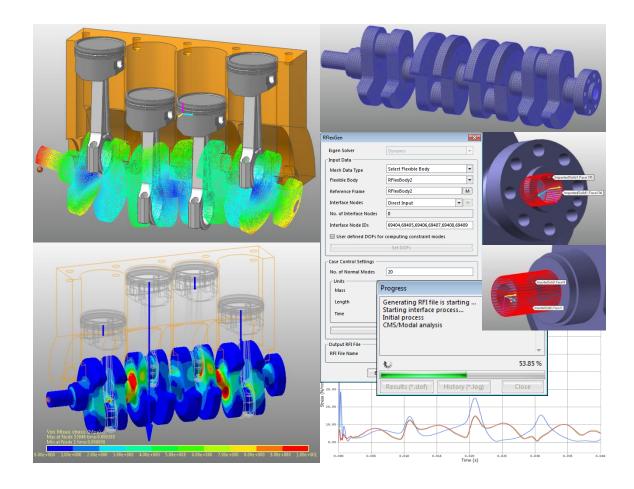


RFlexGen Crankshaft Tutorial (RFlexGen)





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
Tutorial Objectives4
Prerequisites5
Tasks5
Estimated Time to Complete5
Opening the Initial Model6
Task Objectives6
Estimated Time to Complete6
Opening the RecurDyn Model
Running the Initial Simulation on the 4-Cylinder Engine Model
Generating the FFlex Body
Task Objectives9
Estimated Time to Complete9
Creating the Crankshaft Mesh 10
Conducting the Dynamic Analysis on the FFlex Body and Reviewing the Results16
Generating the RFlex Body Using RFlexGen
Task Objectives
Estimated Time to Complete
Running RFlexGen
Creating the RFlex Body25
Conducting a Dynamic Analysis on the RFlex Body and Reviewing the Results27
Running RFlexGen Again
Replacing an RFlex Body
Conducting a Dynamic Analysis on the RFlex Body and Reviewing the Results36
Analyzing and Reviewing the Results
Task Objectives
Estimated Time to Complete this Task
Analyzing the Dynamic Analysis Results



Overview

RecurDyn allows you to use FFlex and RFlex bodies to simulate flexible bodies in multi-body dynamic models. To create FFlex bodies in RecurDyn, you can simply apply the FFlex/Mesher to the rigid bodies that you want to make flexible. However, creating RFlex bodies in versions up to RecurDyn V8R2 required separate finite element (FE) software with an Eigen solver. So, you had to create files using the Eigen solver in the FE software and then process the files in RecurDyn to generate the RFlex bodies.

To remedy this, RecurDyn V8R3 includes a function that can create RFI (Reduced Flexible Body Interface) files using its own Eigen solver. These files can then be used to generate RFlex bodies.

In this tutorial, we will show you the benefits of RecurDyn's new function, RFlexGen, and how to use it to create an RFlex body in a multi-body dynamic model. The model we will use in this tutorial is an in-line 4-cylinder internal combustion engine that simulates the combustion processes that normally occur in an engine. We will achieve the objectives of this tutorial by examining the analysis outcomes when the crankshaft body is replaced by an FFlex or RFlex body.

Tutorial Objectives

In this tutorial, you will learn how to:

- Create FFlex bodies using RecurDyn Mesher
- Examine the dynamic characteristics of flexible bodies made using RecurDyn/FFlex
- Create RFI files from FFlex bodies using RecurDyn/RFlexGen
- Replace RFlex bodies using RecurDyn/RFlex and examine their dynamic characteristics
- Create RFI files from RFlex bodies using RecurDyn/RFlexGen
- Compare the dynamic characteristics of FFlex bodies and RFlex bodies
- Recognize the benefits of using the RFlexGen function

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 we recommend completing 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.

Procedures	Time (minutes)
Opening the Rdyn model	10
Generating the FFlex body using the Mesher	15
Analyzing the dynamic characteristics of the FFlex body	10
Creating the RFI file using RFlexGen	10
Replacing the existing body with the RFlex body	5
Analyzing the dynamic characteristics of the RFlex body	5
Creating another RFI file using RFlexGen	5
Analyzing the dynamic characteristics of the RFlex body	5
Comparing the analysis results	10
Total	70



This tutorial takes approximately 70 minutes to complete.



Opening the Initial Model

Task Objectives

This chapter teaches you how to open the initial model, run a simulation, and observe how the 4-cylinder engine model operates.



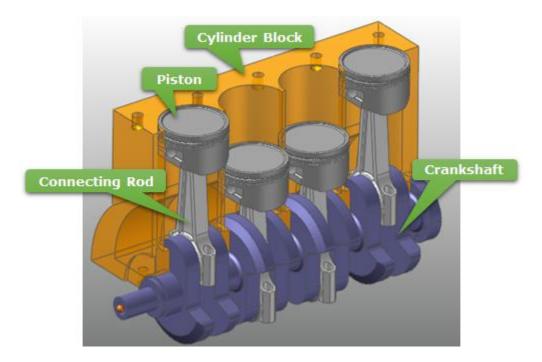
10 minutes

Opening the RecurDyn Model

To run RecurDyn and open the initial model:

- 1. On your Desktop, double-click the **RecurDyn** shortcut.
- 2. When the **Start RecurDyn** dialog box appears, close it.
- 3. In the **File** menu, click **Open**.
- 4. Navigate to the **RFlexGen** tutorial folder, and then select **RD_RFlexGen_4Cyl_Engine_Start.rdyn**. (**The file location:** <Install Dir> \Help \Tutorial \Flexible \RFlexGen, ask your instructor for the location of the directory if you cannot find it).
- 5. Click **Open**.

The model shown below opens.



The configuration of the model is as follows.

The model is 4-cylinder in-line engine. It consists of a cylinder block, pistons, connecting rods, and a crankshaft. In an actual engine, the explosive force generated in the cylinder drives the pistons downward in the cylinder block. This causes the connecting rods attached to the pistons to rotate the crankshaft. To simulate this process in RecurDyn, the force generated by the explosion in the cylinder is converted into a force profile that applies force to each piston at the correct time.

To save the initial model:

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

(Save this model in the different path because it is impossible to simulate directly in the tutorial path.)

Running the Initial Simulation on the 4-Cylinder Engine Model

To understand the behavior of the model, we will perform an initial simulation using the model.

To perform the initial simulation:



►

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

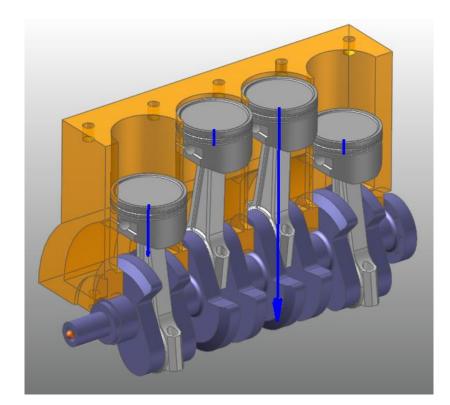
The **Dynamic/Kinematic Analysis** dialog box appears.

2. After verifying the simulation conditions, click **Simulation**.

To view the results:

On the **Analysis** tab, in the **Animation Control** group, click **Play** to run the simulation.

If you look closely at the behavior of the model, you can see that the combustion occurs in the following order: Piston_1 \rightarrow Piston_3 \rightarrow Piston_4 \rightarrow Piston_2. This behavior is represented by the lengthening of the arrows, which represent the force in the animation. Although a typical four-cylinder engine's stroke process consists of intake \rightarrow compression \rightarrow combustion \rightarrow exhaust, only the force generated by the combustion is meaningful in the dynamic model used for this tutorial. Therefore, the explosive force is converted into a force profile that applies force to each piston at the correct time.





Generating the FFlex Body

To analyze the multi-flexible body dynamics (MFBD) of the engine model, you must convert the crankshaft into a flexible body (FFlex body). This procedure maintains most of existing elements from the previous model, such as the joints and force. However, we will replace the rigid crankshaft with a flexible body to improve efficiency.

