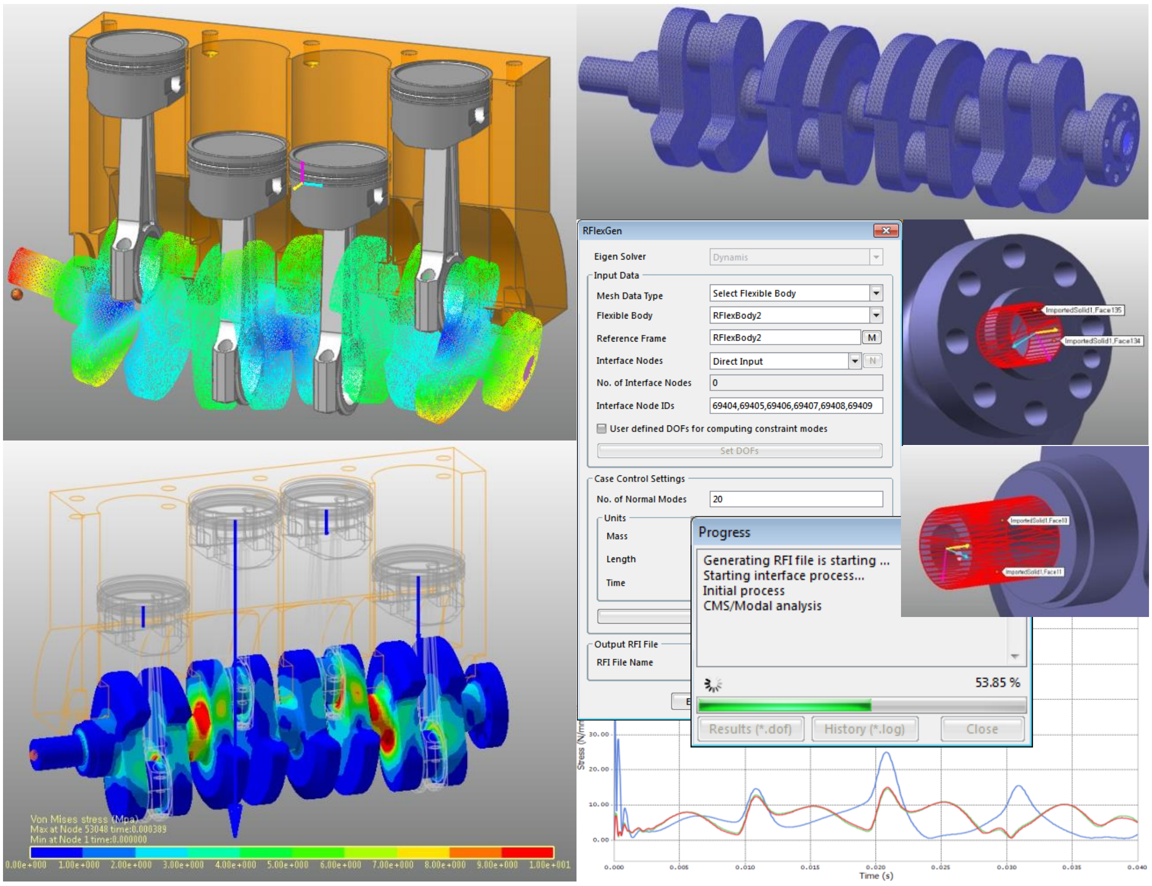
RECURDYN

RFlexGen Crankshaft Tutorial (RFlexGen)



FunctionBay

Edition Note

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

Table of Contents

Chapter

1

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**

Estimated Time to Complete

This tutorial takes approximately 70 minutes to complete.

Chapter

2

Opening the Initial Model

Task Objectives

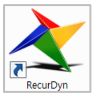
This chapter teaches you how to open the initial model, run a simulation, and observe how the 4-cylinder engine model operates.

Estimated Time to Complete

10 minutes

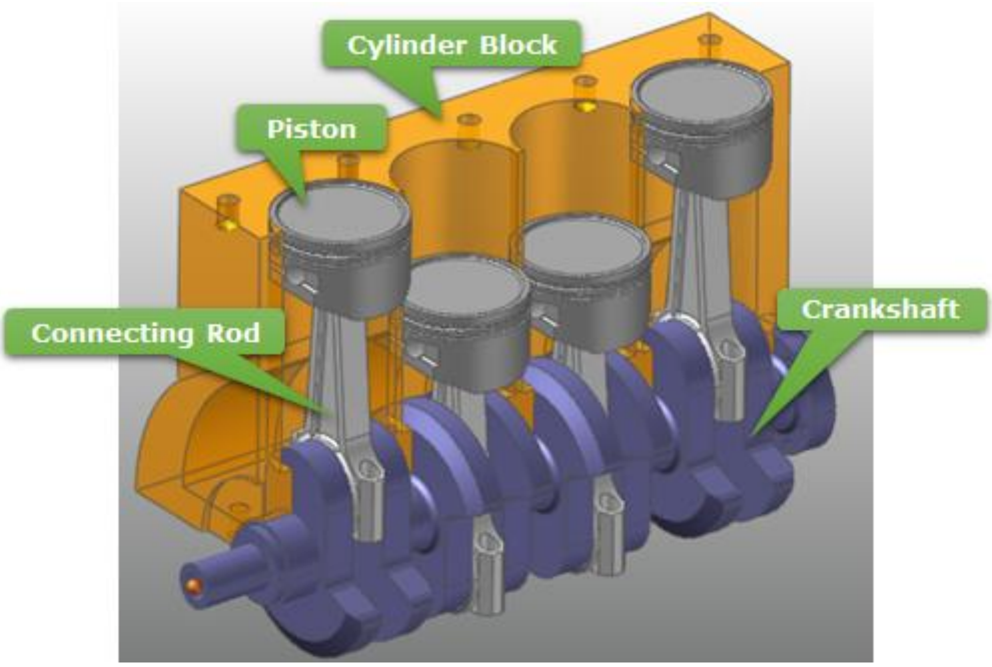
Opening the RecurDyn Model

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**.

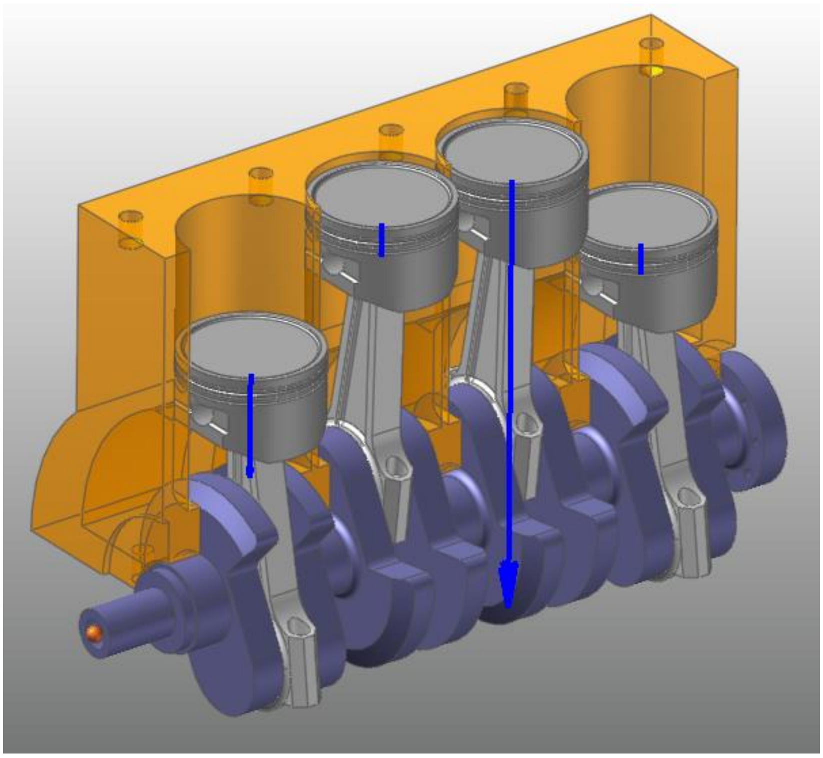
   The **Dynamic/Kinematic Analysis** dialog box appears.

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 ➔ Piston_3 ➔ Piston_4 ➔ 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➔compression➔combustion➔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.



Chapter 3

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).

Estimated Time to Complete

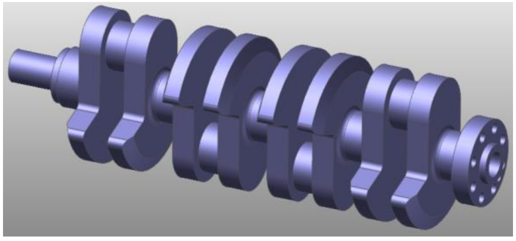
15 minutes

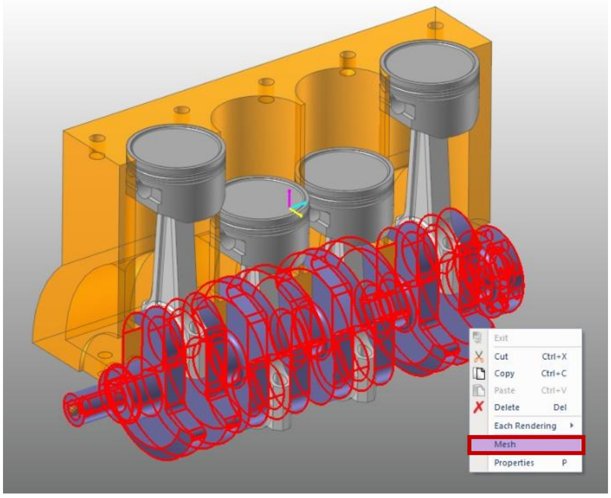
Creating the Crankshaft Mesh

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**.

