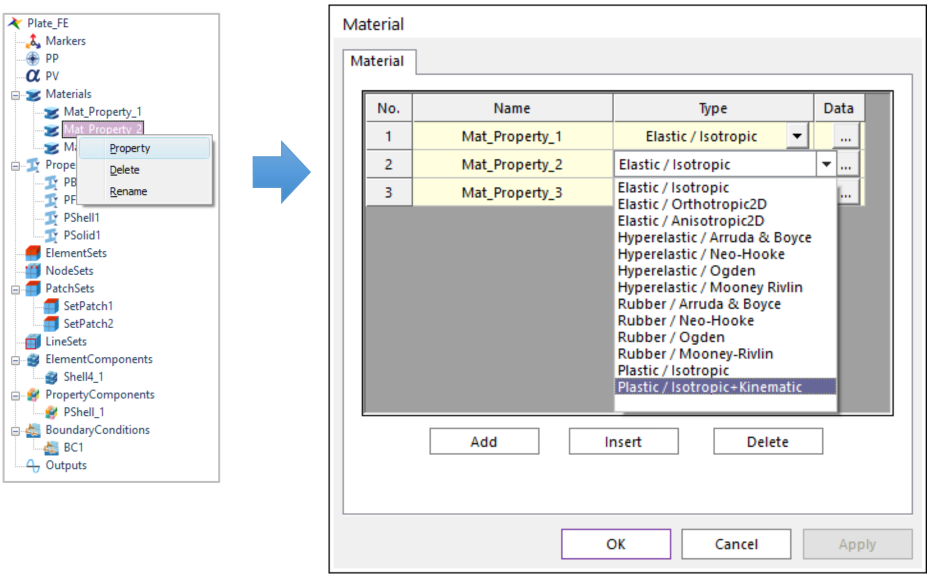

Min 50004 = 0,573474
Max 50678 = 295,217102


Min 50007 = 0,109241
Max 50103 = 227,356903

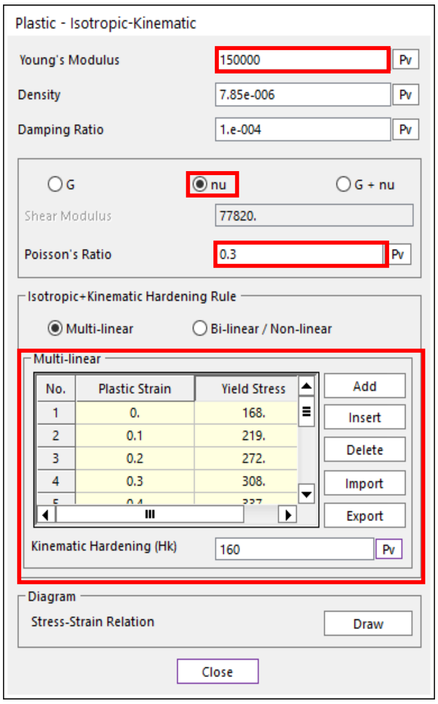
Performing Plastic Modeling (2)

To create a plastic material:

1. Double-click the **Plate_FE** body to enter **Body Edit** mode.
2. In the **Database** pane, right-click **Plate_FE**, and then select Edit in the context menu. Then, double-click Mat_Property_2 to open the **Material Property** dialog window, as shown below.



3. In the **Material** dialog window, select **Plastic / Isotropic + Kinematic** in the dropdown menu to the right of **Mat_Property_2** (No. 2). Then, click the … button to display the **Plastic – Isotropic-Kinematic** dialog window.
4. In the **Plastic – Isotropic-Kinematic** dialog window, perform the following: (If you carried out the procedure described in Performing Plastic Modeling (1), you don't need to change the Young’s Modulus and Nu options.)