Task Objectives

This chapter teaches you how to replace an existing rigid body with a flexible body using the Mesher provided in RecurDyn FFlex (Full Flex).



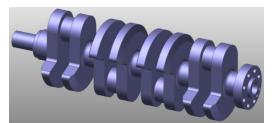
15 minutes

Creating the Crankshaft Mesh

To create the crankshaft mesh:

- 1. In **Assembly** mode, select the **Crankshaft Body**.
- 2. Right-click the screen to display the context menu, and then click **Mesh**.

As shown below, **Mesh** mode only displays the **crankshaft body**.



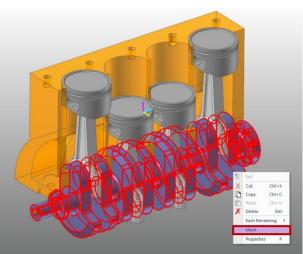


3. On the **Mesher** tab, in the **Mesher** group, click **Assist**.

The **Assist Modeling** dialog box appears.

4. Select the **Preserve Constraint** option.

The joints connected to the crankshaft body appear in the **Assist Modeling** dialog box, as shown in the image to the right.



Name	FDR	Geometry	Add/Remove	Sel.
RevJoint10	~	ImportedSolid1.Face9	Gr	•
RevJoint2	~	ImportedSolid1.Face13	Gr	~
RevJoint3		ImportedSolid1.Face98	Gr	I
RevJoint4	Image: A start of the start	ImportedSolid1.Face66	Gr	I
RevJoint5		ImportedSolid1.Face33,	Gr	I
Rev_Crankshaft		ImportedSolid1.Face140	Gr	
		et & Preserve Contact Geometry	Add/Remove	Geo
] Pre-Create Patch(Name	Line) S Set	et & Preserve Contact Geometry	Add/Remove	Geo
		1	Add/Remove	Geo

Tip: The **Geometries** in the **Preserve Constraint** are applied to the **Assist Modeling** dialog automatically after the user selects the **Target Body**. However, following behind steps are recommended to the user to produce accurate results, especially in the tutorial.

5. In the **Assist Modeling** dialog box, select all the checkboxes in the **FDR** and **Sel**. columns.

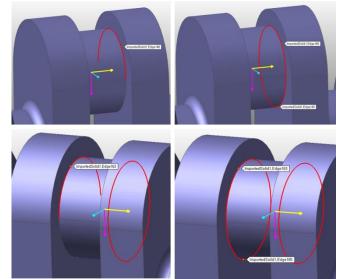
The checkboxes in the FDR column specify whether to generate FDR (Force Distributed Rigid) elements after creating the mesh. The checkboxes in the Sel. column specify whether to maintain the previous joints and forces. For this tutorial, you must select all the checkboxes. (**Tip**: If you have difficulty selecting entities, use the **Select List** function of the right-click popup menu to select it.)

 In the Assist Modeling dialog box, in the Add/Remove column to the right of RevJoint2, click the Gr button. Then, click the appropriate geometries around RevJoint2 to set them as slave nodes.

The edges around the master node will become the slave nodes for **RevJoint2**.

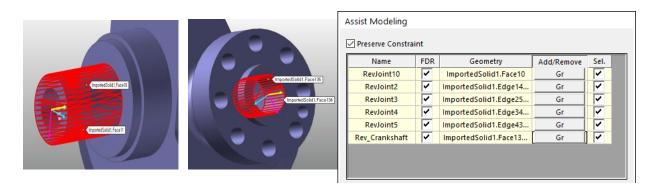
 Select all four edges of the **RevJoint2** master node. Then, right-click the node and click **Finish Operation** in the context menu.

> **Tip**: Clicking the Gr button displays the markers for the master node. By using one, you can see the areas where the slave nodes should be created easily.



Tip: To display the tooltip for the selected edge at the correct location, as shown above, click the Snap to Grid **The selected bar**.

 Repeat the procedure described above to create slave nodes for RevJoint3, RevJoint4, and RevJoint5. However, RevJoint10 and Rev_Crankshaft only require you to select the two surfaces shown below.



9. Confirm that the correct edges are selected for each geometry item, as shown in the figure to the right, and then click **OK**.



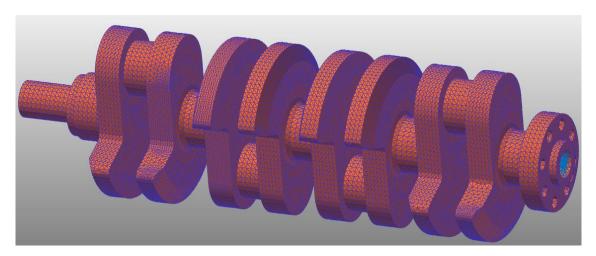
10. On the **Mesher** tab, in the **Mesher** group, click **Advanced**.

Advanced Mesh									
Target Geometry	ImportedSolid1	Gr	Target Face & Type I	nformation ——					
Mesh Type	Solid4(Tetra4)	-	Formal Type	All		•			
Property	PSolid1	Р	Name		Ту	pe		~	
Mesh Option			Face33		4-Si	ded		~	Ξ
	4		Face35		4-Si	ded		~	
Avg. Element Size			Face42		4-Si	ded		~	
Min Element Size	2		Face46		4-Si	ded		~	
			Face55		4-Si	ded		~	
_	tout Simple Pattern								
Structured Out	tput Simple Pattern		Face57		4-51	ded 			-
Include Assist			4 Sided More Type			•••		÷-	
		eview	4 Sided More Type	1st Edge (F	Red)	· · · 3rd Edge (l	and the second se		
_	Modeling Pr	eview	4 Sided More Type Name Face37	1st Edge (F Edge424	Red) Gr	 3rd Edge (I Edge428	Gr		
_	Modeling Pr	eview	4 Sided More Type Name Face37 Face39	1st Edge (F	Red) Gr Gr	3rd Edge (f Edge428 Edge427	Gr Gr	<u> </u>	
_	Modeling Pr	eview	4 Sided More Type Name Face37	1st Edge (F Edge424	Red) Gr	 3rd Edge (I Edge428	Gr	S	
	Modeling Pr	eview	4 Sided More Type Name Face37 Face39	1st Edge (F Edge424 Edge432	Red) Gr Gr	3rd Edge (f Edge428 Edge427	Gr Gr Gr Gr	<u>।</u>	
	Modeling Pr	eview	A Sided More Type Name Face37 Face39 Face44	1st Edge (F Edge424 Edge432 Edge417	Red) Gr Gr Gr	3rd Edge (l Edge428 Edge427 Edge405	Gr Gr Gr	<u> </u>	
	Modeling Pr	eview	A Sided More Type Name Face37 Face39 Face44 Face53	1st Edge (F Edge424 Edge432 Edge417 Edge366	Red) Gr Gr Gr Gr	3rd Edge (l Edge428 Edge427 Edge405 Edge394	Gr Gr Gr Gr	<u> </u>	
	Modeling Pr	eview	4 Sided More Type Name Face37 Face39 Face44 Face53 Face61	1st Edge (F Edge424 Edge432 Edge432 Edge417 Edge366 Edge358	Red) Gr Gr Gr Gr Gr	3rd Edge (l Edge428 Edge427 Edge405 Edge394 Edge373	Gr Gr Gr Gr Gr	<u> </u>	

The Advanced Mesh dialog box appears.

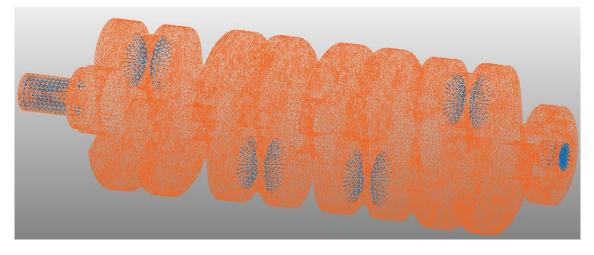
- 11. In the Advanced Mesh dialog box, perform the following steps:
 - In the Mesh Type dropdown menu, select Solid4(Tetra4).
 - In the Mesh Option group, set the Avg. Element Size to 4 and the Min Element Size to 2.

