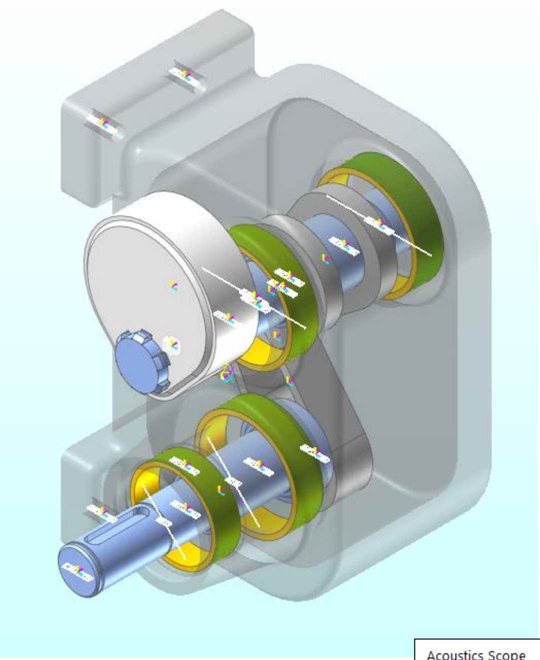
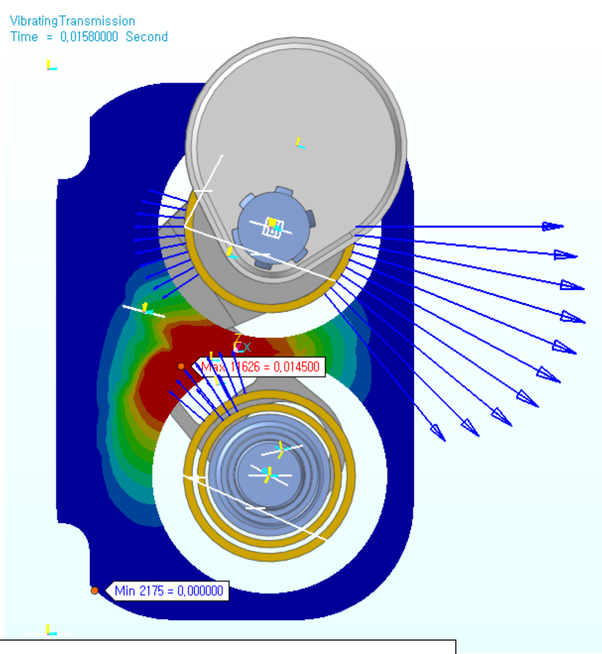
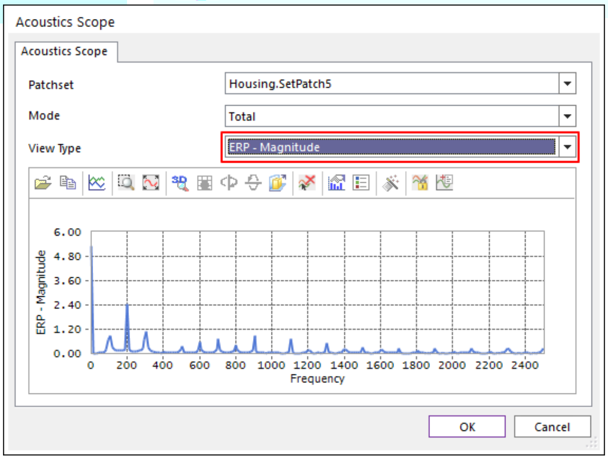
Vibrating Transmission (Acoustics)







FunctionBay

Edition Note

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

Contents

Chapter

1

Overview

As the quality of human life improves, many efforts are being made to control noise and vibration in various industries. In the mechanical industry, it is very important to find out the part that generates a large amount of vibration and the frequency band with a large amplitude.

The example model used in this tutorial is a link system that makes rotary motion into the reciprocating motion with a vibrating transmission. When analyzing the vibration generated in the housing of this mechanical system, it is difficult to analyze clearly where and what kind of vibration occurs from the result of the existing FE analysis alone.

Therefore, this tutorial introduces and explains a method to calculate the ERP (equivalent radiated power) generated in the housing by using the post tool called Acoustics, which analyzes the noise vibration, and a method of analyzing the vibration occurring in the housing using the post functions.