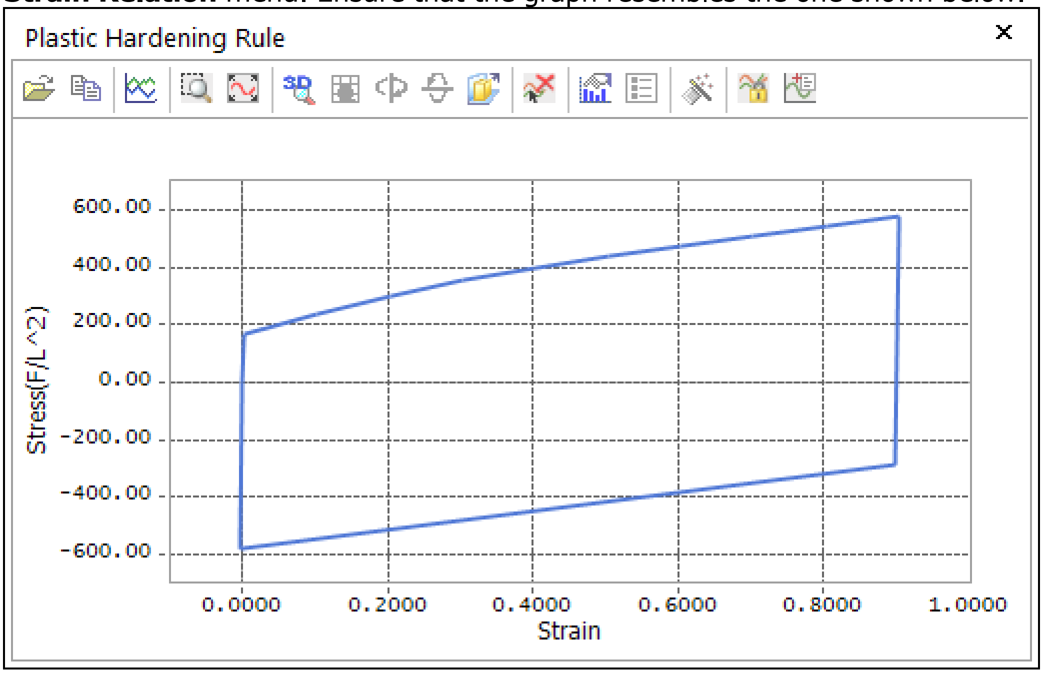


- Change the **Young's Modulus** value from 200,000 to **150,000**.
- Select the **Nu** radio button and change **Poisson's Ratio** to **0.3**.
- In the **Multi-linear** pane, click the **Add** button to add a new row. Then, type in the **Plastic Strain** and **Yield Stress** values shown in the upper table.

No.	Plastic Strain	Yield Stress
1	0	168
2	0.1	219
3	0.2	272
4	0.3	308
5	0.4	337
6	0.5	361
7	0.6	382
8	0.7	401
9	0.8	418
10	0.9	434

- In the **Kinematic Hardening (Hk)** field, type **160**.

5. To ensure that the values entered in the previous step are correct, in the **Diagram** pane, click the **Draw** button to the right of the **Stress-Strain Relation** menu. Ensure that the graph resembles the one shown below.



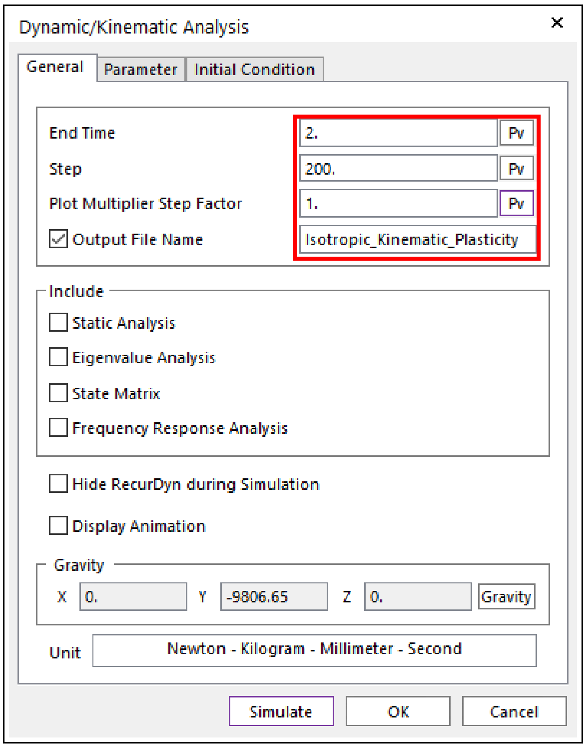
6. On the **FFlex Edit** tab, in the **Exit** group, click the **Exit icon** to return to a higher menu.

   In the **File** menu, click **Save As**, and save the model as **Plasticity_Bending_Machine_Isotropic_Kinematic.rdyn**.