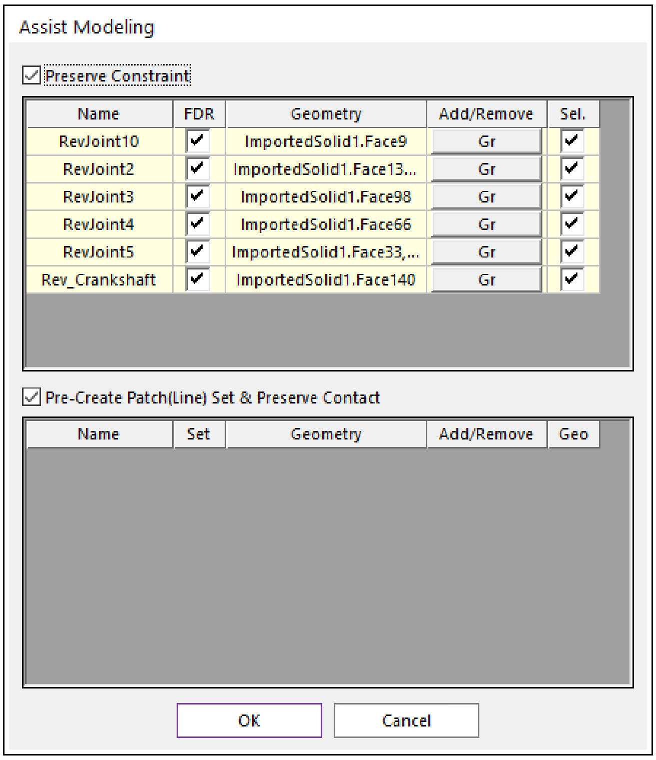




3. On the **Mesher** tab, in the **Mesher** group, click **Assist**.

   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.



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.

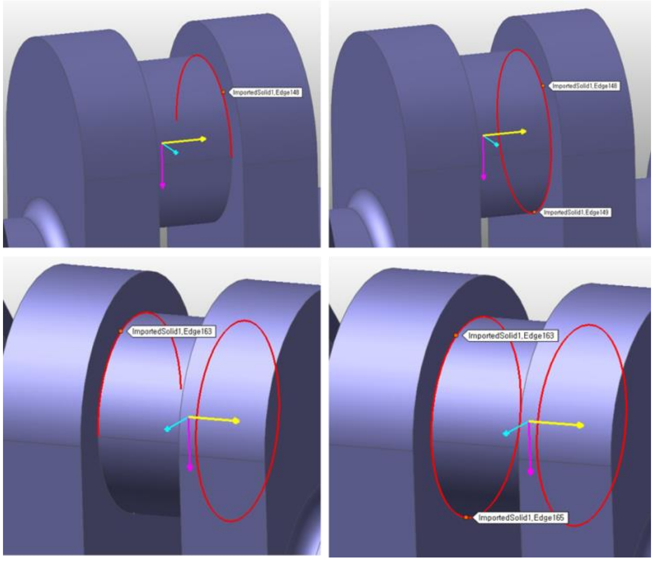
The checkboxes in the FDR column specify whether to generate FDR (Force Distributed Rigid) elements after creating the mesh. The checkboxes in the Sel. column specify whether to maintain the previous joints and forces. For this tutorial, you must select all the checkboxes. (**Tip**: If you have difficulty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

6. 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**.

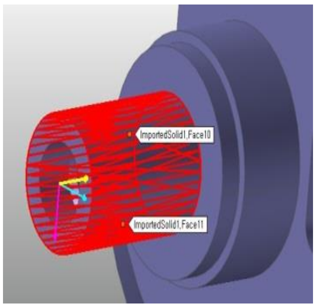
7. Select all four edges of the **RevJoint2** master node. Then, right-click the node and click **Finish Operation** in the context menu.

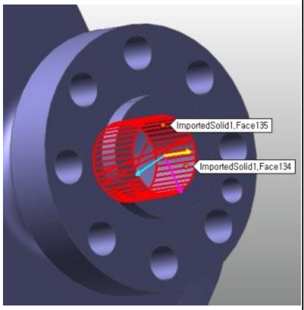
   Tip: Clicking the Gr button displays the markers for the master node. By using one, you can see the areas where the slave nodes should be created easily.

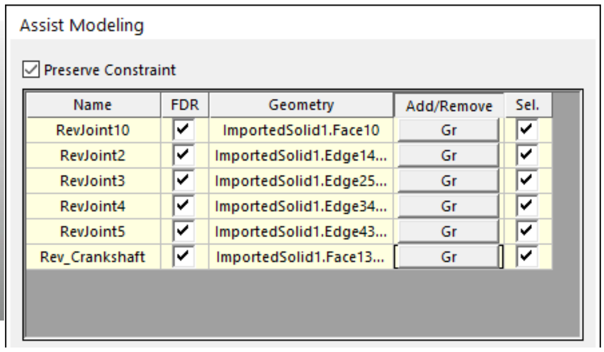


Tip: To display the tooltip for the selected edge at the correct location, as shown above, click the Snap to Grid  button in the tool bar.

8. 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.



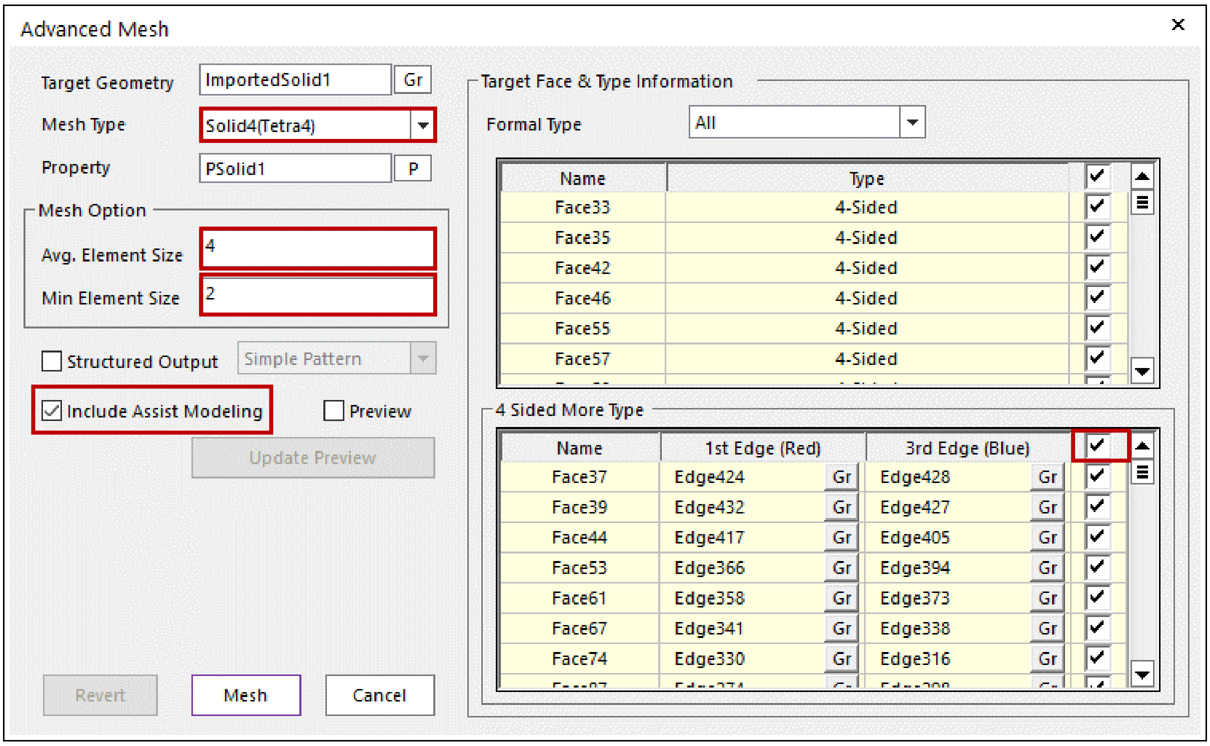




9. Confirm that the correct edges are selected for each geometry item, as shown in the figure to the right, and then click **OK**.

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

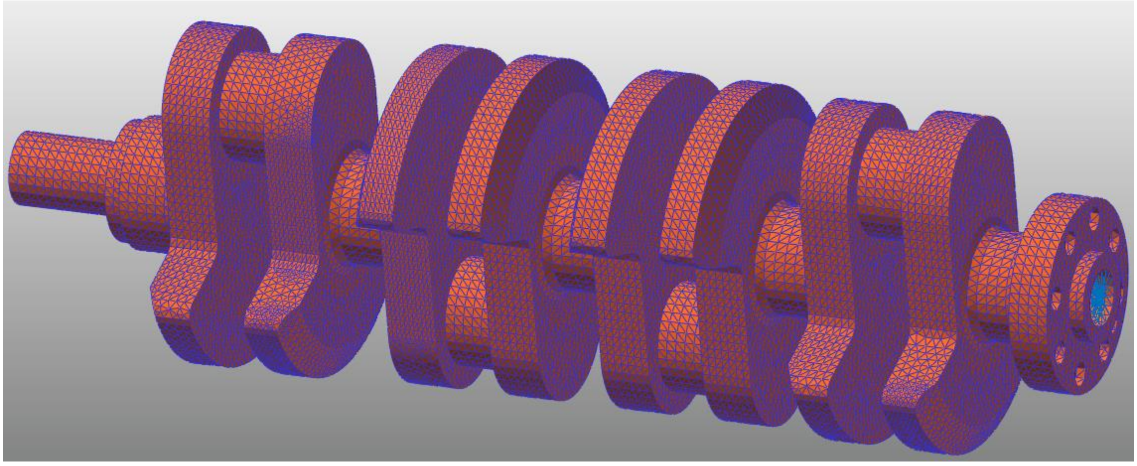
The **Advanced Mesh** dialog box appears.