Task Objectives

This tutorial covers the following topics:

- How to replace RFlex Body through RecurDyn/RFLEX
- How to perform ERP calculation and analysis through RecurDyn/Acoustics

Prerequisites

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

Task

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

Procedures	Time (minutes)
Simulating and analyzing the initial model	5
Changing an existing body to the RFlex body	15
Calculating equivalent radiated power	15
Modifying and analyzing the model	15
Total	50

Estimated Time to Complete this Task

75 minutes

Chapter

2

Simulating and analyzing the initial model

Task Objectives

Open the initial model, perform a simulation, and observe the behavior of the vibrating transmission.

Estimated Time to Complete This Task

5 minutes

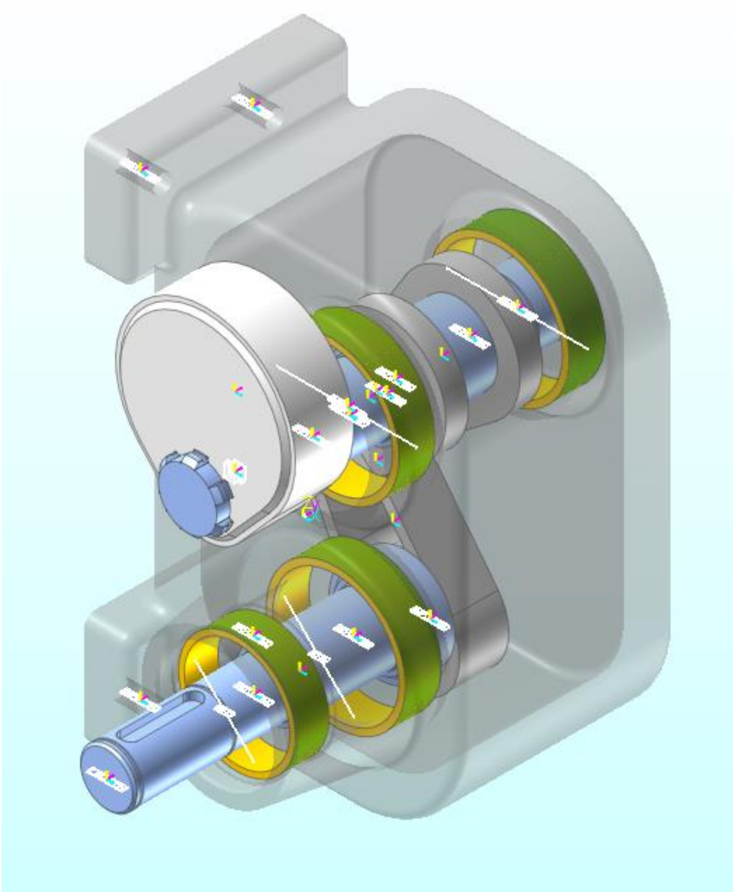
Opening the Model

To copy the example model:

- Copy the Acoustics tutorial example folder provided by RecurDyn to an analyzable location.
- Folder path: <Install Dir>\Help\Tutorial\PostAnalysis\Acoustics\VibratingTransmission

To run RecurDyn and open the initial model:

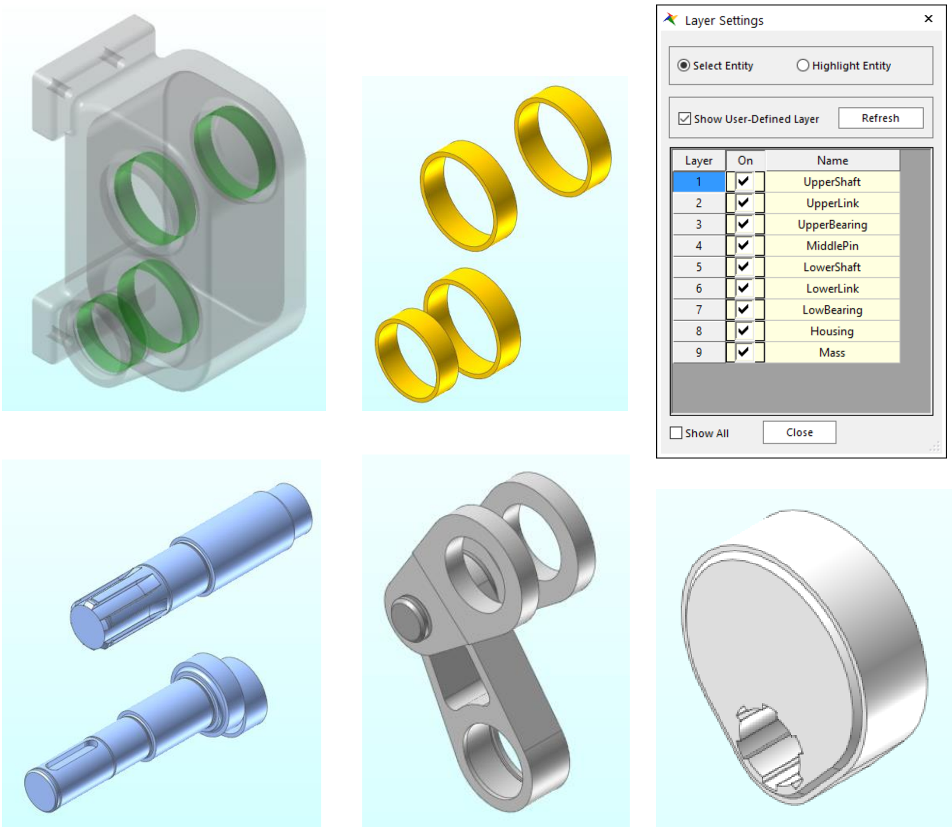
1. On the Desktop, double-click the **RecurDyn** icon to run **RecurDyn**. The **Start RecurDyn** dialog window will appear.
2. When the **Start RecurDyn** dialog window appears, close it.
3. In the **File** menu, click **Open**.
4. In the Acoustics folder copied above, select **VibratingTransmission_Start.rdyn**.
5. Click **Open**. The model appears as shown in the following figure.



To analyze the model:

1. Click **Layer Settings** in the **Render Toolbar**.
2. In the Layer Settings dialog window, turn **on** and off each layer to analyze the model.

The following explains the configuration of the model.



The Model generally consists of 5 parts: **Housing**, **Bearing**, **Shaft**, **Link**, and **Mass**. When the rotary motion is delivered from the lower drive shaft, the pendulum on the upper shaft performs the periodic motion through the link system.

The connection between the shaft, bearing and link are all made up of bushing force, and the housing and bearing are composed of **GeoSurContact**.

The drive shaft is driven by **CMotionGroup** and rotates at 100 * 2PI per second.

Performing Simulation

Run the simulation to help you understand the model system.

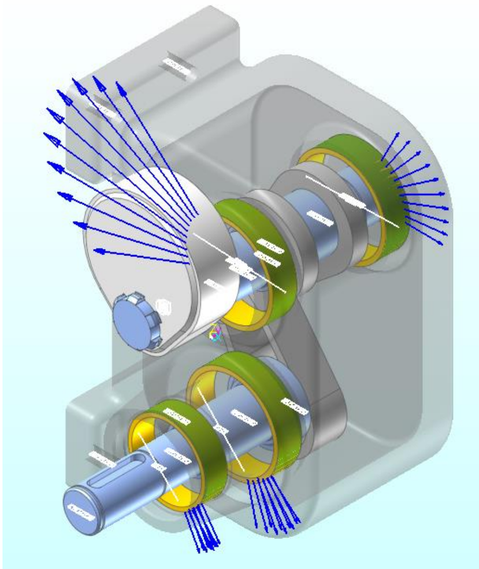
To run the simulation:

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

   The **Dynamic/Kinematic Analysis** dialog window appears.
2. After verifying the simulation conditions, click **Simulation**.
   - **End Time**: 0.1
   - **Step**: 500

To view the result:

- Under the **Analysis** tab, in the **Animation Control** group, press the **Play** button to check if the system operates as shown in the figure below.



As described in the "Analyzing the model" section, the motion delivered from the lower drive shaft is transmitted to the link system, and finally the mass on the upper shaft makes a periodic motion.

In the process of power transmission as shown in the figure, the housing receives contact force from 4 bearings.

If you observe the force display of the contact force, you can notice that it moves at a certain cycle. To analyze how this external force directly affects the housing, it is necessary to change the housing, which is a rigid body, to a flexible body.

Chapter

3

Changing an existing body to the RFlex body

Since the housing of the RecurDyn model is a rigid body, you cannot see the deformation of the body due to external force. Therefore, we will perform the RFlex Body Swap function provided by RecurDyn to replace the housing with a flexible body. In addition, the Acoustics Post Tool to be performed in the later chapter can only be performed on a flexible body.