- To change all of the faces in the **4 Side More Type** list to **4 Side Type**, select the checkbox to the right of the **3rd Edge** heading.
- Select the Include Assist Modeling option.
- **Tip:** You must select this option to generate FDR elements automatically according to the conditions specified in the **Assist Modeling** dialog box.
- Click **Mesh**.
- After the meshing process is complete, click **Cancel**.



The mesh model for the crankshaft body is generated, as shown below.

To see information about the FDR elements generated using the Assist Modeling function, change the view mode to Wireframe. Then, in the Database pane, under Property Components, select PFDR_1 to see all the FDRs generated for the crankshaft, as shown below.



Tip: RecurDyn Mesh mode includes two auto-meshers: Mesh and Advanced Mesh. Mesh applies tessellation to the geometry in the GUI before generating the mesh. Advanced Mesh skips the tessellation stage altogether and generates the mesh directly. By doing so, Advanced Mesh minimizes the distortion of the CAD geometry. However, Advanced Mesh also increases the failure rate when generating the mesh.

To improve the mesh quality when using Advanced Mesh, change the faces in the 4-Side More Type list to 4-Side Type, as described above.



12. To see the stress results for a specific node in the crankshaft as a plot, click **Output** in the **FFlex Edit** group, on the **Mesher** group.

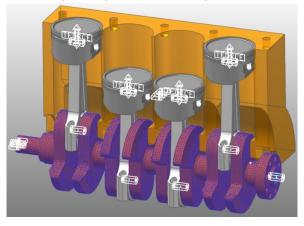
13. In the **Output** dialog box, perform the following steps:

Output [Current Unit : N/kg/n	nm/s/deg]
General Output	
Color	Automatic 🗸
Add/Remove	
Add/Remove (Continuous)	Tolerance (Degree) 45
	Check Reverse Direction
Add by Set	
Add by B.C.	
Add by Marker	
No. of Nodes 0	
	OK Cancel

- Click Add/Remove.
- In the Input Window of the Command Input Toolbar, type 438 (node number), and then press Enter.

MultiNode	* 438	🗲 ÷
-----------	-------	-----

- Right-click the Working window to display the context menu, and then click Finish Operation.
- In the **Output** dialog box, click **OK** to create **Output1** in the **Database** window.
- 14. Perform either of the following steps to return to the parent mode:
 - Right-click the Working window, and from the menu that appears, click **Exit**.
 - From the **Mesher** group in the **Mesher** tab, click **Exit**.
 - The existing crankshaft rigid body is replaced by the crankshaft flexible body (Crankshaft_FE), as in the image below. However, the previous joints and forces remain the same. To see the icons, click Icon Control in the toolbar to select all of the icons in the model pane. (Click All Icons)



Conducting the Dynamic Analysis on the FFlex Body and Reviewing the Results

To conduct the dynamic analysis on the FFlex body:

- 1. Save the model as **RD_RFlexGen_4Cyl_Engine_FFlex.rdyn**.
- 2. Select **Crankshaft_FE**, right-click the screen to open the context menu, and then click **Properties**.

The **Properties of Crankshaft_FE** dialog box appears.

- 3. In the dialog box, on the **Body** tab, click **Initial Velocity**.
- 4. In the **Body Initial Velocity** dialog box, perform the following steps:
 - Select **X** in the **Rotational Velocity** group, and then click the **PV** button.
 - Select Initial_Velocity.
 - In the **Reference Marker** field, click the **M** button.
 - In the Database window, under Ground, click the Inertia Marker and drag it to the Navigation Target window. Click Close to close the dialog window.