Performing Plastic Analysis (2)

To see the results of the plastic analysis for the plastic material, you must run another simulation.

To run the simulation:



1. On the **Analysis** tab, in the **Simulation Type** group, click the **Dyn/Kin** icon.

   The Dynamic/Kinematic Analysis dialog window appears.

2. Set the **End Time** and **Step** fields as follows:
   - **End Time**: 2
   - **Step**: 200
   - **Plot Multiplier Step Factor**: 1
   - **Output File Name**: Isotropic_Kinematic_Plasticity

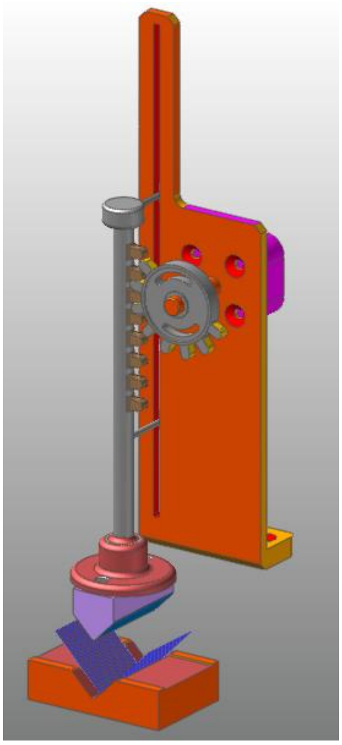
3. Click **Simulate**.

To view the results:

On the **Analysis** tab, in the **Animation Control** group, click the **Play** button.

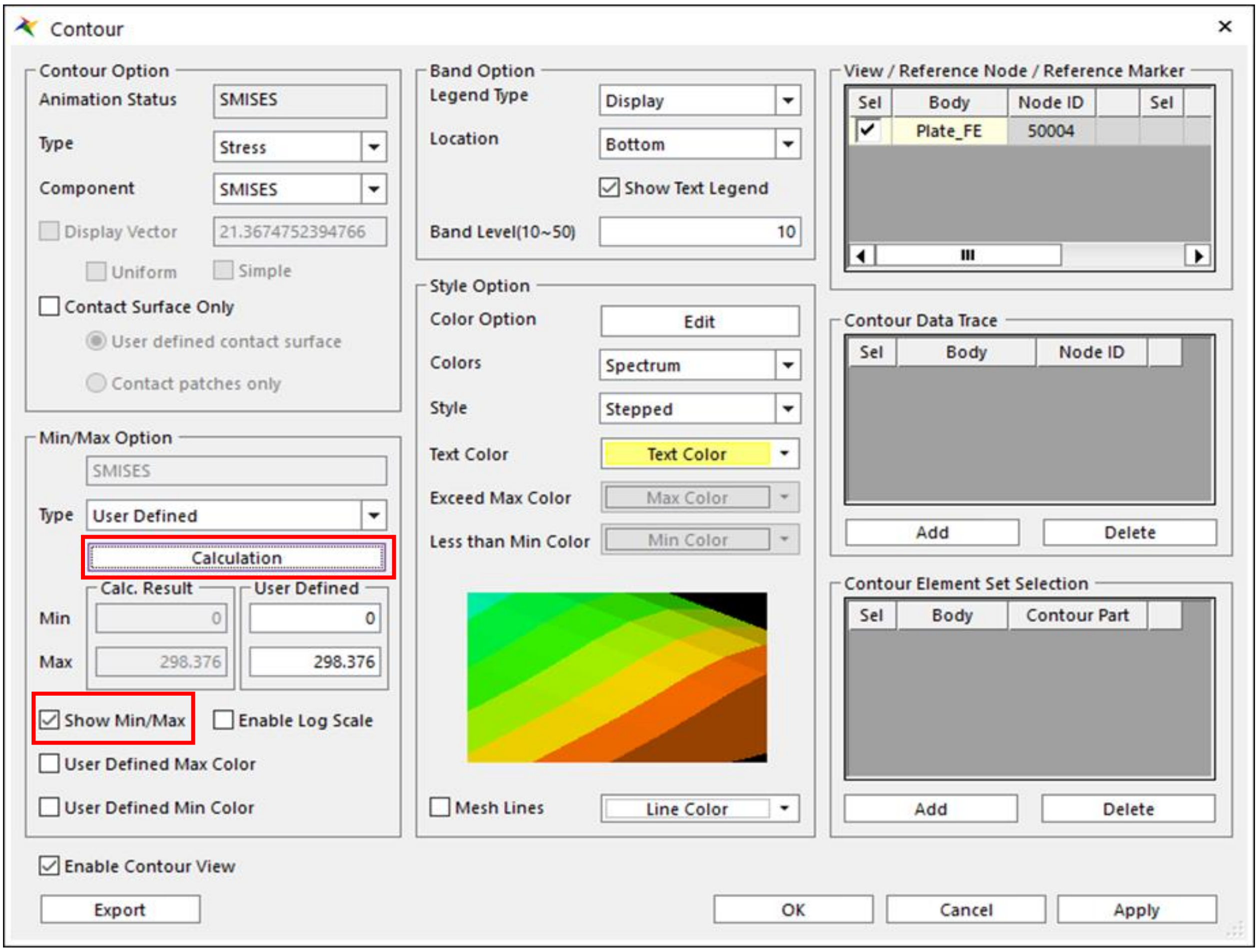
Just like the results of Performing Plastic Analysis (1), the metal plate in this simulation maintains the V shape of the die after the punch strikes it.

