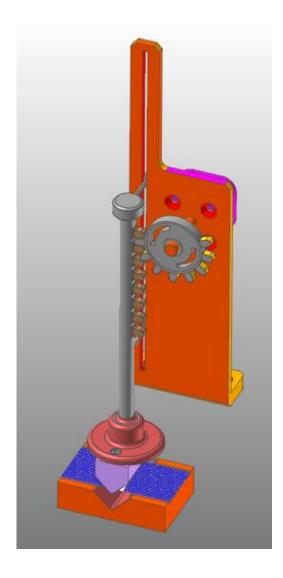


Plasticity Bending Machine Tutorial (FFlex)





Copyright © 2020 FunctionBay, Inc. All rights reserved.

User and training documentation from FunctionBay, Inc. is subjected to the copyright laws of the Republic of Korea and other countries and is provided under a license agreement that restricts copying, disclosure, and use of such documentation. FunctionBay, Inc. hereby grants to the licensed user the right to make copies in printed form of this documentation if provided on software media, but only for internal/personal use and in accordance with the license agreement under which the applicable software is licensed. Any copy made shall include the FunctionBay, Inc. copyright notice and any other proprietary notice provided by FunctionBay, Inc. This documentation may not be disclosed, transferred, modified, or reduced to any form, including electronic media, or transmitted or made publicly available by any means without the prior written consent of FunctionBay, Inc. and no authorization is granted to make copies for such purpose.

Information described herein is furnished for general information only, is subjected to change without notice, and should not be construed as a warranty or commitment by FunctionBay, Inc. FunctionBay, Inc. assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

The software described in this document is provided under written license agreement, contains valuable trade secrets and proprietary information, and is protected by the copyright laws of the Republic of Korea and other countries. UNAUTHORIZED USE OF SOFTWARE OR ITS DOCUMENTATION CAN RESULT IN CIVIL DAMAGES AND CRIMINAL PROSECUTION.

Registered Trademarks of FunctionBay, Inc. or Subsidiary

RecurDyn is a registered trademark of FunctionBay, Inc.

RecurDyn/Professional, RecurDyn/ProcessNet, RecurDyn/Acoustics, RecurDyn/AutoDesign, RecurDyn/Bearing, RecurDyn/Belt, RecurDyn/Chain, RecurDyn/CoLink, RecurDyn/Control, RecurDyn/Crank, RecurDyn/Durability, RecurDyn/EHD, RecurDyn/Engine, RecurDyn/eTemplate, RecurDyn/FFlex, RecurDyn/Gear, RecurDyn/DriveTrain, RecurDyn/HAT, RecurDyn/Linear, RecurDyn/Mesher, RecurDyn/MTT2D, RecurDyn/MTT3D, RecurDyn/Particleworks I/F, RecurDyn/Piston, RecurDyn/R2R2D, RecurDyn/RFlex, RecurDyn/RFlexGen, RecurDyn/SPI, RecurDyn/Spring, RecurDyn/TimingChain, RecurDyn/Tire, RecurDyn/Track_HM, RecurDyn/Track_LM, RecurDyn/TSG, RecurDyn/Valve

are trademarks of FunctionBay, Inc.

Edition Note

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

Table of Contents

Overview	4
Task Objective	4
Prerequisites	5
Tasks	5
Estimated Time to Complete	5
Opening the Initial Model	6
Task Objective	6
Estimated Time to Complete	6
Opening the RecurDyn Model	7
Simulating the Initial Suspension Model	8
Viewing the Result	8
Generating an FFlex Body	9
Task Objectives	9
Estimated Time to Complete	9
Creating the Box Geometry	10
Creating the Box Mesh	11
Performing Elastic Analysis	15
Task Objectives	15
Estimated Time to Complete	15
Performing Dynamic Modeling	16
Performing Elastic Analysis	20
Performing Plastic Analysis	23
Task Objectives	23
Estimated Time to Complete	23
Performing Plastic Modeling (1)	24
Performing Plastic Analysis (1)	26
Performing Plastic Modeling (2)	30
Performing Plastic Analysis (2)	32
Analyzing and Reviewing the Results	35
Task Objectives	35
Estimated Time to Complete	35
Analyzing the Plastic Analysis Results	36
For Reference	39



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



This tutorial takes approximately 65 minutes to complete.