Body Filence Bodies Filence Bodies Filence Bodies Filence Bodies Filence Bodies Filence Bodies F	Body Initial Velocity Prior Connecting Parameters Nade Scope Initial Velocity V222329980711662 V 0. V V 0. <		Properties of Cra	nkshaft_FE [Cur	rent Unit : N/kg/mm/s/deg]		
Nass 17.2822483388 vv 22.253998711562 vv 22.4823765365353 vv 22.48237653659535 vv 22.48237653659535 vv 22.48237653659535 vv 22.48237653659535 vv 22.0 vv 0.	Nass 122223990071192 ody Initial Velocity Y Y 0.						
bit 30367.6439613263 Ivy 22.2253906711682 bit 30367.6433613263 ivy 22.2523906711682 ivy 249994.011679003 ivy 22.4220014406221 ivx 0. Ivy 24892796166256, 115.735591552765, 22 ivx 0. Ivy 1 24892796166256, 115.735591552765, 22 ivx 0. Ivy 1 1 ivx 0. <td< th=""><th>addy Initial Velocity vs 2255/6435813263 vy 222558272653 y 2 2 2 2 2 2 y 0. P Initial Velocity initial Velocity initial Velocity y 0. P P Initial Velocity initial Velocity y 0. P P Initial Velocity initial Velocity y 0. P Parametric Value List Notational Velocity P P Parametric Values x initial Velocity P P Parametric Values x initial Velocity P P Parametric Values x initial Velocity P P P y 0. P P P P x initial Velocity P P P P</th><th></th><th>Body FEI</th><th>nto. Connec</th><th>ting Parameters Node Scop</th><th></th><th></th></td<>	addy Initial Velocity vs 2255/6435813263 vy 222558272653 y 2 2 2 2 2 2 y 0. P Initial Velocity initial Velocity initial Velocity y 0. P P Initial Velocity initial Velocity y 0. P P Initial Velocity initial Velocity y 0. P Parametric Value List Notational Velocity P P Parametric Values x initial Velocity P P Parametric Values x initial Velocity P P Parametric Values x initial Velocity P P P y 0. P P P P x initial Velocity P P P P		Body FEI	nto. Connec	ting Parameters Node Scop		
bdy Initial Velocity Translational Velocity Z 0. Reference Marker Crankshaft_FE.CM M Reference Marker Crankshaft_FE.CM	body Initial Velocity Translational Velocity Translational Velocity Tanslational Velocity Tanslational Velocity Tanslational Velocity To 0. To 0		Mass 17.29232	4833988			
Dody Initial Velocity Translational Velocity Translational Velocity Productional Velo	Dody Initial Velocity Imailational Velocity Imailational Velocity Imailational Velocity Imailatinal Velocity Imailational Velocity		lxx 30367.64	33613263	lxy -22.2529908711662		
Translational Velocity X 0. Y 0.	Translational Velocity rz 2824483763369533 tx 22.822558272633 Istion of Mass Center 1.26442798166258, -115.73591552765, 27 Initial Velocity rational Velocity rational Velocity Initial Velocity Proprint Parametric Value List Initial Velocity Proprint Parametric Value Stational Velocity Initial Velocity Proprint Parametric Value Stational Velocity Extendinative Extendinati	ody Initial Velocity	lyy 249994.0	31679023	lyz 2.41220014406221		
X 0. PV Y 0. PV Y 0. PV Y 0. PV Initial Velocity Pv X Initial Velocity Pv Initial Velocity Pv Y 0. Pv Reference Marker Crankshaft_FE.CM M Reterence Marker Crankshaft_FE.CM M Initial Velocity Pv Pv Y 0. Pv Z 0. Pv Reference Marker Crankshaft_FE.CM M Close Value Comment Body Initial Velocity Fx Velocity Initial Velocity Pv Pv V 0. Pv Reference Marker Crankshaft_FE.CM M V 0. Pv V 0.	X 0. PV Y 0. PV Y 0. PV Z 0. PV Reference Marker Crankshaft_FE.CM M Retational Velocity Pi Name Value X Initial Velocity Pi Name Value X Initial Velocity Pi Name Value Comment Z 0. PV Parametric Value List Factorial Velocity E Y 0. PV Pi Name Value Comment Z Z 0. PV Propie Name Value Ex_End_Time Ex_End_Tim		Izz 262448.9	76369853	Izx -22.8822558272853		
V 0. P Z 0. P Reference Marker Crankshaft_FE.CM M Rotational Velocity P Z 0. P V 0. P Z 0. P V 0. P Z 0. P V 0. P Z 0. P <t< td=""><td>Y 0. Pr Z 0. Pr Indue in P Parametric Value List Reference Marker Comment X Initial Velocity Y 0. Z 0. Prove 2 Y 0. Y 0. Close X Navigational Velocity Ex V 0. V</td><td>_</td><td>sition of Mass</td><td>Center -1.24542</td><td>2798166258 -115.735591552765 2</td><td></td><td></td></t<>	Y 0. Pr Z 0. Pr Indue in P Parametric Value List Reference Marker Comment X Initial Velocity Y 0. Z 0. Prove 2 Y 0. Y 0. Close X Navigational Velocity Ex V 0. V	_	sition of Mass	Center -1.24542	2798166258 -115.735591552765 2		
Z 0. Reference Marker Crankshaft_FE.CM M No P Parametric Value List Parametric Values No P PM 3000. E No No P PM 3000. E No Permetric Values No P PM 3000. E No No P PM 3000. E No Navigation P P Navigation Navigation P P Navigation P Context Navigation P P P Navigation P P P Navigation P P P Navigation P P P Narker2 Narker2 <t< td=""><td>Z 0. Pr Reference Marker Crankshaft_FE.CM M Rotational Velocity Pr X Initial_Velocity Pr Y 0. Pr Z 0. Pr Y 0. Pr Z 0. Pr Reference Marker Crankshaft_FE.CM M Navigation rcg X Navigation rcg Y Q Pr Pr Markers Pr Bodies Groups Pretreine Marker Groups Pr 0. Pr Z 0. Pr Reference Marker Groups Pretreine Markers Y 0. Pr Z Pr Pr Markers Pr Markers Y Pr Pr</td><td></td><td></td><td></td><td></td><td></td><td></td></t<>	Z 0. Pr Reference Marker Crankshaft_FE.CM M Rotational Velocity Pr X Initial_Velocity Pr Y 0. Pr Z 0. Pr Y 0. Pr Z 0. Pr Reference Marker Crankshaft_FE.CM M Navigation rcg X Navigation rcg Y Q Pr Pr Markers Pr Bodies Groups Pretreine Marker Groups Pr 0. Pr Z 0. Pr Reference Marker Groups Pretreine Markers Y 0. Pr Z Pr Pr Markers Pr Markers Y Pr Pr						
Image: Construction of the construle of the construction of the construction of the const	Reference Marker Crankshaft_FE.CM M Rotational Velocity Z 0. Body initial Velocity Close Body initial Velocity Presentational Velocity Close Body initial Velocity Reference Marker Crankshaft_FE.CM M Navigation Fr.g., X Drop here from database Drop here from database Bodies Ground Markers Connect_Rod_1					J	
Reference Marker Crankshaft_FE.CM M Rotational Velocity X Initial_Velocity Z 0. Pv Reference Marker Crankshaft_FE.CM Close Body Initial Velocity Tanslational Velocity Cose Body Initial Velocity Tanslational Velocity X 0. Pv Reference Marker Crankshaft_FE.CM M Reference Marker Crankshaft_FE.CM Reference Marker R	Reference Marker Crankshaft_FE.CM M Rotational Velocity X initial_Velocity X 0. V V 0. V 0 V 0	Z 0. P		Parametric V	/alue List		
Rotational Velocity I V Reference Marker Crankshaft_FE.CM M Navigation V 0. Pr V Initial_Velocity E Initial_Velocity Body Initial Velocity Image: Second S	Rotational Velocity ZX Initial_Velocity PV 0. PV C 0. PV Reference Marker Crankshaft_FE.CM M Close Body Initial Velocity Translational Velocity Reference Marker Crankshaft_FE.CM M Navigation r.g × Drop here from database ! Prop here from database !	Reference Marker Crankshaft_FE.CM N		Parametric Va	lues		
Initial_Velocity Image: space	X initial_Velocity Y 0. Y 0. PV Z 0. PV Z 0. PV Z 0. PV Reference Marker Close Body Initial Velocity Tanslational Velocity Tanslational Velocity Tanslational Velocity Reference Marker Crankshaft_FE.CM Marker V 0. Prop here from database!			No DP	Name	Value	Comment
Y 0. Y 0. Y 0. Y 0. Y 0. Q 0. PN Reference Marker Close Body Initial Velocity X 0. Y 0. Y <tr< td=""><td>Y 0. PY Z 0. PY Z 0. PY Z 0. PY Close Body Initial Velocity Imanuational Velocity</td><td></td><td>1</td><td>1 🗸</td><td></td><td>3000. E</td><td></td></tr<>	Y 0. PY Z 0. PY Z 0. PY Z 0. PY Close Body Initial Velocity Imanuational Velocity		1	1 🗸		3000. E	
Initial_Velocity Body Initial Velocity Body Initial Velocity Close Body Initial Velocity Image: Close Database V 0. V 0. Prop here from database ! V 0. Prop here from database ! Initial_Velocity Initial_Velocity Initial_Velocity V 0. Prop here from database ! Initial_Velocity Image: Initial_Velocity	Initial Velocity Body Initial Velocity Body Initial Velocity Image: Stational Velocity Navigation Navigation V 0. V 1. Navigation V 0. V 1. Navigation V 1. Navigation V 1. Navigation V 1. Navigation V 1. V 1. V 1. V 1. V 2. V 1. V 2. V 1. V 2. V 1. V 2. V 3. V 4. V 4. V 4. V 4. V 5. V 6. V 7.				-		
Z 0. Reference Marker Close Body Initial Velocity Y 0. Prop V Navigation Y 0. Prop Navigation Y 0. Prop Navigation Y 0. Prop Navigation Y 0. Prop	Z 0. Pr Reference Marker Crankshaft_FE.CM M Close Image: Close Image: Close Body Initial Velocity Image: Close Image: Close V 0. Pr V 0. Pr Reference Marker Crankshaft_FE.CM Image: Close Navigation Cg X Drop here from database ! Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close Image: Close<	✓Y 0.					
Tanslational Velocity X 0. Y 0. Prevent Navigation recg × Database A CylinderEngine A CylinderEngine Y 0. Prevent Navigation recg × Drop here from database ! Bodies Image: Crankshaft_FE.CM Markers Y 0. Prevent X Initial Velocity Y 0. Y 0. Y 0. Prevent X Initial Velocity Y 0.	Translational Velocity X V Z 0. PP Z 0. PP V Database Database V Database Database V Drop here from database Database V O V O V O V P Reference Marker Ground.InertiaMarker W O Database V Drop here from database Database V O V O V Directinduction V Directinduction V Directinduction Directinduction Directinduction Directinduction						
X 0. P V 0. P Z 0. P Reference Marker Crankshaft_FE.CM M Navigation Prop. Drop here from database ! Bodies Bodies Ground Markers Markers X Intitial Velocity V 0. V P Z 0. P Reference	X 0. P Y 0. P Z 0. P Reference Marker Crankshaft_FE.CM M Rotational Velocity P Y 0. P Z 0. P Reference Marker Ground.InertiaMarker Marker2 Marker2 RFlex_Reference B Connect_Rod_1		1		_ ~		
Image: Sector of the sector	Image: Sectional Velocity Image: Sectional Velocity				* ^		
Z 0. Pr Reterence Marker Crankshaft_FE.CM M Rotational Velocity Pr V Y 0. Z 0. Pr V Reterence	Z 0. Pr Reference Marker Crankshaft_FE.CM M N Initial_Velocity Pr Z 0. Pr Z 0. Pr Reference Marker Ground.inertiaMarker M Marker2 Connect_Rod_1	✓ Y 0.		Navigat	tion pra ×		igine
Reference Marker Crankshaft_FE.CM M Rotational Velocity P V x Initial_Velocity V y 0. V z 0. P V z 0. P Reference	Reference Marker Crankshaft_FE.CM M Rotational Velocity Y 0. PV Reference Marker Ground.InertiaMarker M Connect_Rod_1	Z 0. Pv					
Rotational Velocity Pr Image: State of the	Rotational Velocity X Initial_Velocity Pv Y 0. Pv Z 0. Pv Reference Marker Ground.InertiaMarker Markers Image: Connect_Rod_1 Image: Connect_Rod_1	Reference Marker Crankshaft_FE.CM M		Drop ne	re from database :		
Initial_Velocity Pv Vr 0. Vz 0. Pv KretiaMarker Kretz	Initial_Velocity Image: Constraint of the second secon						
Image: State of the state	Image: Second						
Image: Constraint of the second	Image: Structure Image: Structure <td></td> <td></td> <td></td> <td></td> <td></td> <td>🕹 InertiaMarker</td>						🕹 InertiaMarker
Kriex_Reference	Reference Marker Ground.InertiaMarker M						🖏 Marker2
Reference Marker Ground.InertiaMarker M							🞝 RFlex_Reference
	Close	Reference Marker Ground.InertiaMarker M				📄 🕕 🗊 Con	nect_Rod_1