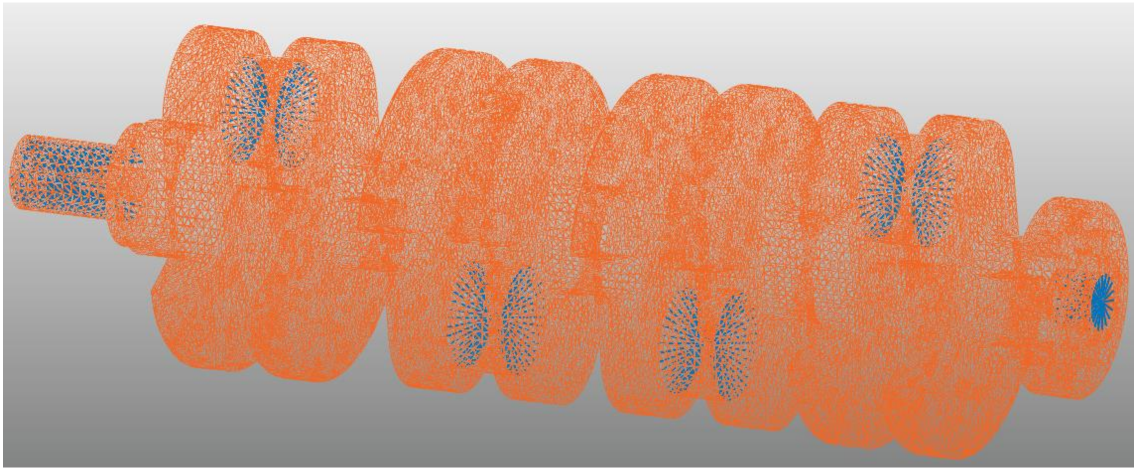
11. In the **Advanced Mesh** dialog box, perform the following steps:
    - In the **Mesh Type** dropdown menu, select **Solid4(Tetra4)**.
    - In the Mesh Option group, set the Avg. Element Size to 4 and the Min Element Size to 2.

- 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**.

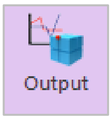
The mesh model for the crankshaft body is generated, as shown below.



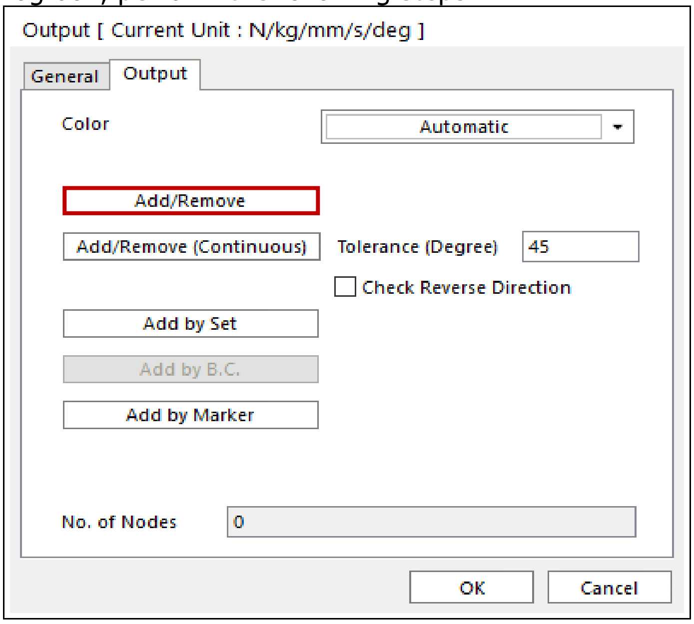
To see information about the FDR elements generated using the Assist Modeling function, change the view mode to Wireframe. Then, in the Database pane, under Property Components, select PFDR_1 to see all the FDRs generated for the crankshaft, as shown below.



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:



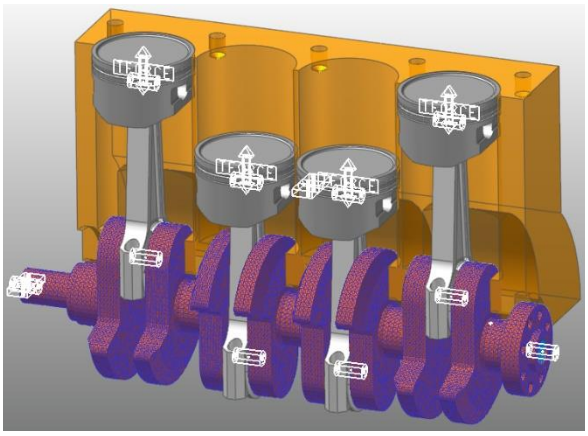
- Click **Add/Remove**.
- In the **Input Window** of the **Command Input Toolbar**, type **438** (node number), and then press **Enter**.

- 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**)



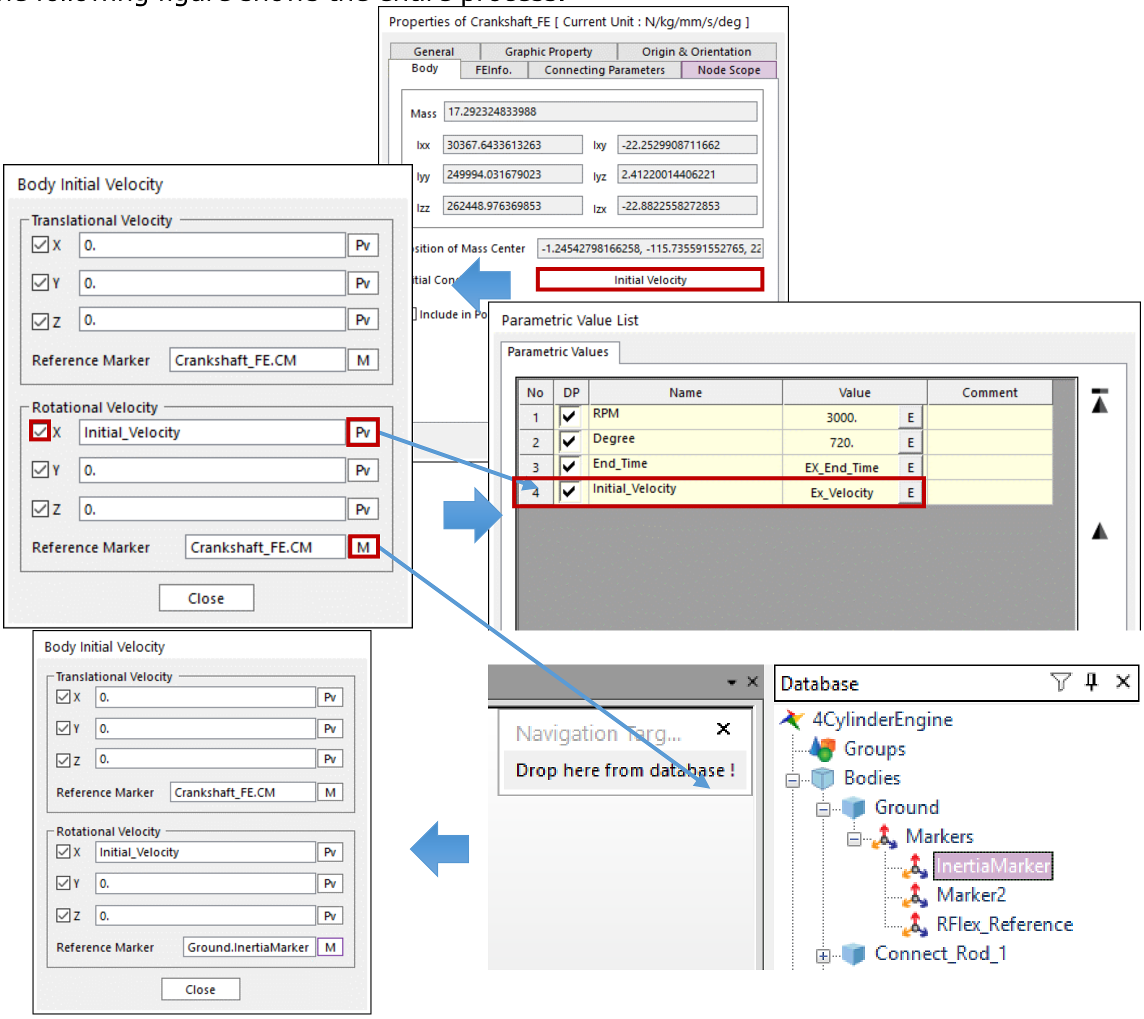
Conducting the Dynamic Analysis on the FFlex Body and Reviewing the Results

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**.

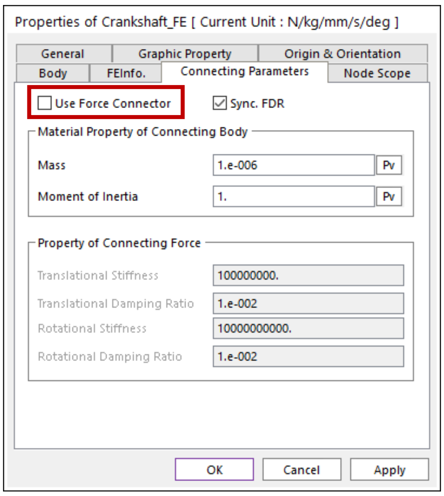
   The **Properties of Crankshaft_FE** dialog box appears.
3. In the dialog box, on the **Body** tab, click **Initial Velocity**.
4. In the **Body Initial Velocity** dialog box, perform the following steps:
   - Select **X** in the **Rotational Velocity** group, and then click the **PV** button.
   - Select **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.

The following figure shows the entire process.