Opening the Initial Model

Task Objective

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



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.
- 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.

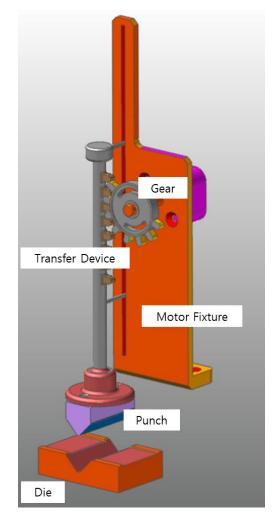
When the punch hits it, the metal plate deforms in the shape of the die. The teeth on the gear then engages the teeth on the transfer device to raise the punch again.

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.)



7

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.

Dynamic/Kinematic Analysis ×							
General Parameter Initial Condition							
End Time	2. Pi	<u>,</u>					
Step	200. Pi	·					
Plot Multiplier Step Factor	1. Pi	·					
Output File Name							
□ □ Include ─────							
Static Analysis							
Eigenvalue Analysis							
State Matrix							
Frequency Response Analysis							
Hide RecurDyn during Simula	tion						
Display Animation							
Gravity							
X 0. Y -9806.65 Z 0. Gravity							
Unit Newton - Kilogram - Millimeter - Second							
Simulate OK Cancel							

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.



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).



20 minutes

Creating the Box Geometry

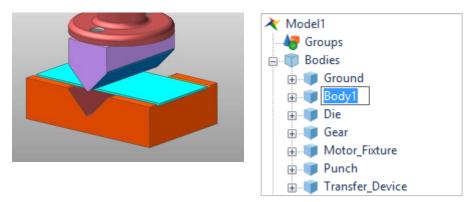
Box

- 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.)

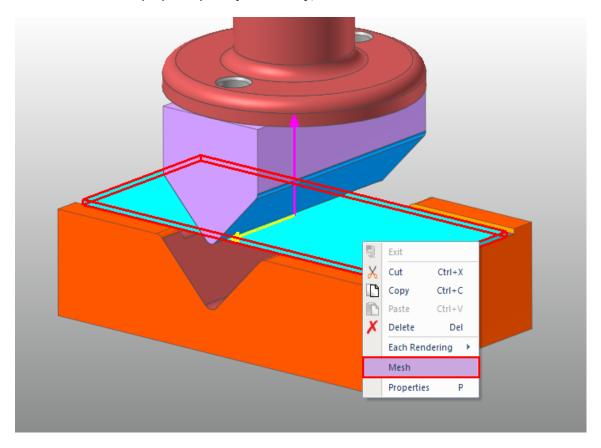
3. After you enter the values, the box geometry (shown below) appears. In the **Database** pane, change the name of **Body1** to **Plate**.



Creating the Box Mesh

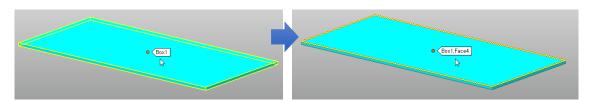
1. In **Assembly** mode, right-click the work pane to display the context menu and click **Mesh**.

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

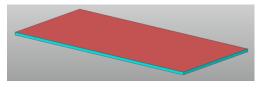




- 2. On the **Geometry** tab, in the **Surface** group, select **Face Surface**.
- 3. Select **Box1** to open the **FaceSurf Operation** dialog window. With the **FaceSurf Operation** dialog window open, select **Box1.Face4** on top of the box.



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





- On the **Mesher** tab, click the **Mesh** icon to open the Mesh dialog window.
- 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.

Mesh	×			
Target Geometry	FaceSurface1 Gr			
Use Current Shell Mesh Info	h			
Mesh Type	Shell4(Quad4) 💌			
Property	PShell1 P			
Mesh Option ————				
Max Element Size	3			
Min Element Size	3			
Chordal Error Ratio	Relative 🔻 0.1			
Structured Output	Simple Pattern			
Close Gaps				
Weight on Quadrangles	w High			
Additio	onal Setting			
Include Assist Modeling				
Auto Flex Merge				
Create Beam Element with Pre-Stress				
Devent 1	dark Consul			
Revert	Mesh Cancel			

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.