Because of this, the transfer device is raised again when the gear engages the teeth on the transfer device, and the process is repeated.

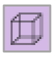


To check the contour results:

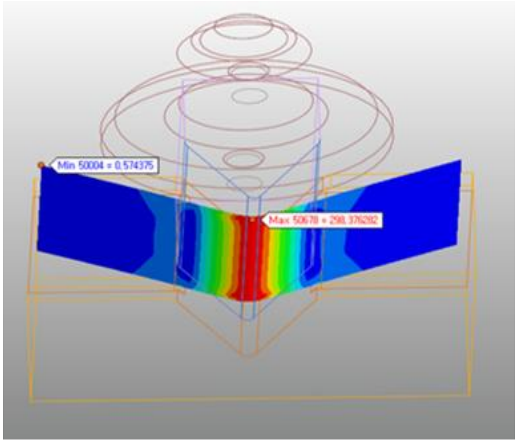
1. On the **Flexible** tab, in the **FFlex** group, click the **Contour icon** to display the **Contour** dialog window.
2. In the **Contour dialog window**, perform the following:
   - Click the **Calculation** button in the middle of the dialog window.
   - "Select the **Show Min/Max** checkbox.
   - Click the **OK** button to see the results.

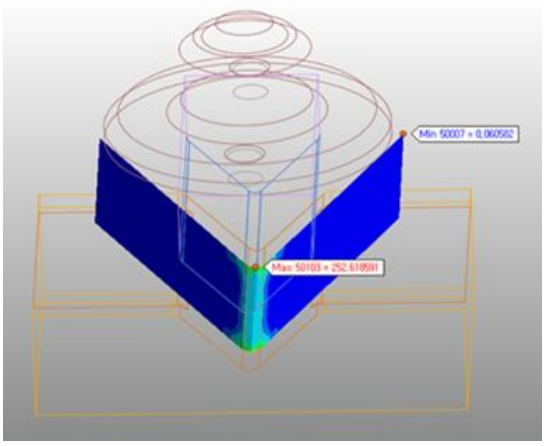


3. On the **Analysis** tab, in the **Animation Control** group, click the **Play** button.



   - To see the contour result for **Plate_FE**, as shown below, click the **Wireframe** icon in the **Toolbar** and run the animation.
   - You can see that, at **1.23 seconds**, the Maximum Von-Mises Stress is approximately **298.38 MPa**. After **1.53 seconds**, approximately **252.62 MPa** of residual stress remains.


Min 50004 = 0.574375
Max 50670 = 298.376282


Min 50007 = 0.060582
Max 50103 = 252.618591

Chapter

6

Analyzing and Reviewing the Results

Task Objectives

This chapter analyzes the results of the two plastic analyses using different plastic materials and compares these analyses with the elastic analysis results.