The following figure shows the entire process.

- 5. In the **Properties of Crankshaft_FE** dialog box, on the **Connecting Parameters** tab, clear the **Use Force Connector** checkbox.
- 6. Click **OK** to close the dialog box.

Material Property of Connecting	tting Parameters Node Scop
Mass	-
	1.e-006 Pv
Moment of Inertia	
	1. Pv
Translational Damping Ratio	1.e-002
Translational Damping Ratio	1.e-002
Rotational Stiffness	1000000000.
Rotational Damping Ratio	1.e-002

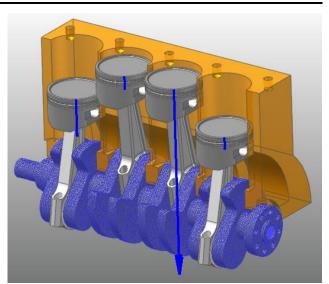
Tip: If joints and forces are connected to a flexible body, then the RecurDyn Solver applies dummy bodies to the markers for the joints and/or forces to improve the efficiency of the calculation. That is, dummy bodies are connected to the flexible body using arbitrary high-strength force elements. In this tutorial, however, we bind the joints completely without using force elements to minimize the effects of the force at the joints where the connecting rods, which are rigid bodies, connect to the crankshaft, which is a flexible body. Also, although FDR elements can be divided into force elements and constraint elements, we exclude the force elements in this tutorial. This allows us to simulate the behavior of the flexible body more realistically, but it takes more time to analyze the model.

7. On the **Analysis** tab, click **Dyn/Kin**.



 In the Dynamic/Kinematic Analysis dialog box, click Simulation to run the analysis without changing any settings.

> The analysis duration may differ depending on the specifications of your PC, but it generally takes about an hour. Once the analysis is finished, play the animation. The animation should show results that are very similar to those obtained when the crankshaft is a rigid body.



If you only want to learn how to use

RFlexGen, you may skip the rest of this chapter and proceed directly to chapter 4.

5
Contour
*

9. On the **FFlex** tab, click **Contour**.

10. When the **Contour** dialog box appears, perform the following steps:

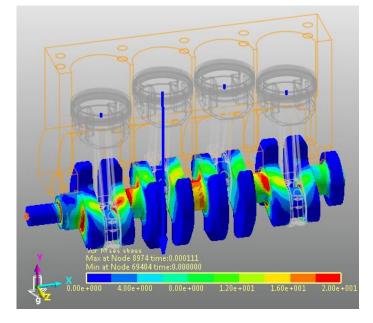
	-		Band Option					lode / Referen		
nimation Status	SMISES		Legend Type	Display	-	Sel	Body	Node ID	Sel	Or
pe	Stress	•	Location	Bottom	-		Crankshaf	1		
omponent	SMISES	-		Show Text Le	gend					
] Display Vector	25.00562731	48833	Band Level(10~50)		10					
🛄 Uniform	Simple		Style Option			•				D
Contact Surface	Only		Color Option	Edit		Conto	our Data Trace			
	ed contact surfa	ice	Colors	Spectrum	•	Sel	Body	Node	ID	J
Contact page	itches only		Style	Stepped	•					
in/Max Option -]	Text Color	Text Color	•					
pe User Defined			Exceed Max Color	Max Color	*					
	Calculation		Less than Min Color	Min Color	*		Add		Delete	_
Calc. Result		ined —				Conto	our Element S	et Selection -		
in	0	0				Sel	Body	Contour P	Part	
lax 102.	596	20								
		Carlo								
] Show Min/Max	Enable Log	y scale								
User Defined Ma	ax Color									
User Defined Mi	n Color		Mesh Lines	Line Color			Add		Delete	_

- In the Min/Max Option group, click Calculation.
- In the **Min/Max Option** group, set the **Type** to **User Defined**. .
- In the **Max** value text box, type **20**. .
- Click **OK** to close the dialog . box.



11. Click Animation Play.

The results should resemble those shown in the image to the right.





Generating the RFlex Body Using RFlexGen

Task Objectives

In this chapter, you will learn how to transform the FFlex body into an RFlex body using RFlexGen.



40 minutes

Running RFlexGen

To run RFlexGen:

1. On the **Flexible** tab, in the **Flex Interface** group, click **RFlexGen**.



Eigen Solver	Sunshine
	sunsnine
Input Data	
Mesh Data Type	Select Flexible Body
Flexible Body	Crankshaft_FE
Reference Frame	Crankshaft_FE M
Interface Nodes	Select Multi Node
No. of Interface Nodes	0
Interface Node IDs	
	ode in NodeSet Set DOFs
Case Control Settings —	
Case Control Settings	
-	Set DOFs
No. of Normal Modes	Set DOFs
No. of Normal Modes	Set DOFs
No. of Normal Modes No. of Saved Modes Units	Set DOFs 20 Auto Calculation
No. of Normal Modes No. of Saved Modes Units Mass	Set DOFs 20 Auto Calculation kilogram
No. of Normal Modes No. of Saved Modes Units Mass Length	Set DOFs 20 Auto Calculation kilogram millimeter
No. of Normal Modes No. of Saved Modes Units Mass Length	Set DOFs 20 Auto Calculation kilogram millimeter second

In this tutorial, **Crankshaft_FE** is the only FFlex body. Therefore, the **FFlex Body and Reference Frame** fields are automatically filled, as shown in the figure above.

2. In the **Interface Nodes** dropdown menu, select **Select Multi FDR**. Then, click **N** and select FDR elements with dragging the mouse on **Working Window**.

- Crankshatt_FE, SetNodel
- 3. click **Finish Operation** on a mouse Right Menu. that was created earlier by selecting the master nodes in the model pane.

- 4. In the **RFlexGen** dialog box, set the following options:
 - In the Case Control Settings group, set the No. of Normal Modes to 20.
 - In the **Units** group, maintain the default settings.
 - You do not need to change the units if the FFlex body was generated using **RecurDyn/Mesher**.
 - Click ..., and then specify the name and folder path of the **RFI** file to create.
 - When you specify the path of the file, be sure to include the file name as well.
 - Ensure that the settings match those in the figure to the right.