✓ Plate_FE	Proper Proper				
	No.	Name	Туре	Material	Data
Mat_Property_1	1	PBeam1	PBeam	Mat_Propert Mt.	
Mat_Property_2 Mat_Property_3	2	PShell1	PShell	Mat_Propert Mt.	
Properties	3		PSolid		
PBeam1	2	PSolid1	PSolid	Mat_Propert Mt.	
PSheut	4	PFDR1	PFDR	Mt.	
PSol Property					
PFD <u>D</u> elete					
Element Rename					
NodeSet					
ElementComponents					
Shell4_1				at Datata	
PropertyComponents		Add	Inse	rt Delete	
PShell_1					
- Outputs			ОК	Cancel	Apply

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.

Property Shell		
Thickness	0.5	Pv
Drilling Stiffness Factor [L^2]	1.	Pv
	Close	

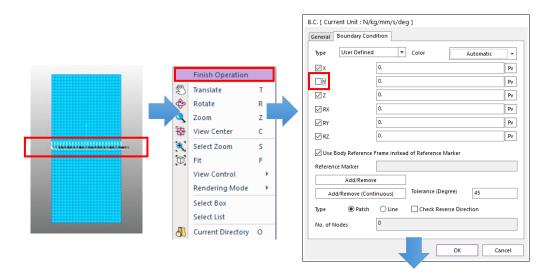
- 9. In the **Property** dialog window, click the **Mt.** button to the right of No. 2 PShell1. The **Material** dialog window appears (shown below).
 - In the **Material** dialog window, click the ... button to the right of No. 2, and then change the following settings:

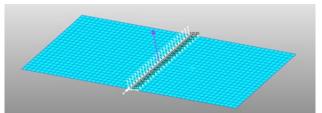
N	1aterial				1	🕻 Elastic - Isotropio	c				x
	Material					Young's Modulus		150000.		Pv]
	No.	Name	Туре	Data		Density		7.85e-006		Pv	
	1	Mat_Property_1 Mat Property 2	Elastic / Isotropic Elastic / Isotropic			Damping Ratio		1.e-004		Pv	
	3	Mat_Property_2	Elastic / Isotropic		Π/	Damping Ratio		1.6-004			
						O G Shear Modulus (G) Poisson's Ratio (nu)	57692.30769230 0.3	⊖ G + nu 77	Pv	
		Add	nsert Delete			Thermal Coefficient		1.2e-005		Pv	
						Thermal Reference T	emp[c]	20.		Pv	
			DK Cancel	Apply		Specific Heat [J/(kgc)]	420.		Pv	
						Thermal Conductivity	y [W/(mc)]	80.		Pv	
								Close			:

- 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.



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.

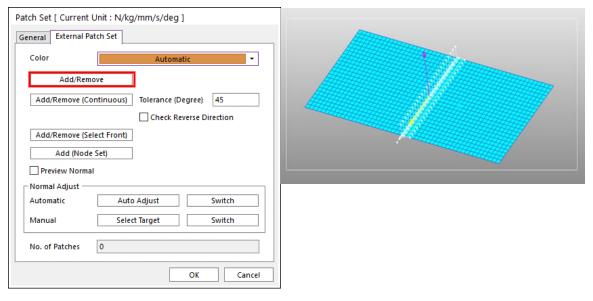


10 minutes

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.)



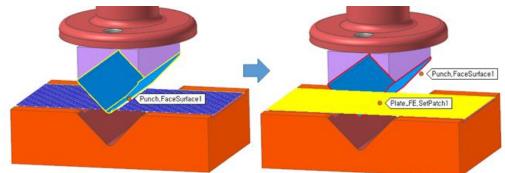
(All and a second secon	Patch Set [Current Unit : N/kg/mm/s/deg]
	General External Patch Set
	Color Automatic 👻
	Add/Remove
	Add/Remove (Continuous) Tolerance (Degree) 45
	Check Reverse Direction
AAAAAAAA	Add/Remove (Select Front)
******	Add (Node Set)
	Normal Adjust
	Automatic Auto Adjust Switch
	Manual Select Target Switch
	No. of Patches 1150
	OK Cancel

- 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:

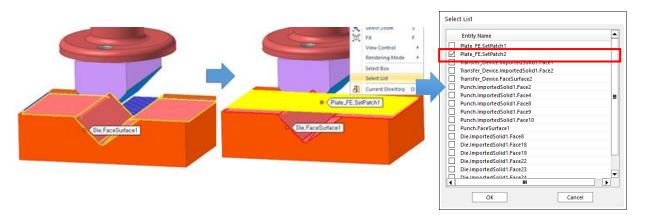
GeoSur

 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.