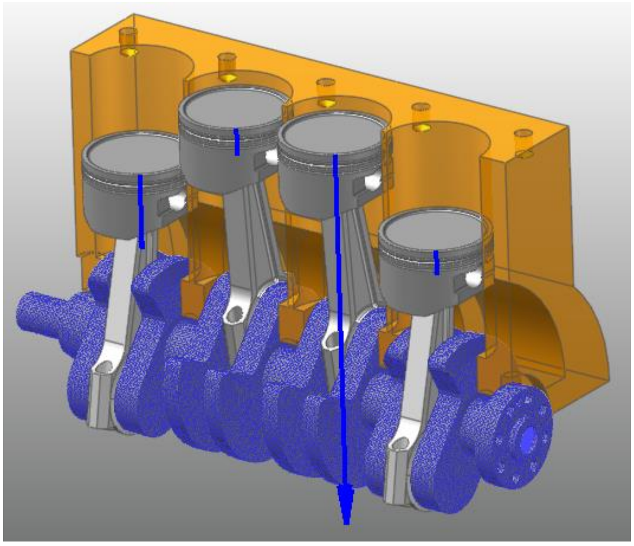


5. In the **Properties of Crankshaft_FE** dialog box, on the **Connecting Parameters** tab, clear the **Use Force Connector** checkbox.

6. Click **OK** to close the dialog box.



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.



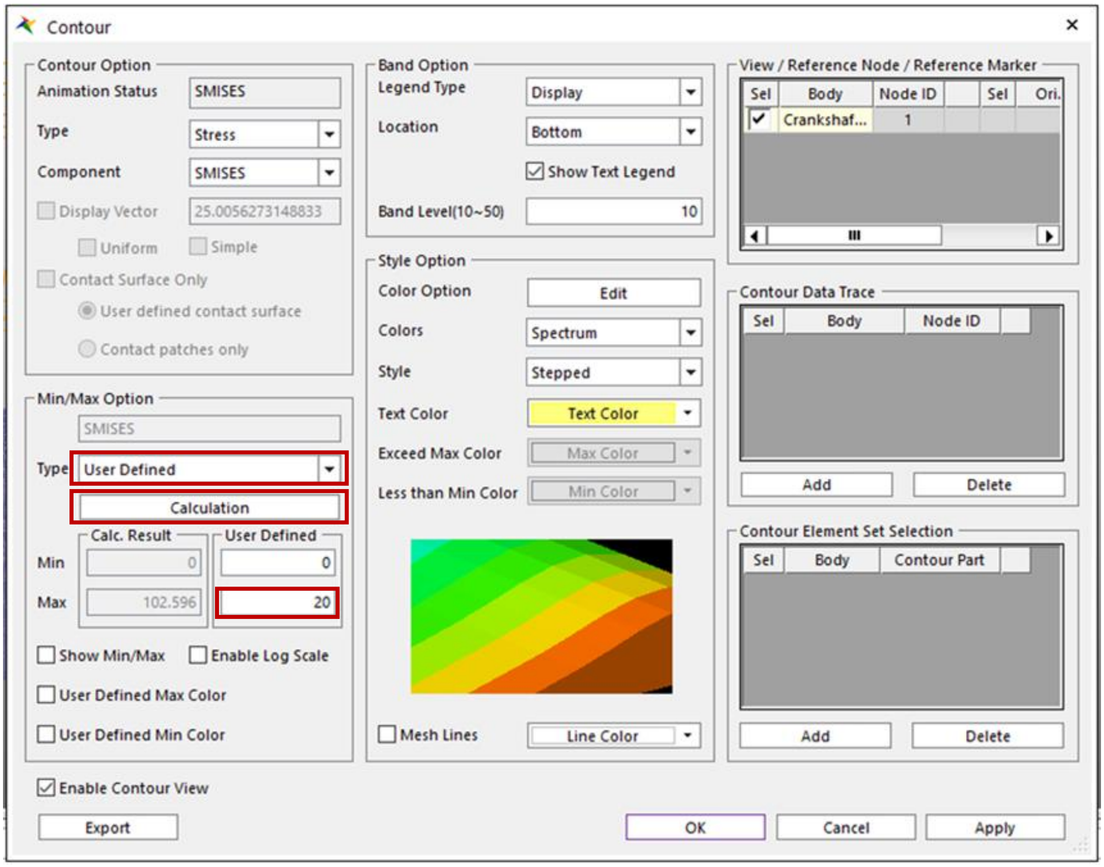
7. On the **Analysis** tab, click **Dyn/Kin**.
8. In the **Dynamic/Kinematic Analysis** dialog box, click **Simulation** to run the analysis without changing any settings.

   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.

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

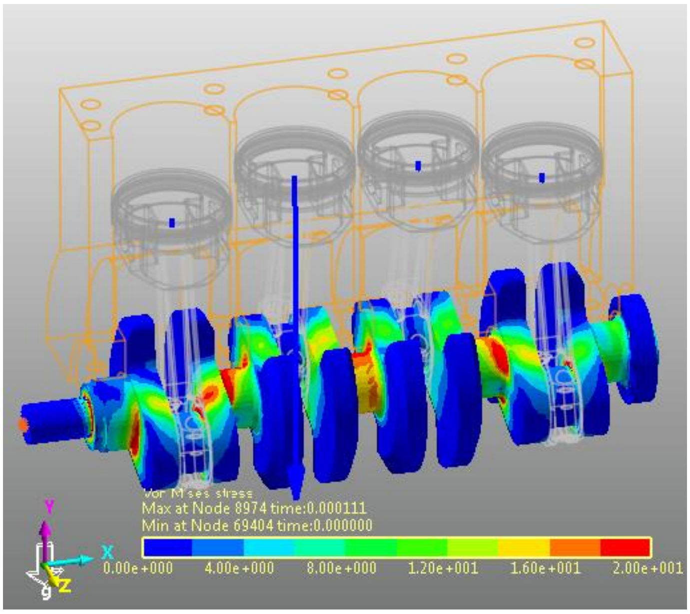
10. When the **Contour** dialog box appears, perform the following steps:



- 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.



Chapter

4

Generating the RFlex Body Using RFlexGen

Task Objectives

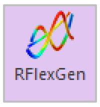
In this chapter, you will learn how to transform the FFlex body into an RFlex body using RFlexGen.

Estimated Time to Complete

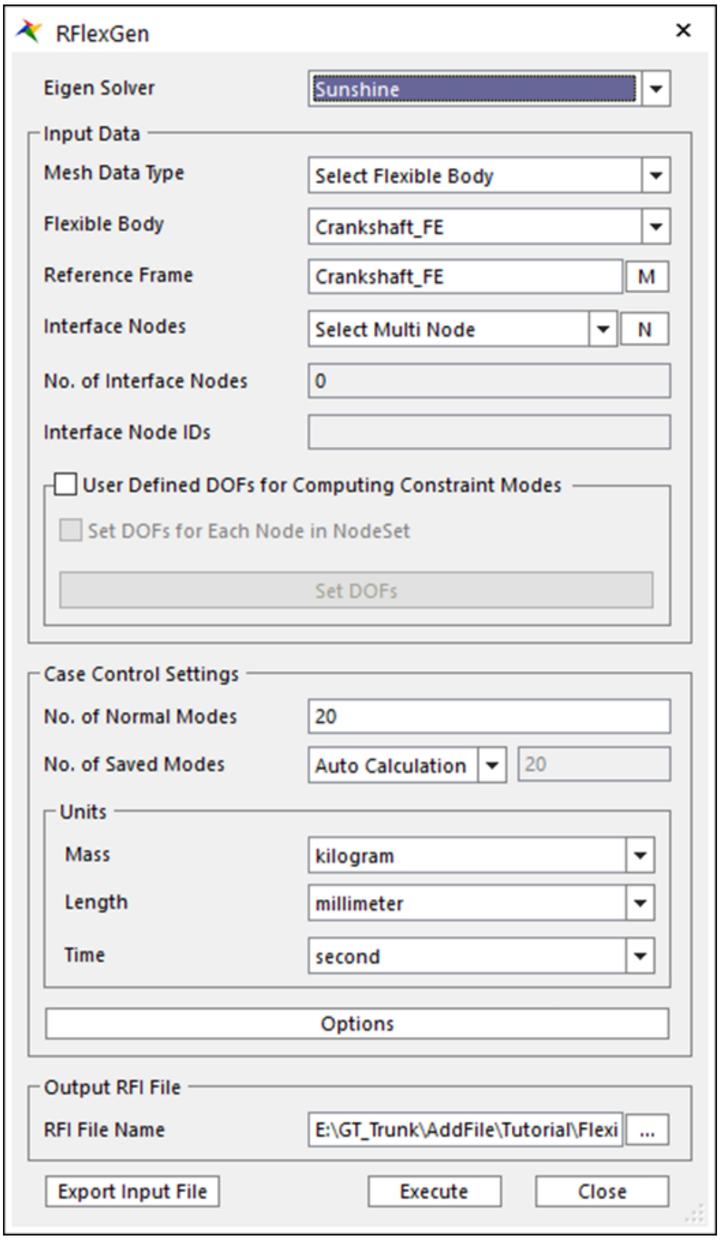
40 minutes

Running RFlexGen

To run RFlexGen:



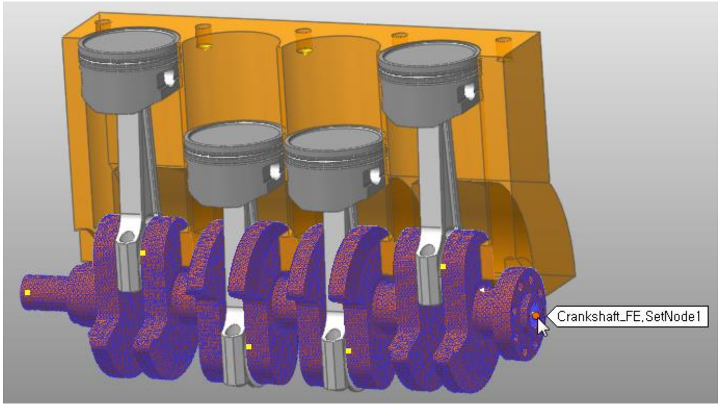
1. On the **Flexible** tab, in the **Flex Interface** group, click **RFlexGen**.