Estimated Time to Complete

10 minutes

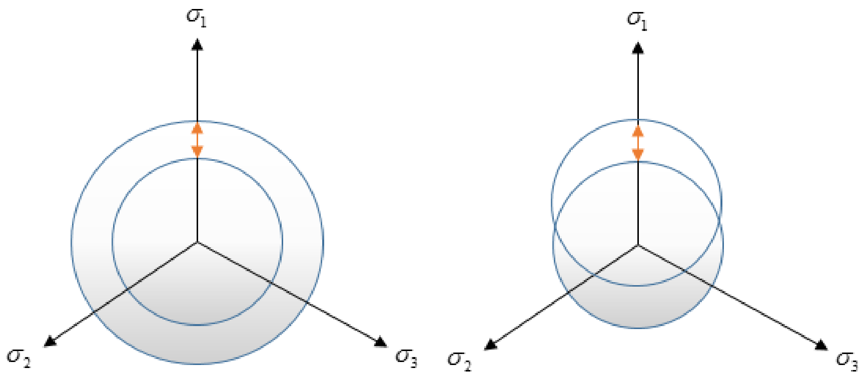
Analyzing the Plastic Analysis Results

Theoretical Explanation of Metal Plasticity

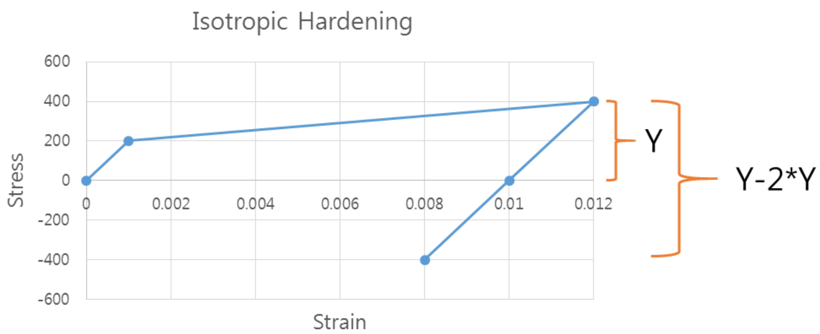
In the final analysis from the previous chapter, the metal plate modeled using a flexible body underwent plastic deformation due to the force of the punch striking it. The stress generated at the time of deformation is called yield stress. The stress can be expressed as a three-dimensional shape, such as a sphere or cube, by specifying the amount of stress along the X, Y, and Z axes in the three-dimensional space. The surface of such a sphere or cube is called the yield surface.

If the load is applied again to the flexible material constituting the metal plate after the material has passed from the elastic range into the plastic range, then the yield stress of the material increases along with the deformation of the material. This process is called hardening. Hardening is a fundamental property of a plastic material. RecurDyn can model two types of hardening.

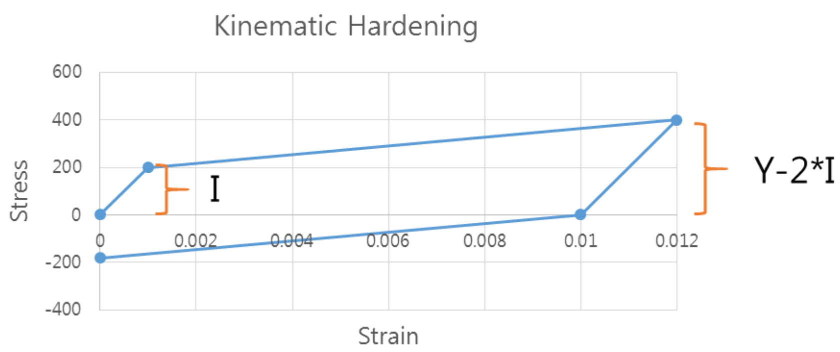
The first is isotropic hardening, in which the yield surface of the stress sphere expands in all directions at the same rate. The second one is kinematic hardening, in which the size of the yield surface remains the same, but the center of the surface shifts.



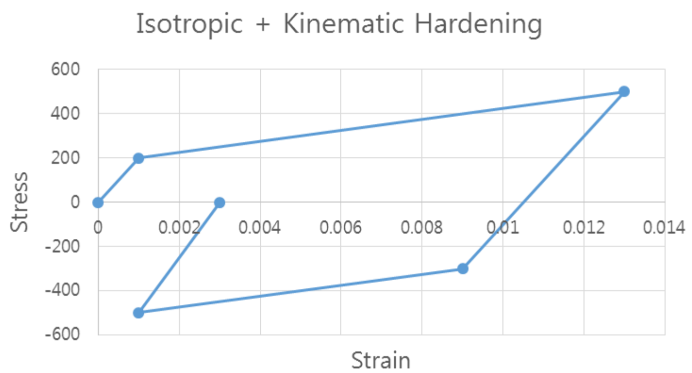
If only isotropic hardening is used, when the tensile load is removed and a compressive load is applied after the material has entered the plastic range, the yield stress of the material changes from +Y to –Y (Y - 2 × Y). During this process, the stress-strain relationship of the material shows linear elastic behavior.