- 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.

	Properties of GeoSurContact4 [Current Unit : N/kg/mm/s/deg]
	General Characteristic Geo Contact
	Definition of the Base Geometry
4	Name Die.FaceSurface1 Gr
	Normal Direction Up Down Node Contact
	Calc. Pressure Preview Contact Geometry
* * *	Definition of the Action Geometry
	Name Plate_FE.SetPatch2 Gr
	Geometry Type Surface 💌
	Normal Direction Up O Down Node Contact
	Preview Contact Geometry
	Edge Contact Advanced Setting
	No. of Max Contact Points 10 🚖
	Generate a Contact Output File (*.con) Ground.InertiaMark
	Force Display Inactivate Force Vector
	Reference Marker of Force Display
	Scope OK Cancel Apply

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.**

Properties of GeoSurContact4 [Current Unit : N/kg/mm/s/deg]	Surface Patch	×
General Characteristic Geo Contact	Surface Name Plate_FE.Set	tPatch2
Definition of the Base Geometry Name Die.FaceSurface1 Gr	Surface Type Quad	T
Normal Direction O Up O Down Node Contact	Bounding Buffer Length 25	Pv Cal.
Calc. Pressure Preview Contact Geometry	Thickness	
Definition of the Action Geometry Name Plate FE.SetPatch2 Gr	Original Specific 0,	Pv
Name Plate_FE.SetPatch2 Gr Geometry Type Surface	Cubic Cell Size (X, Y, Z) 100, 1, 45	
Normal Direction	ОК	Cancel
Image: Contact Advanced Setting No. of Max Contact Points 10		
Generate a Contact Output File (*.con) Ground.InertiaMark		
Force Display Inactivate Force Vector Reference Marker of Force Display		
Scope OK Cancel Apply		

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.

То	view	the	results:
••			I COMILOI

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

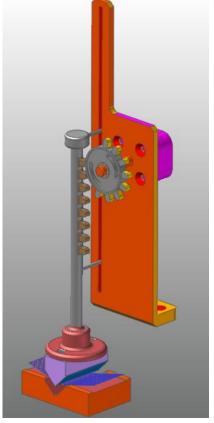
Unlike the transfer device in previous simulation, in which the punch came into direct contact with the die, the transfer device in this simulation behaves differently because the contact occurs between the Plate_FE flexible body and the punch and die.

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.

Dynamic/Kinematic Analysis						
General Parameter Initial Condition						
End Time	2.	Pv				
Step	200.	Pv				
Plot Multiplier Step Factor	1.	Pv				
Output File Name						
_ Include						
Static Analysis						
Eigenvalue Analysis						
State Matrix						
Frequency Response Analysis						
Hide RecurDyn during Simulation						
🗹 Display Animation						
Gravity	Gravity					
X 0. Y -9806.65 Z 0. Gravity						
Unit Newton - Kilogram - Millimeter - Second						
Simulate OK Cancel						



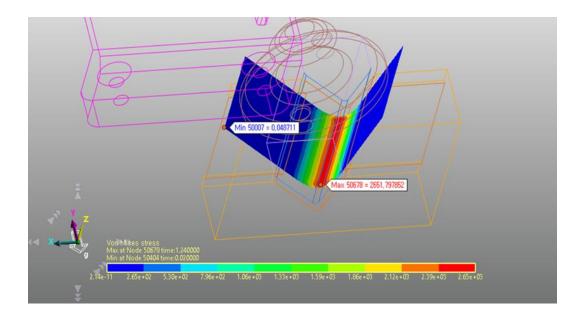
To check the contour results:



- 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.