Task Objectives

In this chapter, you will learn how to change an existing rigid body to a flexible body using the RFlex Body Swap function provided by RecurDyn/RFlex.

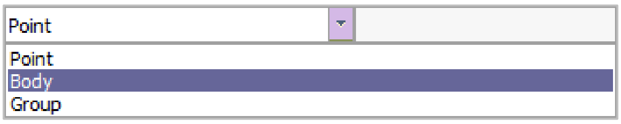
Estimated Time to Complete This Task

15 minutes

Modifying a Model Using a RFlex Body

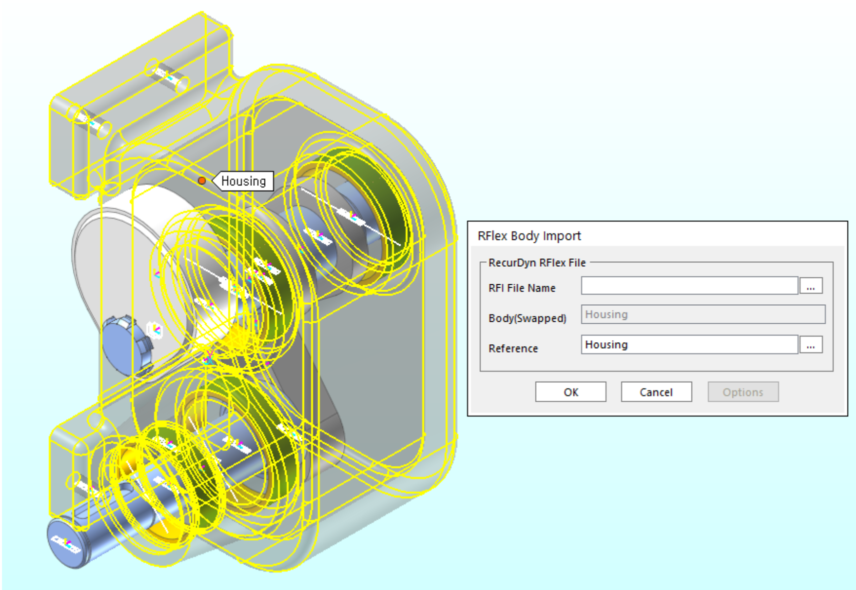
To swap with a RFlex Body:

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



3. In the working window, select the **housing** using the mouse as shown below.

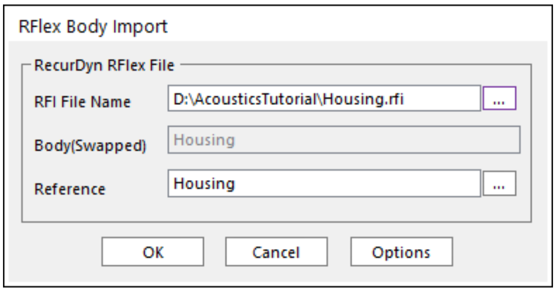
   The RFlex Body Import dialog window appears.



4. In the RFlex Body Import dialog window, perform the following:
   a. Press **...** button in the **RFI File Name** field.
   b. Select the **Housing.rfi** file located in the Acoustics folder copied in Chapter 2.

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

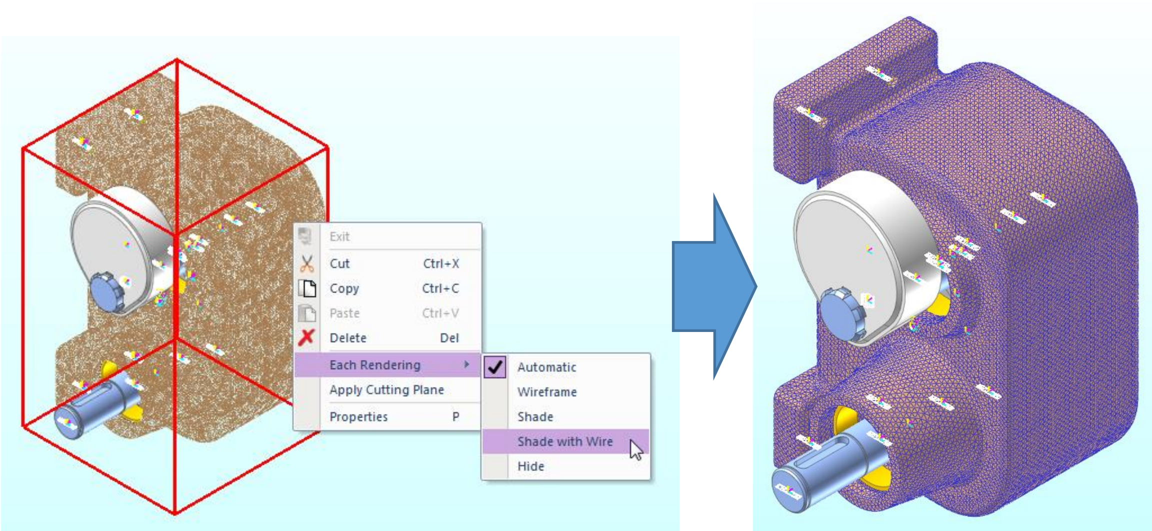
   You can verify that the **housing** was replaced by a **RFlex body**.