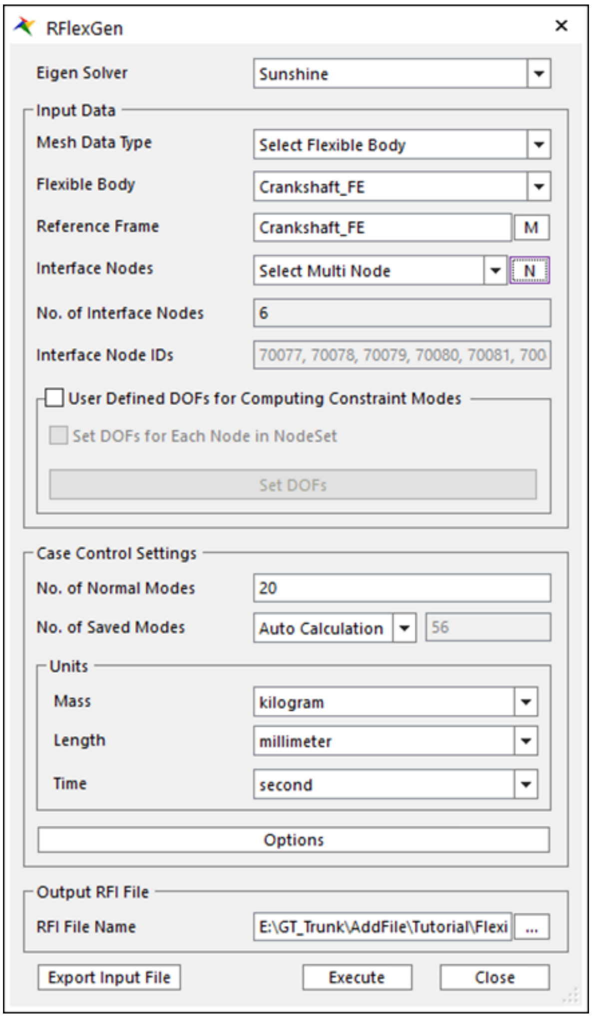
In this tutorial, **Crankshaft_FE** is the only FFlex body. Therefore, the **FFlex Body and Reference Frame** fields are automatically filled, as shown in the figure above.

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**.

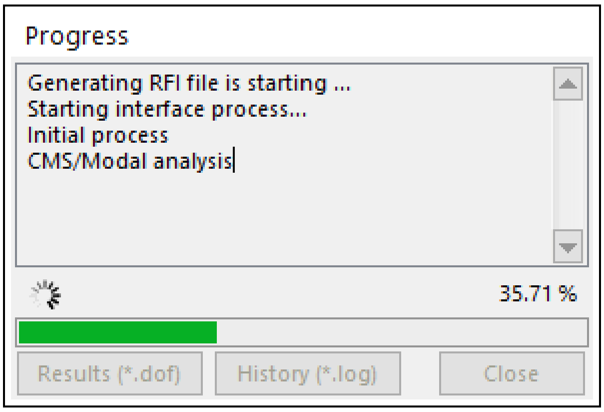
3. click **Finish Operation** on a mouse Right Menu. that was created earlier by selecting the master nodes in the model pane.



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.



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



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.

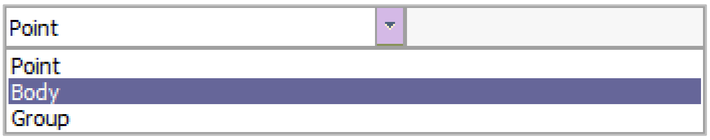
---

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

Creating the RFlex Body

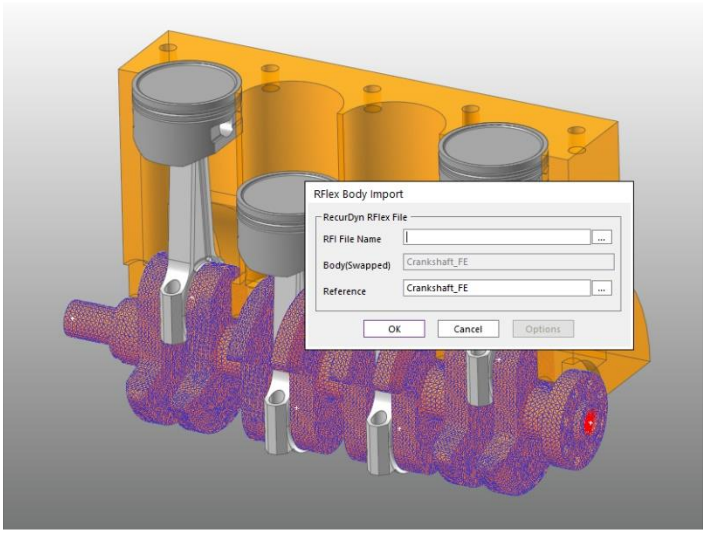
To create the RFlex body:

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



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

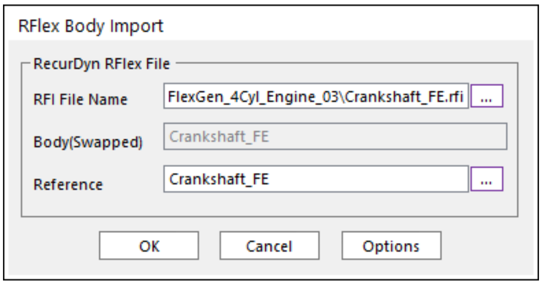
   The **RFlex Body Import** dialog box appears.



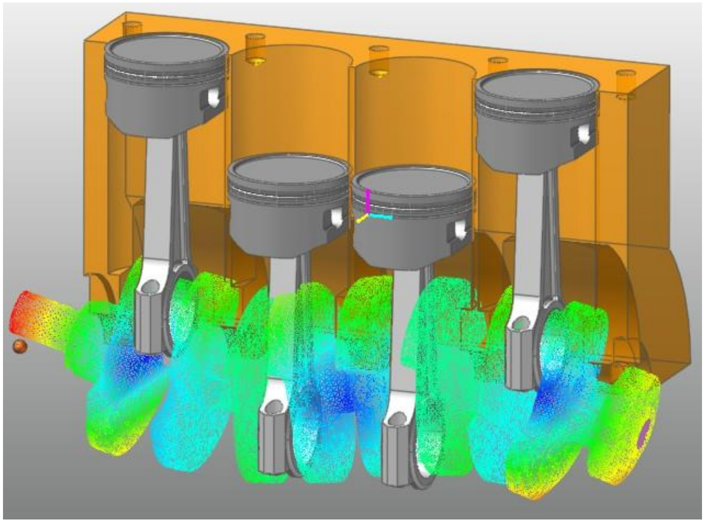
4. In the **RFlex Body Import** dialog box, perform the following:
   - Click **...** to the right of the **RFI File Name** field.
   - Select the RFI file that you created in the **Running RFlexGen** chapter.
   - Click **...** to the right of the **Reference** field, and then select **Crankshaft_FE**.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

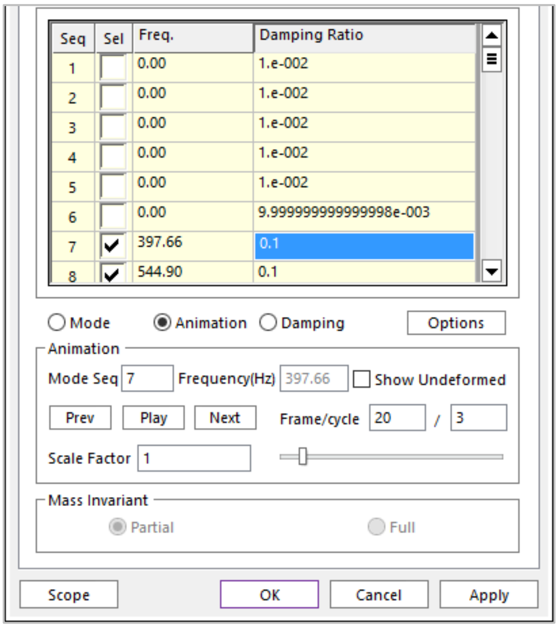
Tip: You can use the name of FFlex body or RFlex body in the Reference field to automatically set the Body Reference Frame of the flexible body. Therefore, the above procedure sets the Body Reference Frame of the crankshaft FFlex body to the reference frame of the Flex body instead of the center marker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

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



6. Click **Icon Control** in the toolbar.
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.
    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.
    c. You can also observe the model in other modes by performing the above steps and changing the mode.