nimation Status	Contour Option		Band Option			View / Reference Node / Reference Mark				
Animation Status SMISES			Legend Type	Display	-	Sel	Body	Node ID	Sel	
Type Stress 👻		Location	Bottom	•		Plate_FE	50004			
omponent	SMISES	•		Show Text Leg	end					
Display Vector	21.367475239	94766	Band Level(10~50)		10					
Uniform	Simple		Style Option			4			Þ	
Contact Surface			Color Option	Edit		Contou	ir Data Trace	-		
	ed contact surfa	ce	Colors	Spectrum	•	Sel	Body	Node IE	>	
Contact pa	tenes only		Style	Stepped	•					
lin/Max Option -]	Text Color	Text Color	•					
pe Display		-	Exceed Max Color	Max Color	*					
	alculation		Less than Min Color	Min Color	•		Add	D	elete	
Calc. Result	User Def	ined —		_	ſ	Contou	ur Element Se	et Selection —		
lin 2.13843e-	11 2.138	43e-11				Sel	Body	Contour Pa	rt	
lax 265	1.8	2651.8								
Show Min/Max	Enable Log	Scale								
User Defined Ma	_				_					
User Defined Mir			Mesh Lines	Line Color	.		Add	De	elete	
						-				

- 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.





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.



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.

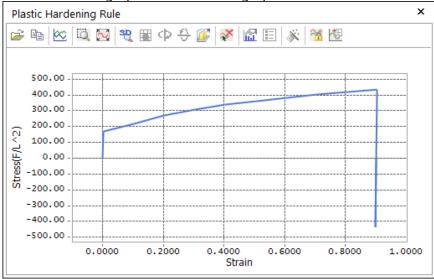
▼ Plate_FE	Materia			
→ Aarkers → PP	Materia			
- CC PV				
🖃 🥪 Materials	No.	Name	Туре	Data
Mat_Property_1	1	Mat_Property_1	Elastic / Isotropic 🔻	
Mi Property	2	Mat_Property_2	Plastic / Isotropic	▼
Prope Delete	3	Mat_Property_3	Elastic / Isotropic Elastic / Orthotropic2D	
PSolid1 Policies Policies Patchets SetPatch1 SetPatch1 SetPatch2 ElementComponents SetPl4_1			Elastic / Anisotropic2D Hyperelastic / Arruda & Boyc Hyperelastic / Neo-Hooke Hyperelastic / Noo-Hooke Hyperelastic / Mooney Rivlin Rubber / Arruda & Boyce Rubber / Neo-Hooke Rubber / Neo-Hooke Rubber / Ogden Rubber / Ogden Rubber / Jooropy-Rivlin Plastic / Isotropic	
PropertyComponents PShell_1 BoundaryConditions BC1 Outputs		Add	Insert Delete	
V			OK Cancel	Apply

- 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

Plas	tic - Is	otropic		
You	ng's M	odulus	150000	Pv
Den	sity		7.85e-006	Pv
Dam	iping F	Ratio	1.e-004	Pv
	G		🖲 nu	⊖G + nu
She	ear Mo	dulus	77820.	
Poi	sson's	Ratio	0.3	Pv
	lulti-lir No.	Plastic Strain	Yield Stress	Add
	8	0.7	401	Insert
	9 10	0.8	418	Delete
	10	0.9	434	Import
	•	ш		Export
_ Dia	agram			
	-	rain Relation		Draw
			Close	

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.

Output [Current Unit : N/kg/mm/s/deg] General Output Color Automatic Add/Remove
Add/Remove (Continuous) Tolerance (Degree) 45 Type Patch Line Check Reverse Direction Add by Set Add by B.C. Add by Marker
No. of Nodes 5 OK Cancel

- 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.