The swapped housing looks like a wire frame. Change each rendering just like the previous **housing**.

To change each rendering:

1. In working window, select the **housing** that changed to a RFlex body.
2. Right-click the working window.
3. In the pop-up menu, change Each Rendering to **Shade with Wire**.



Redefining Contact

When changing from a rigid body to a RFlex body, the information about contact patches will disappear and all existing GeoSurContact will also disappear. After creating the patches again in the housing, you need to redefine the contact.

To create PatchSet:

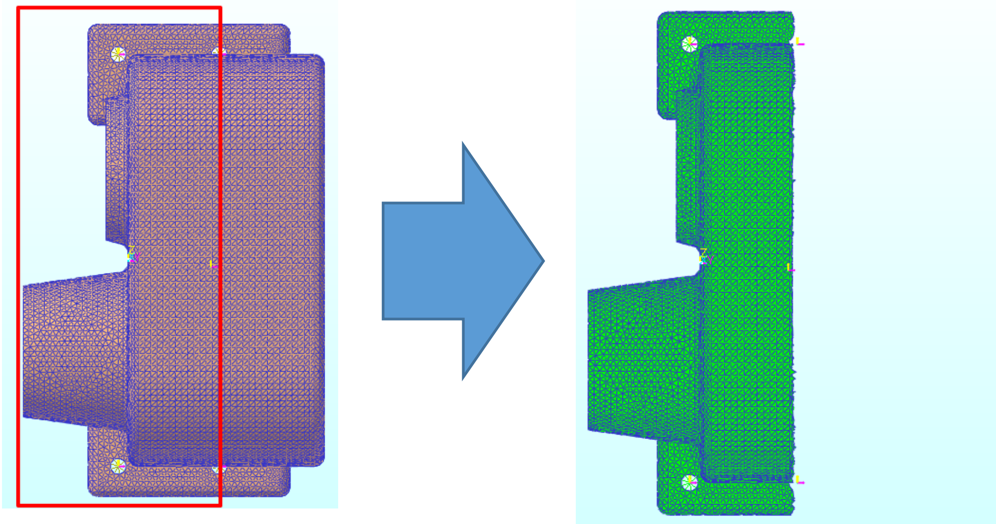
1. Enter the **Edit Mode** of the RFlex body.

2. Change the plane to the **YZ** plane.

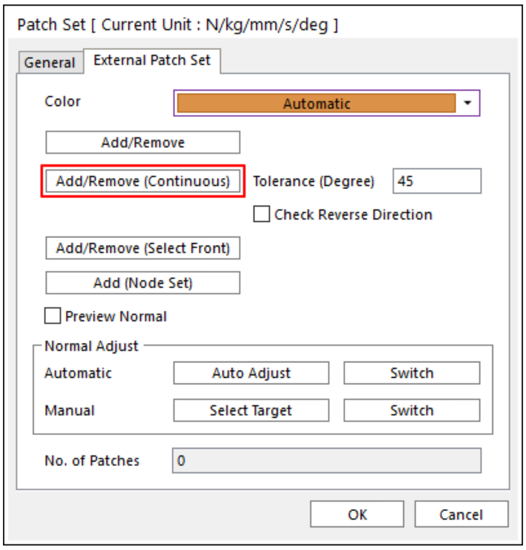
3. In **Select Toolbar**, click **Element**.
4. In the working window, select a half of the housing.