If only kinematic hardening is used, when the tensile load is removed and the compressive load is applied after the yield stress has exceeded the initial yield stress (I) and the material has entered the plastic range, the yield stress of the material changes from +Y to Y - 2 × I. During this process, the stress-strain relationship of the material shows linear elastic behavior.



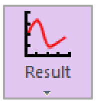
RecurDyn provides two different hardening options. The first option allows for only using isotropic hardening. The second option is to use isotropic + kinematic hardening, which combines both isotropic hardening and kinematic hardening.



In the bending machine used for this tutorial, no additional force is applied after the punch falls on the metal plate to cause deformation. Therefore, there is not much physical difference between isotropic hardening and kinematic hardening. However, if you use a non-zero value for Kinematic Hardening (Hk), the effective hardening modulus changes, causing the slope in the plastic range (plastic modulus) to change and leading to different residual stress and permanent strain results.

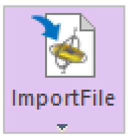
Comparing the Results of Plastic Analyses (1) and (2)

To compare the plot results:



1. On the **Analysis** tab, in the **Plot** group, click **Plot** to enter Plot mode.

   The Modeling work pane changes to the Plot work pane.



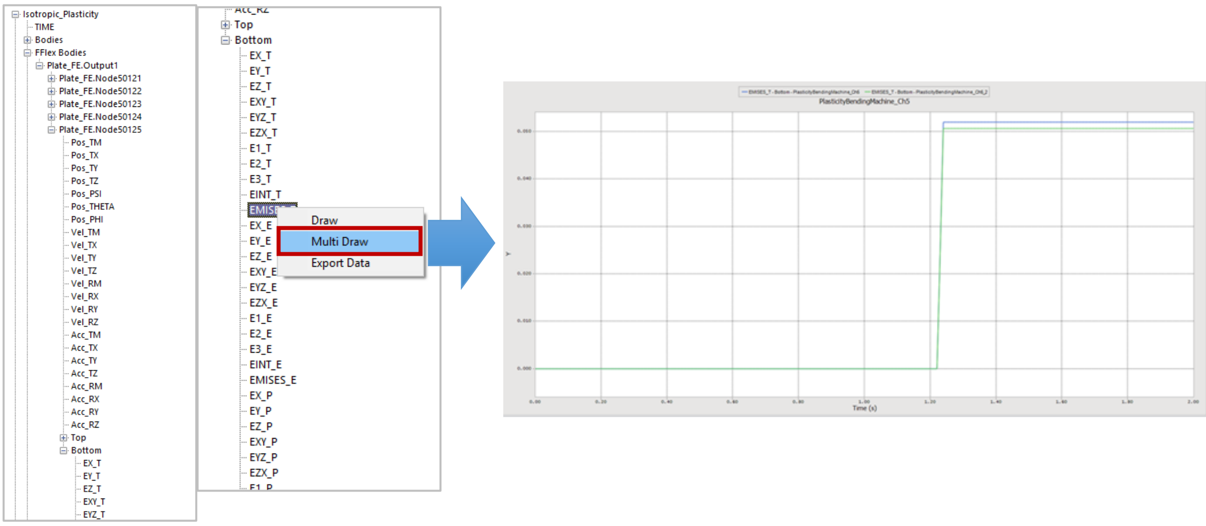
2. On the **Home** tab, in the **Import and Export** group, click **Import**.

   In the Import dialog window, select the ***.rplt files** created from the two plastic analyses.

   - In the output folder created in the first analysis, select **Isotropic_Plasticity.rplt.**
   - Click the **Import** icon again and, in the output folder created after the second simulation, select **Isotropic_Kinematic_Plasticity.rplt**. (If you entered Plot mode directly after finishing the **Isotropic+Kinematic** analysis, then this file is imported automatically.)