Dynamic/Kinematic Analysis	<
General Parameter Initial Condition	
	٦
End Time 2. Pv	
Step 200. Pv	
Plot Multiplier Step Factor 1. Pv	
Output File Name Isotropic_Plasticity	
Static Analysis	
Eigenvalue Analysis	
State Matrix	
Frequency Response Analysis	
Hide RecurDyn during Simulation	
Display Animation	
Gravity	
X 0. Y -9806.65 Z 0. Gravity	
Unit Newton - Kilogram - Millimeter - Second	
Simulate OK Cancel	

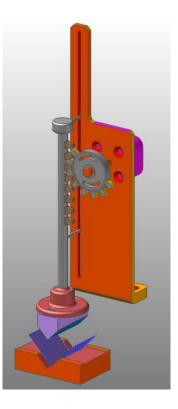
To view the results:

►

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:

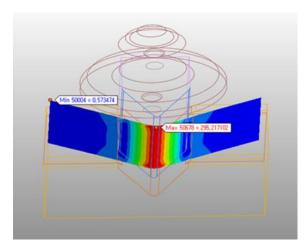


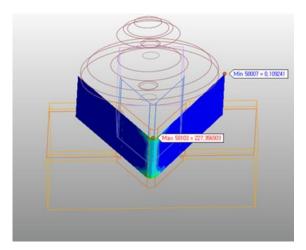
- 1. On the **Flexible** tab, in the **FFlex** group, click the **Contour icon** to display the Contour dialog window.
- 2. In the **Contour dialog window**, perform the following:
 - Click the **Calculation** button in the middle of the dialog window.
 - Select the **Show Min/Max** checkbox.
 - Click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

Contour								
Contour Option —		Band Option		Vi	ew/F	Reference No	ode / Reference	Marker -
Animation Status	SMISES	Legend Type	Display		Sel	Body	Node ID	Sel
Type	Stress 💌	Location	Bottom	•	~	Plate_FE	50004	
Component	SMISES 💌		Show Text Legend					
Display Vector	21.3674752394766	Band Level(10~50)	1	10				
Uniform	Simple	Style Option				ш		1
Contact Surface		Color Option	Edit		ontou	r Data Trace		
	ed contact surface	Colors	Spectrum	-	Sel	Body	Node ID	
Contact pa	atches only	Style	Stepped	-				
Min/Max Option -		Text Color	Text Color	•				
Type Display	 	Exceed Max Color	Max Color	-				
	Calculation	Less than Min Color	Min Color	-		Add	De	lete
Calc. Result	User Defined			L_CC	ontou	r Element Se	t Selection —	
Min	0 0				Sel	Body	Contour Part	
Max 295.3	217 295.217							
295.4	255.217							
Show Min/Max	Enable Log Scale							
User Defined Ma	ix color							
User Defined Mi	n Color	Mesh Lines	Line Color	- [Add	Del	ete
Enable Contour	View	J [
	7							
Export				OK		Cancel	A	pply

- 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.







Performing Plastic Modeling (2)

To create a plastic material:

- 1. Double-click the **Plate_FE** body to enter **Body Edit** mode.
- 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.

	No.	Name	Туре	Data
Mat Property 2 Mi Property	1	Mat_Property_1	Elastic / Isotropic 🔹	
Prope Delete	2	Mat_Property_2	Elastic / Isotropic	•
PP B PF PF PSolid1 PSolid1 ElementSets NodeSets PatchSets SetPatch1 SetPat	3	Mat_Property_3	Elastic / Isotropic Elastic / Orthotropic2D Elastic / Anisotropic2D Hyperelastic / Aruda & Boyce Hyperelastic / Neo-Hooke Hyperelastic / Mooney Rivlin Rubber / Neo-Hooke Rubber / Neo-Hooke Rubber / Mooney-Rivlin Plastic / Isotropic	
BoundaryConditions		Add	Insert Delete	

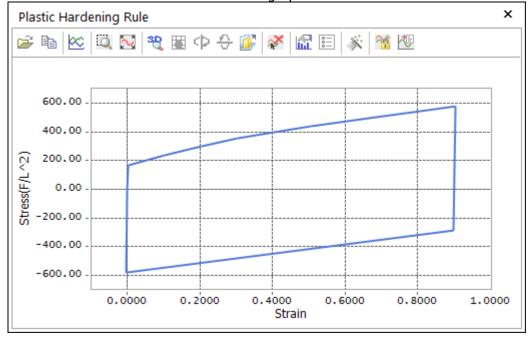
- 3. In the **Material** dialog window, select **Plastic / Isotropic + Kinematic** in the dropdown menu to the right of **Mat_Property_2** (No. 2). Then, click the ... button to display the **Plastic Isotropic-Kinematic** dialog window.
- 4. In the **Plastic Isotropic-Kinematic** dialog window, perform 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