→ RFlexGen	×
Eigen Solver	Sunshine 💌
Input Data	
Mesh Data Type	Select Flexible Body
Flexible Body	Crankshaft_FE
Reference Frame	Crankshaft_FE M
Interface Nodes	Select Multi Node
No. of Interface Nodes	6
Interface Node IDs	70077, 70078, 70079, 70080, 70081, 700
Set DOFs for Each Node	e in NodeSet Set DOFs
Case Control Settings	
No. of Normal Modes	20
No. of Saved Modes	Auto Calculation 🔻 56
Units	
Mass	kilogram 💌
Length	millimeter
Time	second 💌
	Options
Output RFI File	
RFI File Name	E:\GT_Trunk\AddFile\Tutorial\Flexi
Export Input File	Execute Close

5. After adjusting the settings, click **Execute**.

Generating RFI file is startin Starting interface process Initial process CMS/Modal analysis	g
we	35.71 %

Wait as the Eigen solver performs the component mode synthesis (CMS) analysis.

Tip: Click the History (*.log) button to see details about the CMS analysis. If a problem occurs during the analysis, check the log file. Click the Results (*.dof) button to see the Eigen values for each mode after the analysis.

6. Once the analysis is complete, click **Close** to exit the **RFlexGen** progress window.

Creating the RFlex Body

To create the RFlex body:

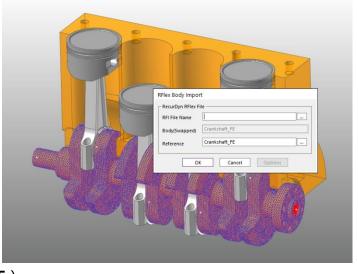


- 1. On the **Flexible** tab, in the **RFlex** group, click **Import RFI**.
- 2. Change the modeling option to **Body**.
- In the model pane, select Crankshaft_FE, as shown below.

The **RFlex Body Import** dialog box appears.

- 4. In the **RFlex Body Import** dialog box, perform the following:
 - Click ... to the right of the RFI File Name field.
 - Select the RFI file that you created in the Running RFlexGen chapter.
 - Click ... to the right of the Reference field, and then select Crankshaft_FE. (Ensure that the Reference field displays Crankshaft_FE.)

Point	×
Point	
Body	
Group	



Tip: You can use the name of FFlex body or RFlex body in the Reference field to automatically set the Body Reference Frame of the flexible body. Therefore, the above procedure sets the Body Reference Frame of the crankshaft FFlex body to the reference frame of the Flex body instead of the center marker (CM) of the body.

5. Ensure that the selected conditions match those shown in the figure to the right, and then click **OK**.

RFlex B	ody Impo	rt		
Recur	RecurDyn RFlex File			
RFI Fil	e Name	FlexGen_4Cyl_Engine_03\Crankshaft_FE.rfi		
Body(Swapped)	Crankshaft_FE		
Refere	ence	Crankshaft_FE		
	0	K Cancel Options		

6. Click Icon Control in the toolbar.

c.

- 7. In the **Icon Control** dialog box, select **All Icons**.
- 8. In the model pane, ensure that all of the joints that were applied to the crankshaft are still applied to **RFlexBody1**.
- 9. After verifying the joints, clear the checkboxes in the **Icon Control** dialog box.
- 10. In the **Database** window, double-click **RFlex** body to open the **Property** dialog box.
- 11. In the **Property** dialog box, perform the following:
 - a. Click 7th mode, as shown in the figure to the right.

You can also observe the model in other

modes by performing the above steps and

- b. Click Play to observe the behavior of the selected mode. The animation shown in the following figure appears so that you can observe the behavior of the mode.
- <image>

Seq	Sel	Freq.	Damping Ratio		
1	Γ	0.00	1.e-002		
2		0.00	1.e-002		
3		0.00	1.e-002		
4		0.00	1.e-002		
5	\square	0.00	1.e-002		
6		0.00	9.999999999999998e-003		
7	~	397.66	0.1		
8		544.90	0.1		
		Animation	O Damping Options		
Prev	tion - Seq	7 Frequency Play Next	(Hz) 397.66 Show Undeformed		
nima lode Prev	tion - Seq 7	7 Frequency Play Next	(Hz) 397.66 Show Undeformed		
nima lode Prev cale F	tion - Seq	7 Frequency Play Next	(Hz) 397.66 Show Undeformed		
nima lode Prev cale F	tion - Seq 7 	7 Frequency Play Next	(Hz) 397.66 Show Undeformed		

Conducting a Dynamic Analysis on the RFlex Body and Reviewing the Results

To conduct a dynamic analysis on the RFlex body:

1. Select **RFlexBody1**, right-click it to display the context menu, and then click **Properties**.

The **Properties of RFlexBody1** dialog box appears.

- 2. In the **Properties of RFlexBody1** dialog box, on the **Body** tab, click **Initial Velocity**.
- 3. In the **Body Initial Velocity** dialog box, perform the following steps:
 - Select **X** in the **Rotational Velocity** group, and then click the **PV** button.
 - Select Initial_Velocity.
 - In the **Reference Marker** field, click the **M** button.
 - In the Database window, under Ground, click the Inertia Marker and drag it to the Navigation Target window. Click Close to close the dialog window.
 - The following figure shows the entire process.

	Properties of Crankshaft_FE [Current Unit : N/kg/mm/s/deg]			/s/deg]			
	General Graphic Property Origin & Orientation			rientation			
	Body	FEInfo.	RFlex No	de Scope			
	Mass 17.3716938643466						
		881.2256508931	Ixy 3.581740210171	135e-002			
Body Initial Velocity	×	93.657997228	lyz 4.578322874294				
		59.062440403	Izx 0.289630668979				
Translational Velocity							
□ X 0.	Pv	Mass Center 0.371	361874022, -115.741374	4278685, 22			
□Y 0.	Pv	tic 📃	Initial Velocity				
Z 0.	Pv	Parametr	ic Value List				
Orientation		Parametri	c Values				
	m	No	DP Name		Value	Comment	l - 1
Rotational Velocity			RPM		00. E	Comment	Ā
x 0.	Pv		Degree		20. E		
Πγ [0.	Pv		End_Time	EX_En	d_Time E		
		4	Initial_Velocity	Ex_Ve	locity E		
Z 0.	Pv						
Orientation							
Close	Cancel						
	L						
	Scope	0	Cancel	Apply			
Body Initial Velocity	×	_		 X Databa 	se	V	7 F 🗙
Translational Velocity				× 40	/linderEngi	ine	
□ x _0.	Pv	Navi	gation Targ		Groups		
□ Y 0.	Pv	Drop	here from databa		Bodies		
□ Z 0.	Pv			Ŧ Ŧ	Groun	d	
Orientation Ground.InertiaMarker	M				-		
- Rotational Velocity					🚊 🦾 Ma		
X Initial_Velocity	Pv					InertiaMarker	
□ Y 0.	Pv					Marker2	
Z 0.	Pv					RFlex_Reference	e
Orientation Ground.InertiaMarker	M				🇊 Conne	ct_Rod_1	
Close Can	cel						
<u> </u>							

4. In the **Database** pane. double-click **RFlexBody1** to enter **RFlex Edit** mode.

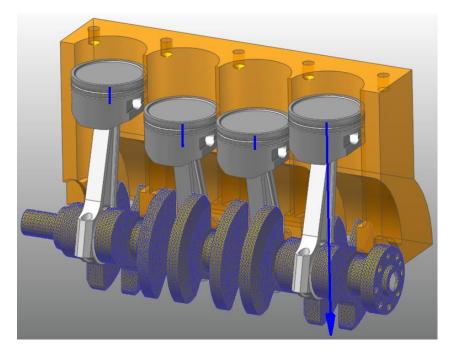


- To see the stress result for a specific node in RFlexBody1 as a plot, click **Output** in the **RFlex Edit** menu.
- In the **Output** dialog window, perform the following steps:
 - Click Add/Remove.
 - In the input field of the Command Input toolbar, type 438 (node number), and then press Enter.
 - Right-click the screen to display the context menu, and then click **Finish Operation**.
 - In the Output dialog box, click OK to create Output1 in the Database.