3. As shown in the figure below, the data from both rplt files appears in the **Database** pane.

- Click the + button to the left of **Isotropic_Plasticity**. In the expanded tree, click the + button to the left of **FFlex_Bodies-Plate_FE.Output1-Plate_FE_Node5032, Shell-Bottom**, and then select **EMISES_T**.
- Right-click EMISES_T to display the context menu, and then click **Multi Draw**.



4. Two graphs appear in the Plot work pane, as shown in the figures to the right.

   To match the figure shown on the right, right-click **SMISES** instead of **EMISES_T** in step 3 and select **Multi Draw.**

Analyzing the Results

The graphs drawn in the preceding procedure indicate the following:

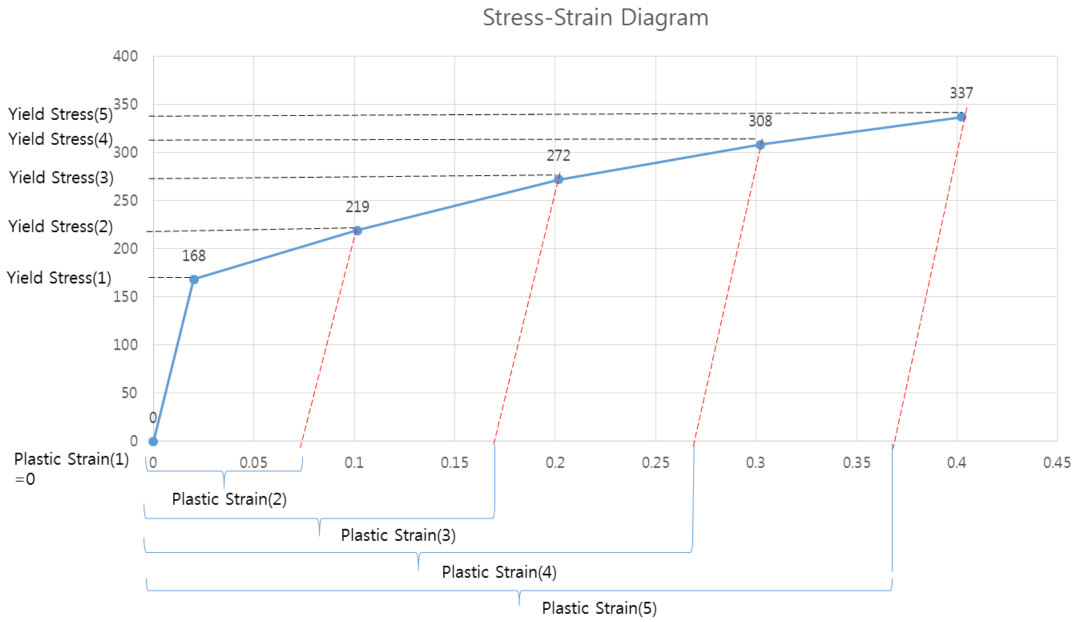
The total strain is composed of the combination of plastic strain and elastic strain. The plastic strain causes the permanent deformation of the metal plate. Also, a difference can be detected between the behavior of plasticity with isotropic hardening and plasticity with isotropic + kinematic hardening. As was explained earlier, this is due to the difference in the slope of the plastic range.

As you can see in the Von-Mises stress results, the metal plate maintains a certain residual stress after deformation of the plate occurs due to outside impact.

For Reference

Multi-linear Models

The stress-strain relationship for the plastic range can be expressed using a formula. This tutorial used a multi-linear model that calculates the plastic modulus using multiple linear data. This method extracts the plastic strain and yield stress values from empirical results obtained from experiments and uses the values to generate a multi-linear model.



As shown in the figure above, you can obtain the Plastic Strain(x) and Yield Stress(y) values through experiments and enter the values in the **Multi-linear** pane of the **Plastic – Isotropic** dialog window to create the blue stress-strain graph. In the stress-strain graph, the strain in the plastic range is the total strain that consists of both the plastic strain and elastic strain. Therefore, you must ensure that you enter only the plastic strain when you enter the data in the Multi-linear pane. In the experiment, you can obtain the plastic strain by applying a load larger than the yield stress and measuring the degree of deformation in the material.

Thanks for participating in this tutorial!