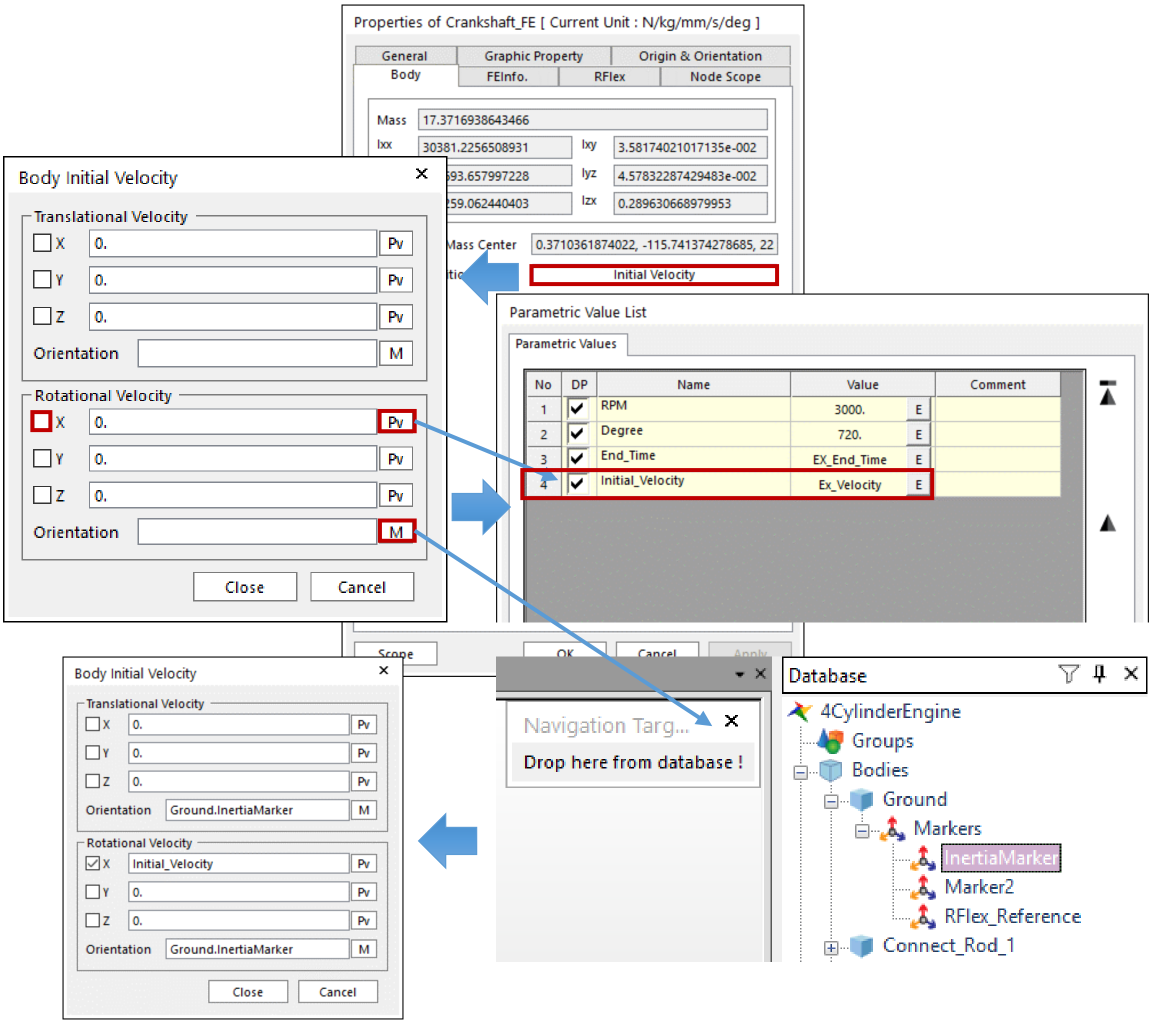


Conducting a Dynamic Analysis on the RFlex Body and Reviewing the Results

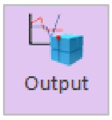
To conduct a dynamic analysis on the RFlex body:

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.



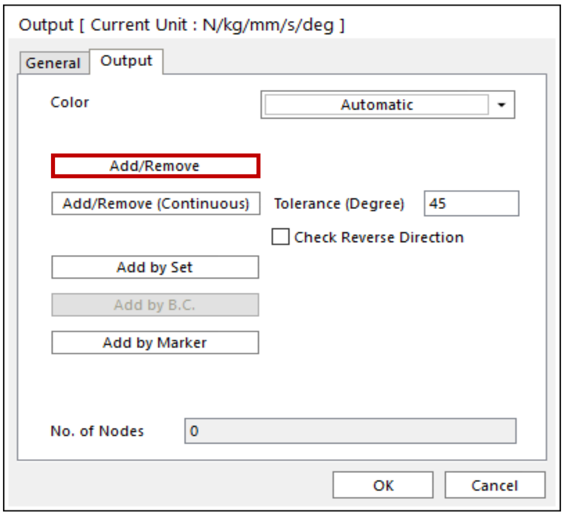
4. In the **Database** pane. double-click **RFlexBody1** to enter **RFlex Edit** mode.



5. To see the stress result for a specific node in RFlexBody1 as a plot, click **Output** in the **RFlex Edit** menu.

6. 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**.

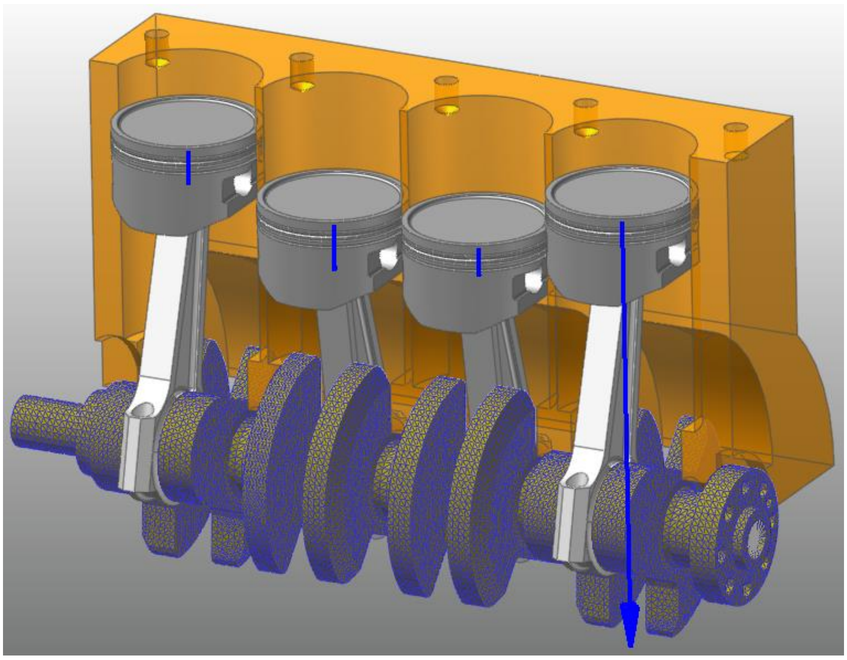


7. Right-click the screen to display the context menu, and then click **Exit**.

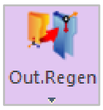
8. On the **Analysis** tab, click **Dyn/Kin** to open the **Dynamic/Kinematic Analysis** dialog box.

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.

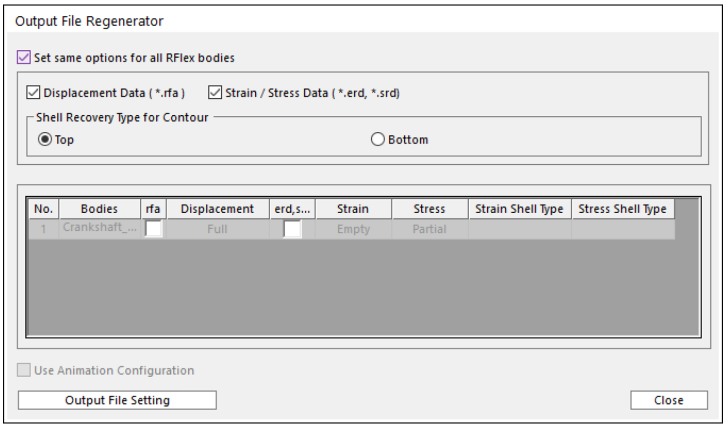


To display the distribution of stress in the RFlex body:



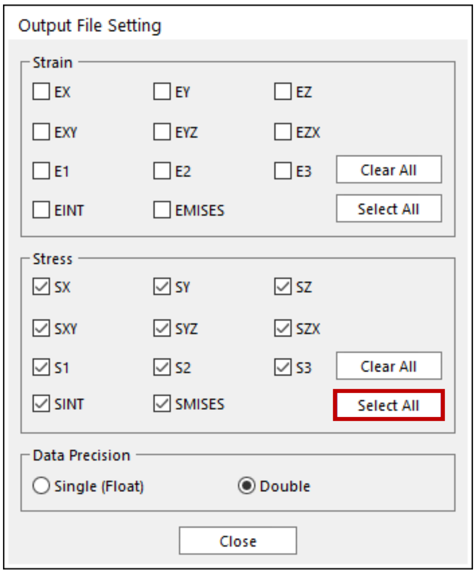
1. On the **Flexible** tab, in the **FFlex** group, click the **Contour** dropdown menu, and then click **Output Regenerator**.

   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.

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



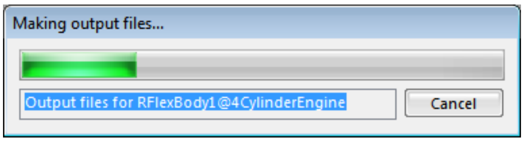
4. In the **Stress** group, click **Select All**.
5. Click **Close**.

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.

---

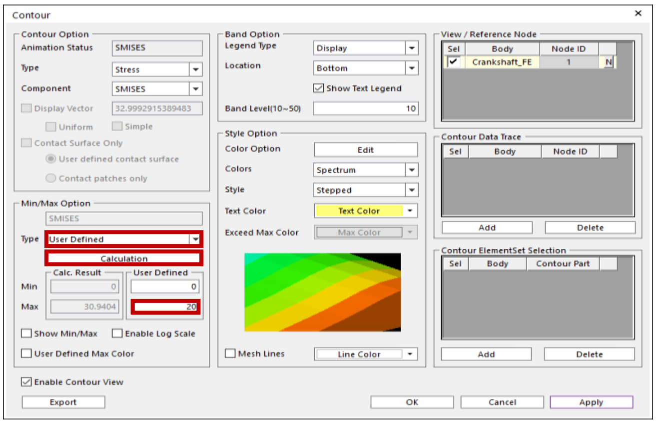
6. In the **Output File Regenerator** dialog box, click **Generate**.