Plastic -	Isotropic-Kinematic	:	
Young's I	dulus	150000	Pv
Density		7.85e-006	Pv
Damping	Ratio	1.e-004	Pv
0	. Г	🖲 nu	⊖G+nu
Shear M		77820.	
Poisson	s Ratio	0.3	Pv
Isotropi	c+Kinematic Hardenii	ng Rule	
	/ulti-linear	🔵 Bi-linear / Non-lin	ear
Multi-I	inear —		
Multi-I	inear Plastic Strain	Yield Stress	Add
No.	Plastic Strain 0.	168. =	Add
No. 1 2	Plastic Strain 0. 0.1	168. = 219.	
No.	Plastic Strain 0.	168. =	Insert Delete
No. 1 2 3	Plastic Strain 0. 0.1 0.2	168. = 219. 272.	Insert Delete Import
No. 1 2 3 4 €	Plastic Strain 0. 0.1 0.2 0.3	168. 219. 272. 308.	Insert Delete
No. 1 2 3 4 c Kinema	Plastic Strain 0. 0.1 0.2 0.3 0.3 0.4 III It Hardening (Hk)	168. ■ 219. 272. 308.	Insert Delete Import Export
No. 1 2 3 4 c Kinema	Plastic Strain 0. 0.1 0.2 0.3 0.3 0.4 III It Hardening (Hk)	168. ■ 219. 272. 308.	Insert Delete Import Export

- 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.



 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.

Dynamic/Kinematic Analysis

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:

Dyn/Kin

►

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.

General Parameter Initial Condit	ition						
End Time	2. Pv						
Step	200. Pv						
Plot Multiplier Step Factor 1. Pv							
✓ Output File Name Isotropic_Kinematic_Plasticity							
Include							
Static Analysis							
Eigenvalue Analysis							
State Matrix							
Frequency Response Analysis	s						
Hide RecurDyn during Simula	ation						
Display Animation							
_ Gravity							
X 0. Y -9806.65	5 Z O. Gravity						
Unit Newton - Kilogra	am - Millimeter - Second						
Simul	Ilate OK Cancel						

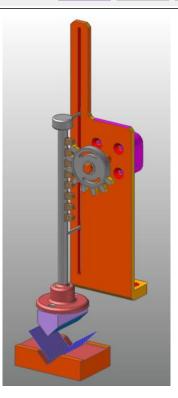
×

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:

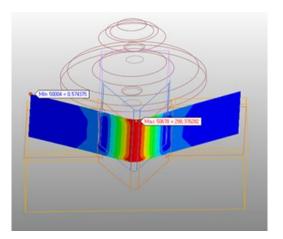


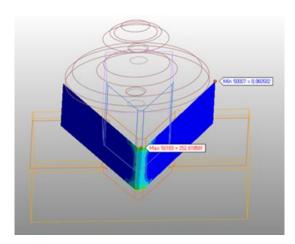
Ø

- 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.

Contour Option —	5.5		Band Option			View /	Reference N	ode / Reference	e Marker -
Animation Status	SMISES		Legend Type	Display	-	Sel	Body	Node ID	Sel
lype	Stress	•	Location	Bottom	•		Plate_FE	50004	
Component	SMISES	-		Show Text Leg	end				
Display Vector	21.36747523947	766	Band Level(10~50)		10				
Uniform	Simple		Style Option				Ш		U
Contact Surface			Color Option	Edit		Conto	ur Data Trace	(
	ed contact surface		Colors	Spectrum	•	Sel	Body	Node I	>
Contact pa	itches only		Style	Stepped	•				
In/Max Option -			Text Color	Text Color	•				
		_	Exceed Max Color	Max Color	-				
/pe User Defined	Calculation		Less than Min Color	Min Color	•		Add	D	elete
Calc. Result		ed 🖳			_	Conto	ur Element Se	et Selection —	
1in	0	0				Sel	Body	Contour Pa	rt
Max 298.	376 298	.376							
Show Min/Max	Enable Log S	cale							
User Defined Ma									
User Defined Mi			Mesh Lines	Line Color			Add	D	elete
Enable Contour	View								

- 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.