Output [Current Unit : N/kg/m	m/s/deg]
General Output	
Color	Automatic 👻
Add/Remove	
Add/Remove (Continuous)	Tolerance (Degree) 45
	Check Reverse Direction
Add by Set	
Add by B.C.	
Add by Marker	
No. of Nodes 0	
	OK Cancel

- 7. Right-click the screen to display the context menu, and then click **Exit**.
- 8. On the Analysis tab, click Dyn/Kin to open the Dynamic/Kinematic Analysis dialog box.
- 9. In this **Dynamic/Kinematic Analysis** dialog box, click **Simulation** to run the analysis without changing any settings.

After about 10 seconds of analysis, you can see that the animation is similar to the previous crankshaft body animation. The analysis duration differs depending on whether you use an FFlex body or RFlex body. Therefore, we recommend performing the analysis with the body that best suits your purposes to improve efficiency.



To display the distribution of stress in the RFlex body:

- Out.Regen
 - On the Flexible tab, in the FFlex group, click the Contour dropdown menu, and then click Output Regenerator.

The **Output File Regenerator** dialog box appears, as shown in the figure to the right.

- 2. In the **Information** pane, select all of the checkboxes in the **Sel** column.
- 3. Click **Output File Setting** in the bottom right corner of the dialog box.

Output File Regenerator						
Set same options for all RFlex bodies						
Displacement Data (*.rfa) 🛛 Strain / Stress Data (*.erd, *.srd)						
Shell Recovery Type for C	ontour					
Top			0	Bottom		
No. Bodies rfa	Displacement	erd, s	Strain	Stress	Strain Shell Type	Stress Shell Type
1 Crankshaft	Full	eru,s	Empty	Partial	strain shen type	stress shell type
Use Animation Configuration						
Output File Setting	1					Close

The **Output File Setting** dialog box appears, as shown in the figure to the right.

- 4. In the **Stress** group, click **Select All**.
- 5. Click Close.

Output File S	etting		
- Strain	etting		
	EY	EZ	
	_	_	
EXY	EYZ	EZX	
E1	E2	E3	Clear All
EINT	EMISES		Select All
- Stress			
∠ SX	SX	🗹 SZ	
⊡ sxy	SYZ	✓ szx	
<mark>⊻ </mark> 51	✓ S 2	🗹 S3	Clear All
SINT	SMISES		Select All
⊢ Data Precisio	n		
Single (Fl		Double	
0.000			
	Clos	e	

Tip: When creating an output file, only the Von-Mises, Sx, Sy, and Sz tests are included to reduce the file size. If you would like to view other results in contour mode, you must select other tests.

6. In the **Output File Regenerator** dialog box, click **Generate**.

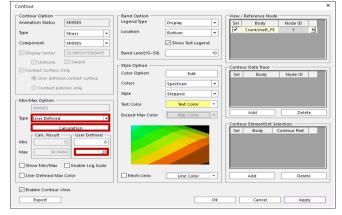
When the progress bar is full, the **Stress** column in the Information group changes from **Empty** to **Full**.

Making output files	
Output files for RFlexBody1@4CylinderEngine	Cancel

(**Tip**: If the output file was created using the default settings, then the Stress column changes to Partial rather than Full. This is because, out of the 11 stress tests available, only the Von-Mises, Sx, Sy, and Sz tests were conducted.)

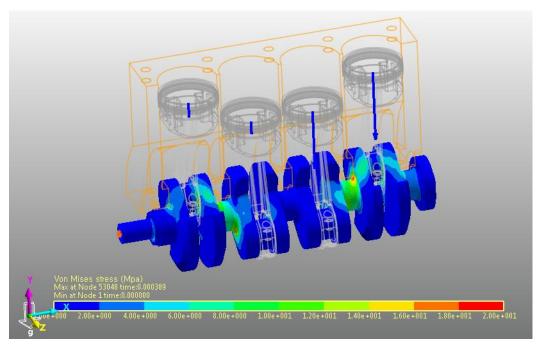


- 7. On the **Flexible** tab, in the **RFlex** group, click **Contour** to open the **Contour** dialog box.
- 8. In the **Contour** dialog box, perform the following:
 - In the Min/Max Option group, click Calculation.
 - In the Min/Max Option group, set the Type to User Defined.
 - In the Max value text box, type 20.
 - Click **OK** to close the dialog box.



9. Click Animation Play.

You should see the following results.



- 10. On the Analysis tab, click Dyn/Kin to open the Dynamic/Kinematic dialog box.
- 11. In the **Dynamic/Kinematic** dialog box, perform the following:
 - a. Select Output File Name.
 - b. In the Output File Name text box, type RFlexGen_20modes.
 - c. To see the animation while the analysis is being performed, select the Display Animation checkbox.

Dynamic/Kinematic Analysis					
General Parameter Initial Condition					
End Time	End_Time Pv				
Step	Step 720. Pv				
Plot Multiplier Step Factor	1. Pv				
🗹 Output File Name	RFlexGen_20modes				
Static Analysis					
Eigenvalue Analysis					
State Matrix					
Frequency Response Analysis	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					

Tip: Since the analysis was performed in the previous procedure, you do not need to perform the analysis again to see the simulation results. However, to see the stress results for a specific output node of the RFlex body using a plot (the next procedure), you must perform the analysis again. This is because the original analysis was performed on the RFlex body without a stress shape, so the results plot (*.rplt file) does not include the stress results. Therefore, to check the stress or strain results for a specific output node of the RFlex body, you must apply the stress or strain shape to the RFlex body before performing the analysis.

Running RFlexGen Again



To increase the number of modes and repeat the analysis, perform the following steps:

1. On the **Flexible** tab, in the **Flex Interface** group, click **RFlexGen**.

In the **Generate RFI** dialog box, you can see that **Flexible Body** and **Reference Frame** match the information for the previously created RFlex body, unlike when you opened the RFlexGen dialog box in the previous procedure.

- In the Interface Nodes dropdown menu, select Select Multi FDR. Then, click N and select FDR elements with dragging the mouse on Working Window.
- 3. click Finish Operation on a mouse Right Menu. that was created earlier by selecting the master nodes in the model pane.
- 4. In the **RFlexGen** dialog box, perform the following:
 - a. In the No. of **Normal Modes** field, type **100**.
 - b. In the **Units** group, maintain the default settings.
 - c. In the **Output RFI File** group, click ..., and then specify the name and folder path of the **RFI** file.

RFlexGen	×
Eigen Solver	Sunshine
_ Input Data	
Mesh Data Type	Select Flexible Body
Flexible Body	Crankshaft_FE 💌
Reference Frame	Crankshaft_FE M
Interface Nodes	Select Multi Node
No. of Interface Nodes	6
Interface Node IDs	70077, 70078, 70079, 70080, 70081, 700
User Defined DOFs for C	Computing Constraint Modes
	Set DOFs
Case Control Settings	
No. of Normal Modes	100
No. of Saved Modes	Auto Calculation 👻 136
Units	
Mass	kilogram 💌
Length	millimeter 💌
Time	second
	Options
Output RFI File	
RFI File Name	E:\GT_Trunk\AddFile\Tutorial\Flexi
Export Input File	Execute Close

5. After adjusting the settings, click **Execute**.

The Eigen solver performs **the** CMS (component mode synthesis) analysis, as shown in the figure to the right.

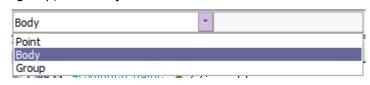
6. Once the analysis is complete, click **Close** to stop the **RFlexGen** process.

Progress	
Generating RFI file is starting Starting interface process Initial process CMS/Modal analysis	
n	28.57 %
Results (*.dof) History (*.log)	Close

Replacing an RFlex Body

To replace an RFlex body:

- 1. On the **Flexible** tab, in the **RFlex** group, click **Import RFI**.
- 2. Set the modeling option to **Body**.