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



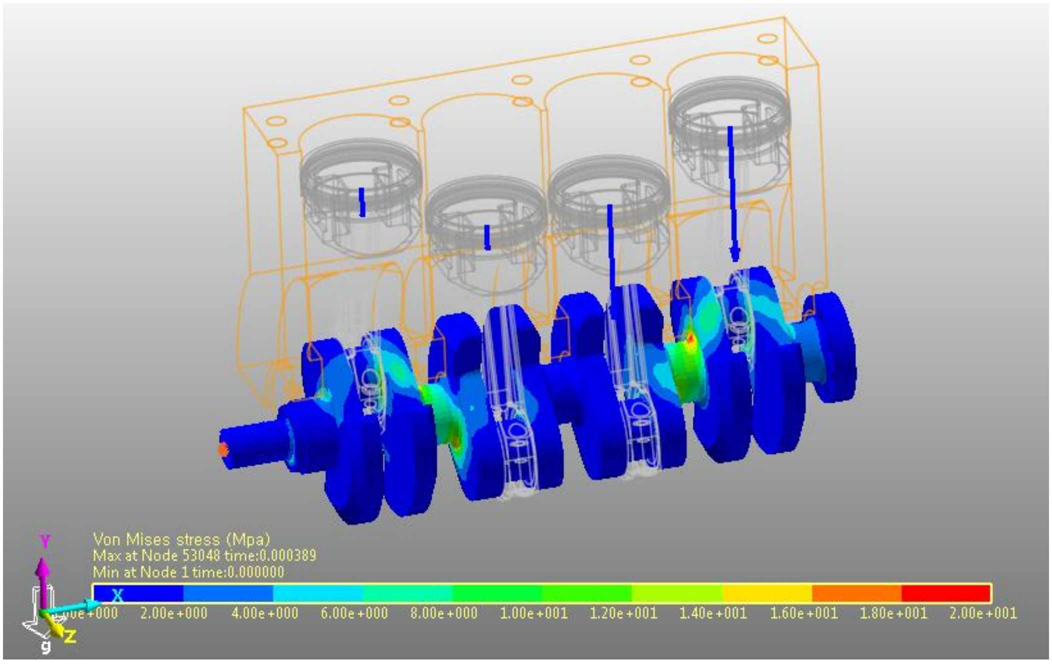
   (**Tip**: If the output file was created using the default settings, then the Stress column changes to Partial rather than Full. This is because, out of the 11 stress tests available, only the Von-Mises, Sx, Sy, and Sz tests were conducted.)

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.



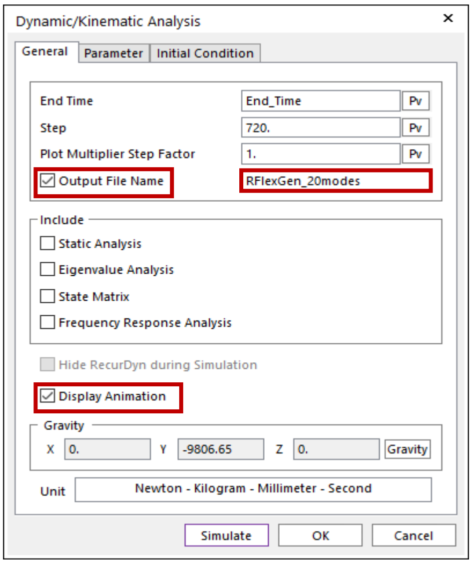
10. On the **Analysis** tab, click **Dyn/Kin** to open the **Dynamic/Kinematic** dialog box.

11. In the **Dynamic/Kinematic** dialog box, perform the following:

    a. Select Output File Name.

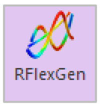
    b. In the Output File Name text box, type RFlexGen_20modes.

    c. To see the animation while the analysis is being performed, select the Display Animation checkbox.



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.

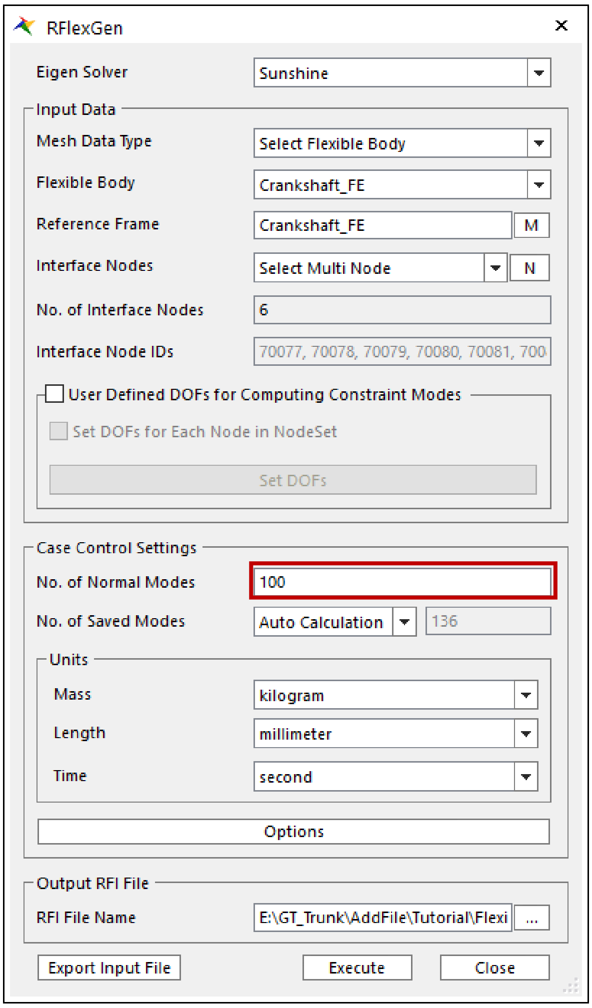
Running RFlexGen Again



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.



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**.
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**.

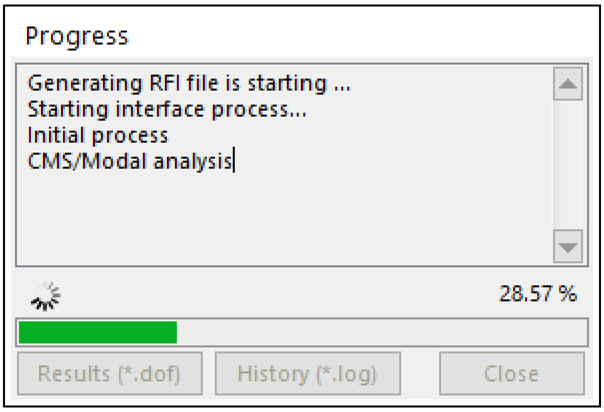
   b. In the **Units** group, maintain the default settings.

   c. In the **Output RFI File** group, click **...**, and then specify the name and folder path of the **RFI** file.

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.

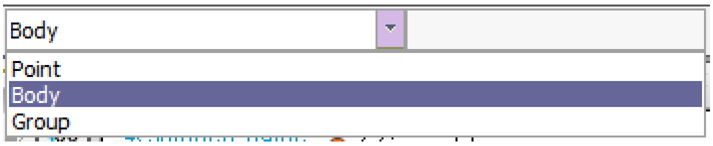
6. Once the analysis is complete, click **Close** to stop the **RFlexGen** process.



Replacing an RFlex Body

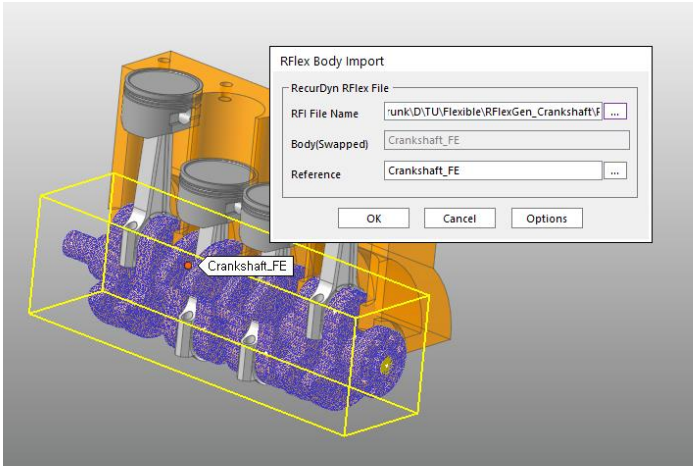
To replace an RFlex body:

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

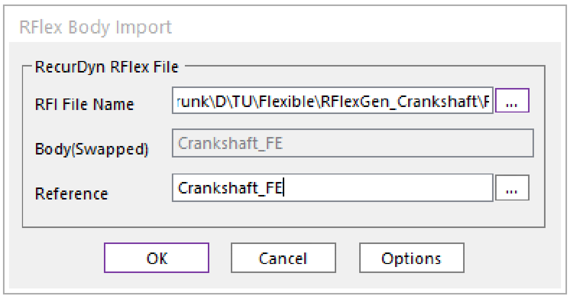


3. In the model pane, select the **Crankshaft_FE**, as shown below.

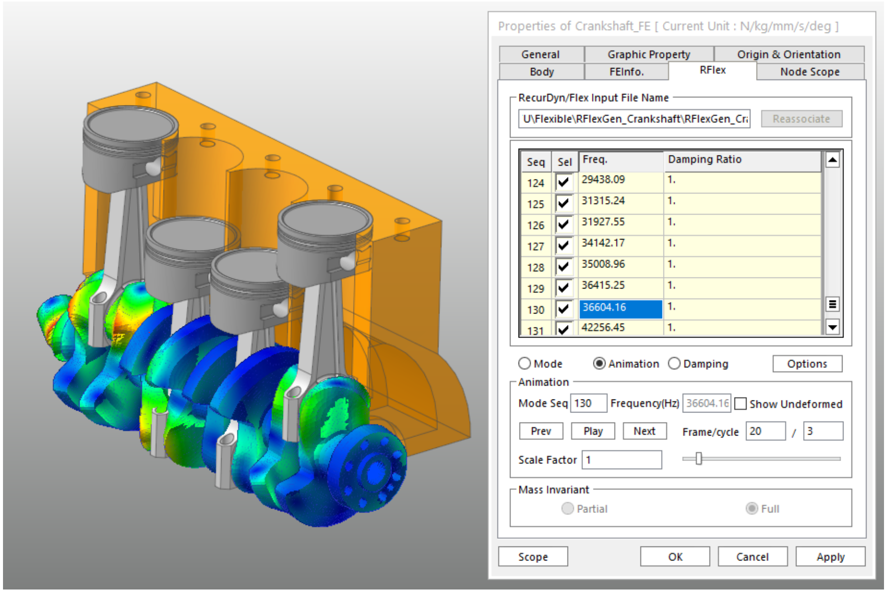
   The **RFlex Body Import** dialog box appears.
4. In the **RFlex Body Import** dialog box, perform the following:
   a. Click **...** to the right of the RFI File Name field, and then specify the path and name of the **RFI** file (as described in the Running **RFlexGen** chapter).
   b. Click **...** 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**.