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

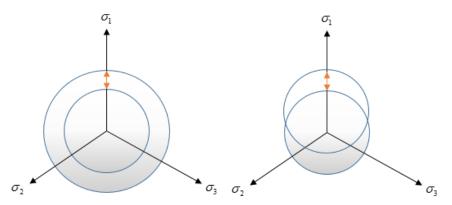
Analyzing the Plastic Analysis Results

Theoretical Explanation of Metal Plasticity

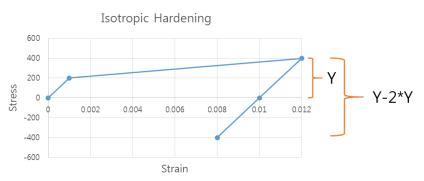
In the final analysis from the previous chapter, the metal plate modeled using a flexible body underwent plastic deformation due to the force of the punch striking it. The stress generated at the time of deformation is called yield stress. The stress can be expressed as a three-dimensional shape, such 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.

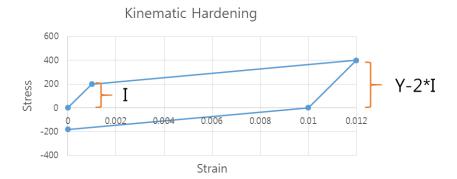
The first is isotropic hardening, in which the yield surface of the stress sphere expands in all directions at the same rate. The second one is kinematic hardening, in which the size of the yield surface remains the same, but the center of the surface shifts.



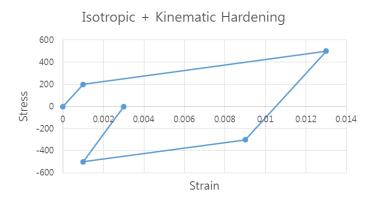
If only isotropic hardening is used, when the tensile load is removed and a compressive load is applied after the material has entered the plastic range, the yield stress of the material changes from +Y to -Y (Y - 2 × Y). During this process, the stress-strain relationship of the material shows linear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

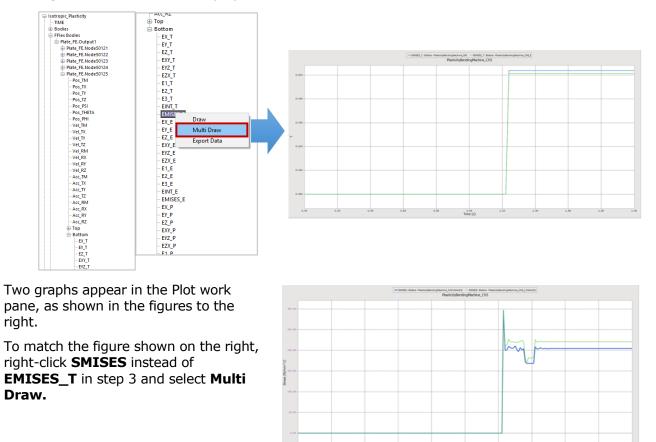
The Modeling work pane changes to the Plot work pane.

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**.



Analyzing the Results

4.

The graphs drawn in the preceding procedure indicate the following:

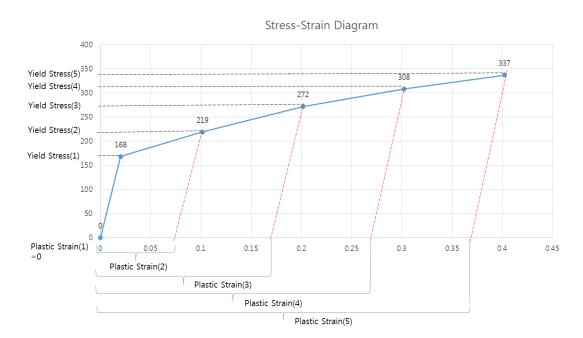
The total strain is composed of the combination of plastic strain and elastic strain. The plastic strain causes the permanent deformation of the metal plate. Also, a difference can be detected between the behavior of plasticity with isotropic hardening and plasticity with isotropic + kinematic hardening. 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!