3. In the model pane, select the **Crankshaft_FE**, as shown below.

The RFlex Body Import dialog box appears.

- 4. In the **RFlex Body Import** dialog box, perform the following:
 - a. Click ... to the right of the RFI File Name field, and then specify the path and name of the **RFI** file (as described in the Running **RFlexGen** chapter).
 - b. Click ... to the right of the Reference field, and then ensure that the name in the reference field has changed from Crankshaft_FE.CM to Crankshaft_FE.
- 5. Verify that the selected conditions match those shown in the figure to the right, and then click **OK**.

Canishar, F	RFlex Body Impor RecurDyn RFlex Fi RFI File Name Body(Swapped) Reference	
		Crankshaft_FE
	OK Cancel Options	

RFlex Body Import			
RFI File Name	runk\D\TU\Flexible\RFlexGen_Crankshaft\F		
Body(Swapped)	Crankshaft_FE		
Reference	Crankshaft_FE		
OK Cancel Options			

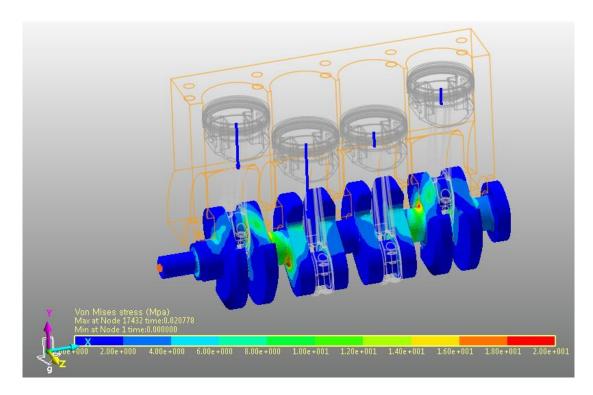
- 6. In the **Database** pane, double-click **Crankshaft_FE** to display the **Property** dialog box.
- 7. In the Property dialog box, perform the following
 - Select the **130th mode**, as shown in the figure to the right.
 - Click **Play** to observe the behavior of the selected mode.

Body File Current Unit : N/kg/mm/s/deg] General Graphic Property Origin & Orientation Body FEinfo. RFlex Node Scope
RecurDyn/Flex Input File Name U\Flexible\RFlexGen_Crankshaft\RFlexGen_Cr; Reassociate
Seq Set Freq. Damping Ratio 124 29438.09 1. 125 31315.24 1. 126 31315.24 1. 127 31422.15 1. 128 35006.96 1. 129 36415.25 1. 130 36004.16 1. 131 42256.45 1. Mode @ Animation O Damping Options Animation Damping Options Animation Scale Factor [. Mass Invariant . .
Partial © Full Scope OK Cancel Apply

Conducting a Dynamic Analysis on the RFlex Body and Reviewing the Results

To conduct a dynamic analysis on the RFlex body:

Repeat the steps on pages 25 – 27





Analyzing and Reviewing the Results

Task Objectives

In this chapter, we will analyze the results obtained from performing multi flexible body dynamics (MFBD) analyses on models containing FFlex or RFlex bodies and describe the benefits of using the RFlexGen function.



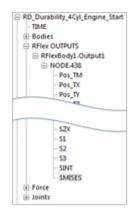
5 minutes

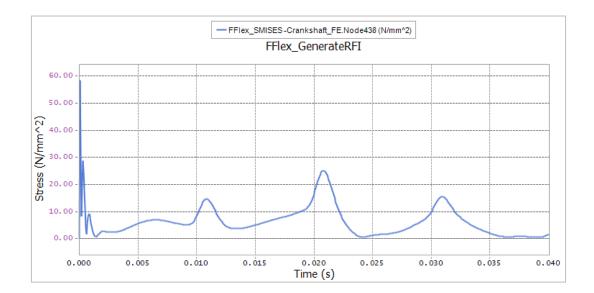
Analyzing the Dynamic Analysis Results

Analysis of the results obtained from an FFlex body that was generated using the Mesher function

Earlier in this tutorial, we changed the crankshaft rigid body into a flexible body using the FFlex/Mesher and then performed a dynamic analysis. The flexible body included about 50,000 nodes (about 0.15 million degrees of freedom). Because of this, it took about 1 hour to perform the dynamic analysis. (The actual time may differ depending on the specifications of your PC.)

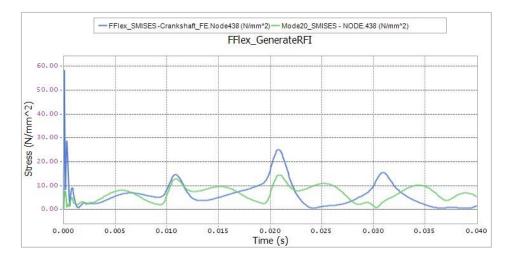
The following figure shows the results of the Von-Mises Stress test performed on the nodes that were included in the area of interest in the analysis results. (Since we defined the Output in the earlier steps, the test results were stored as a plot.)





Comparison of these results with the results obtained using an RFlex body that was generated using RFlexGen

The following figure compares the results of the Von-Mises Stress test using an FFlex body with those of an RFlex body. RFlexGen was used to transform the FFlex body into an RFlex body. The Von-Mises Stress test was performed on the nodes included in the area of interest in the analysis results.

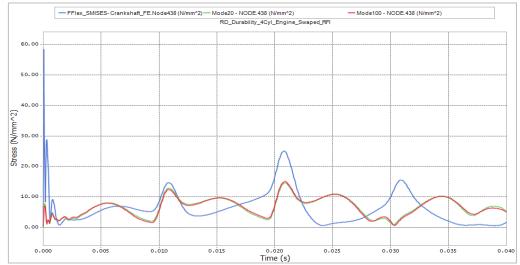


For the crankshaft used in this tutorial, we do not need to consider its contact with other bodies and the crankshaft is only bound by joints. Therefore, we don't need an FFlex body to simulate the flexibility of the crankshaft because an RFlex body is sufficient for that purpose. That is, in an analysis situation that has many nodes for the flexible body and does not require any contact conditions, it is more efficient to use an RFlex body than an FFlex body if you require faster feedback more than accurate results.

As we can see in the above figure, there is not much difference between the results obtained with an FFlex body and the results obtained with an RFlex body. Considering the fact that the analysis using the RFlex body took only about 10 seconds while the analysis using the FFlex body took about 1 hour, you can see that using an RFlex body drastically reduces the time required for analysis while producing almost the same results as the FFlex body.

Comparison of the results after increasing the number of normal modes

The following figure shows the results after quintupling the number of normal modes for the RFlex body.



As you can see from the graph above, increasing the number of normal modes has little effect on the analysis results. From this, we can conclude that using a small number of normal modes in the engine system used for this tutorial still produces fairly accurate results.

Pros and cons of using an RFlex body generated by RFlexGen

This tutorial reveals the following pros and cons for using FFlex bodies and RFlex bodies.

Analyses using FFlex bodies produce more accurate results than those using RFlex bodies. This is because such analyses include the non-linear behavior of the analysis targets, including targets that have several contact conditions. In these cases, an FFlex body allows you to obtain more accurate results. However, the analysis takes longer if the number of nodes constituting the FFlex body or the number of non-linear properties of the target increases.

One benefit of using RFlex bodies is that they significantly reduce the analysis time. This is because the analysis uses the number of normal modes, instead of the number of total nodes, as the degree of freedom. Also, if the target behavior does not include non-linear properties that cause large scale transformations, then you can use an RFlex body to simulate the properties of a flexible body. However, in an MFBD model that requires an analysis of non-linear behavior, using an RFlex body instead of an FFlex body may reduce the accuracy of the analysis results. If you don't need to simulate non-linear properties, then using an RFlex body can produce results much more quickly than using an FFlex body. This allows you to apply the analysis results in the design of the actual product much more quickly.

Thanks for participating in this tutorial