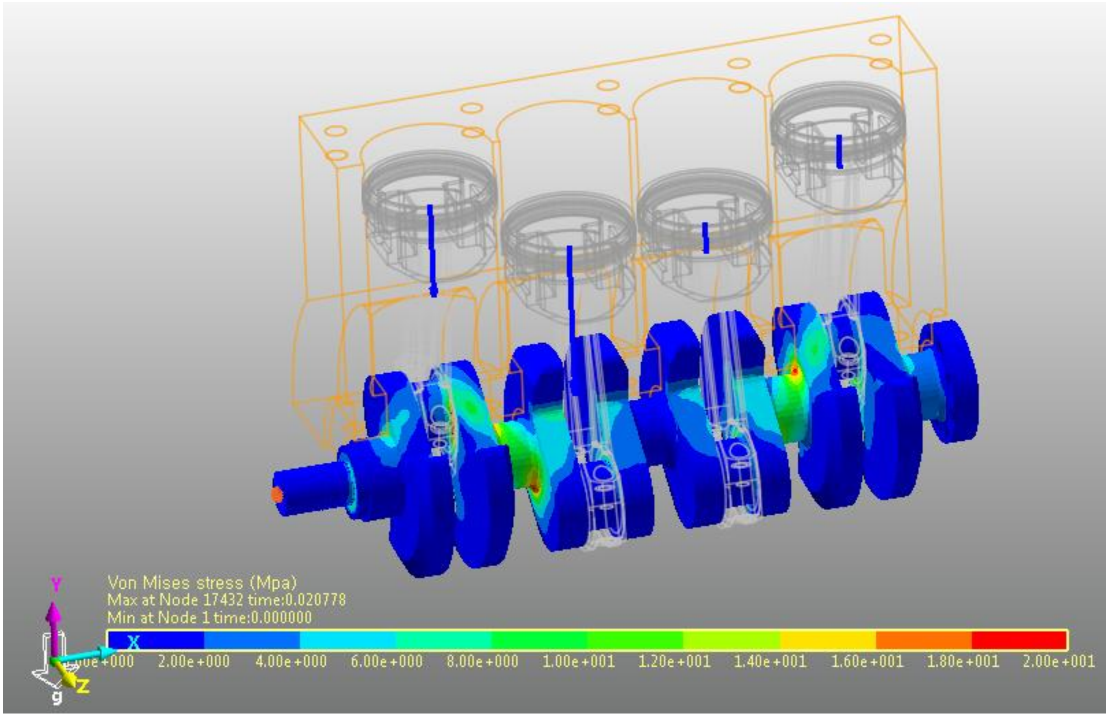
6. In the **Database** pane, double-click **Crankshaft_FE** to display the **Property** dialog box.
7. In the **Property** dialog box, perform the following
   - Select the **130th mode**, as shown in the figure to the right.
   - Click **Play** to observe the behavior of the selected mode.


Properties of Crankshaft_FE [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
Sel
Freq.
Damping Ratio
124 29438.09 1.
125 31315.24 1.
126 31927.55 1.
127 34142.17 1.
128 35008.96 1.
129 36415.25 1.
130 36604.16 1.
131 42256.45 1.
Mode
Animation
Damping
Options
Animation
Mode Seq 130
Frequency(Hz) 36604.16
Show Undeformed
Prev
Play
Next
Frame/cycle 20 / 3
Scale Factor 1
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



Chapter

5

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.

Estimated Time to Complete

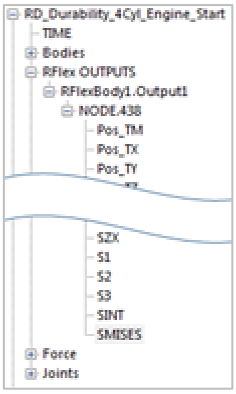
5 minutes

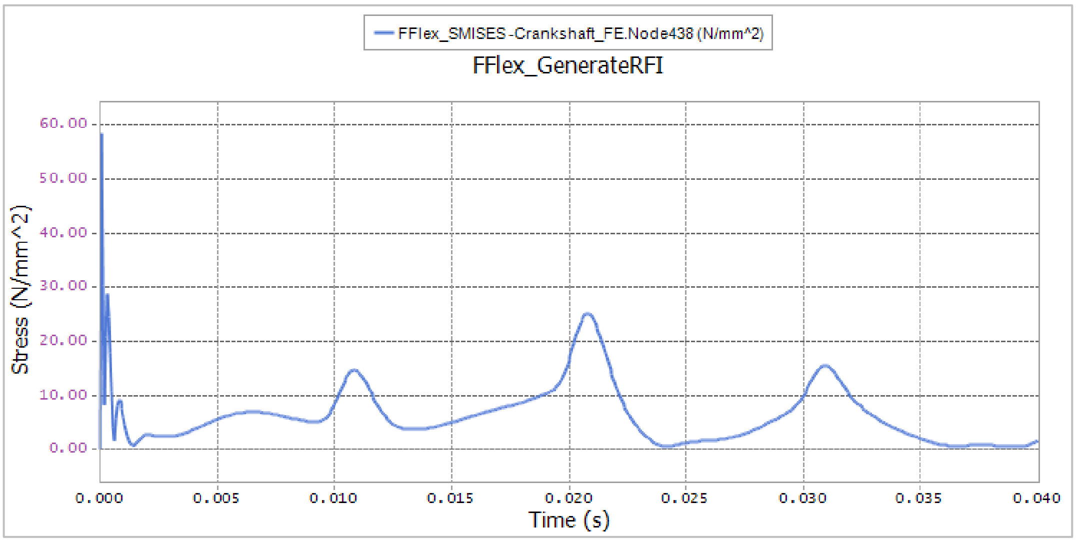
Analyzing the Dynamic Analysis Results

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.)

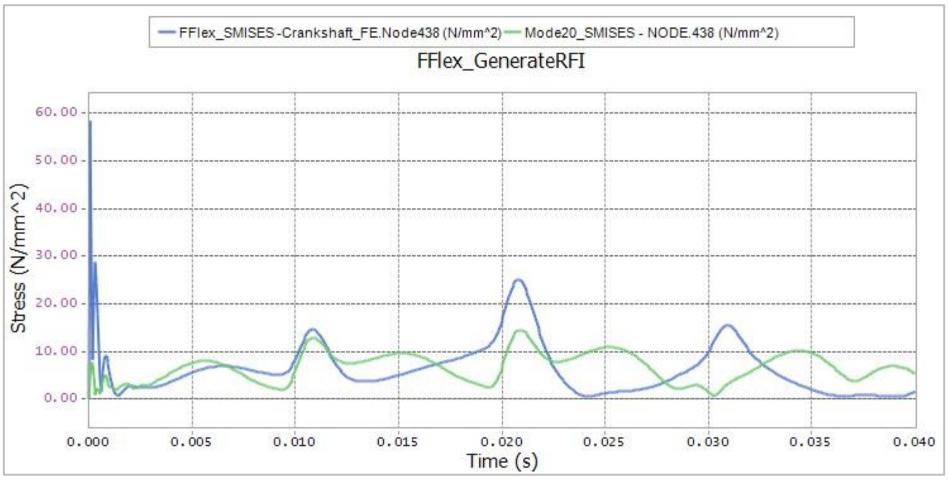
The following figure shows the results of the Von-Mises Stress test performed on the nodes that were included in the area of interest in the analysis results. (Since we defined the Output in the earlier steps, the test results were stored as a plot.)





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.

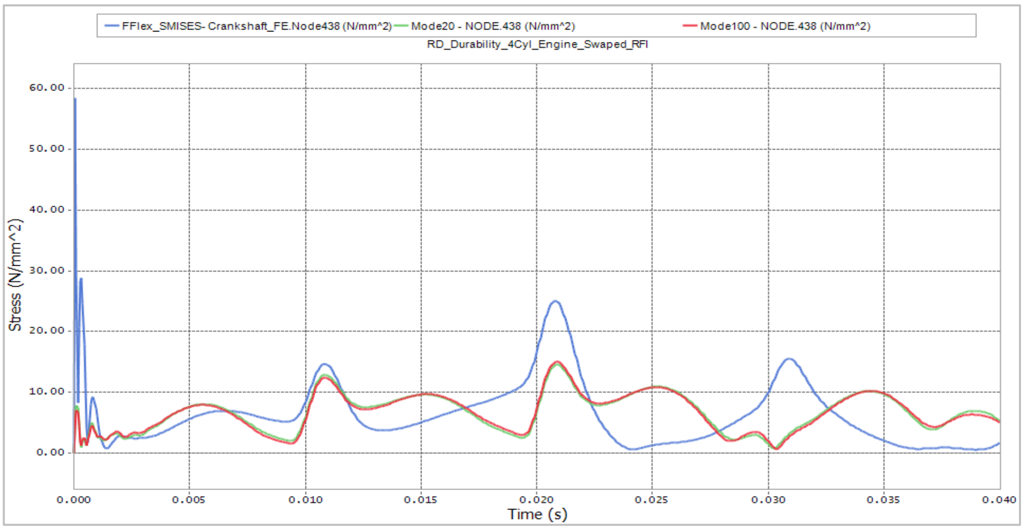


For the crankshaft used in this tutorial, we do not need to consider its contact with other bodies and the crankshaft is only bound by joints. Therefore, we don't need an FFlex body to simulate the flexibility of the crankshaft because an RFlex body is sufficient for that purpose. That is, in an analysis 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