5. In **Select Toolbar**, click **Set Masking**.

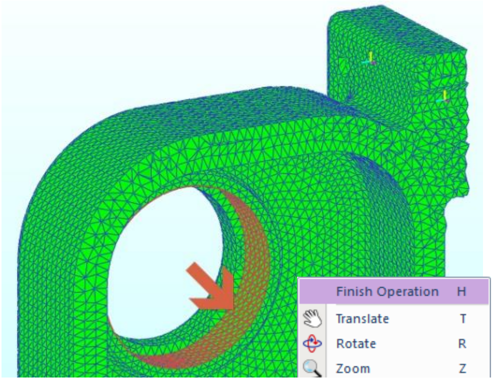
   Only the selected part of the housing will be masked as shown below. Now you can easily select the inside of the housing.



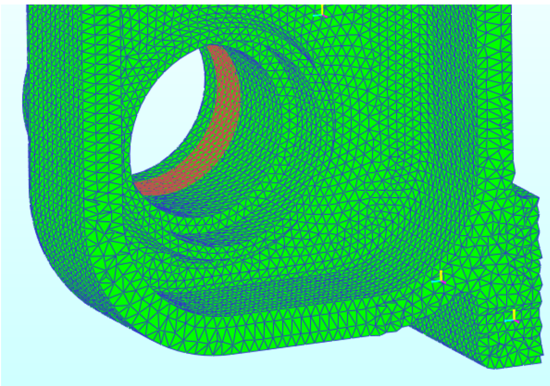
6. In the **Set** group of the F**RFlex Edit** tab, click **Patch**.
7. Click **Add/Remove (Continuous)** in the Patch Set dialog window.

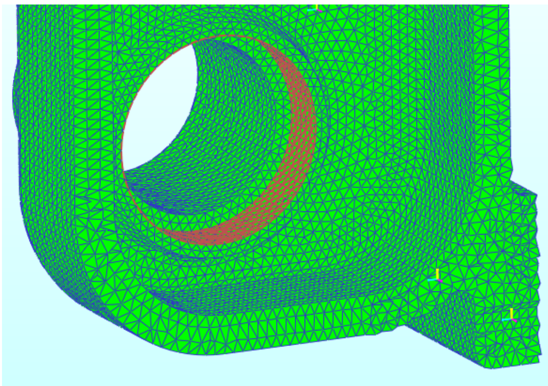


8. Select the upper bearing position as shown on the right.
9. Right-click the working window and click **Finish Operation** in the pop-up menu.
10. Click **OK** in the Patch Set dialog window.



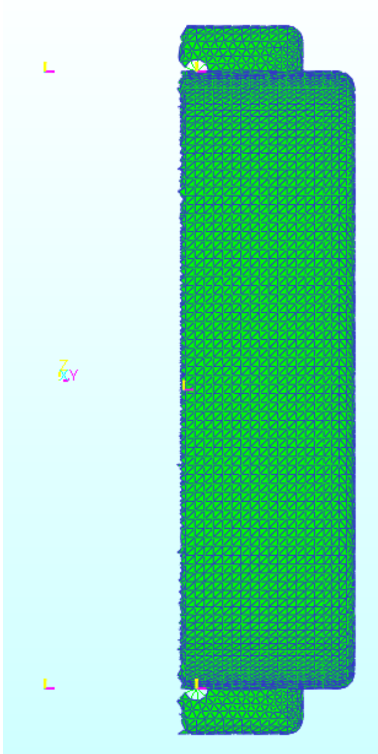
11. Repeat steps **3 to 10** to create a patch set in other bearing positions as shown below.



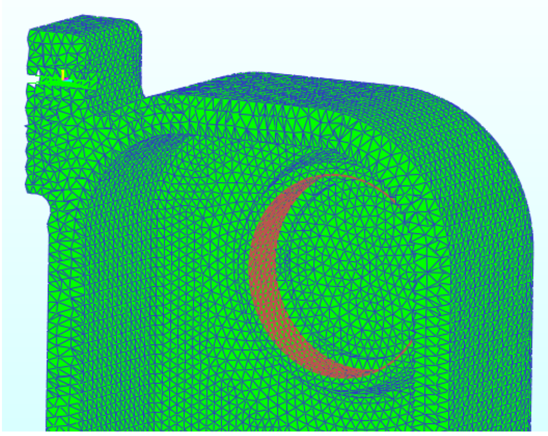


12. In **Select Toolbar**, click **Set Reverse Masking**.

    Only except for the selected part above will be masked as shown right.

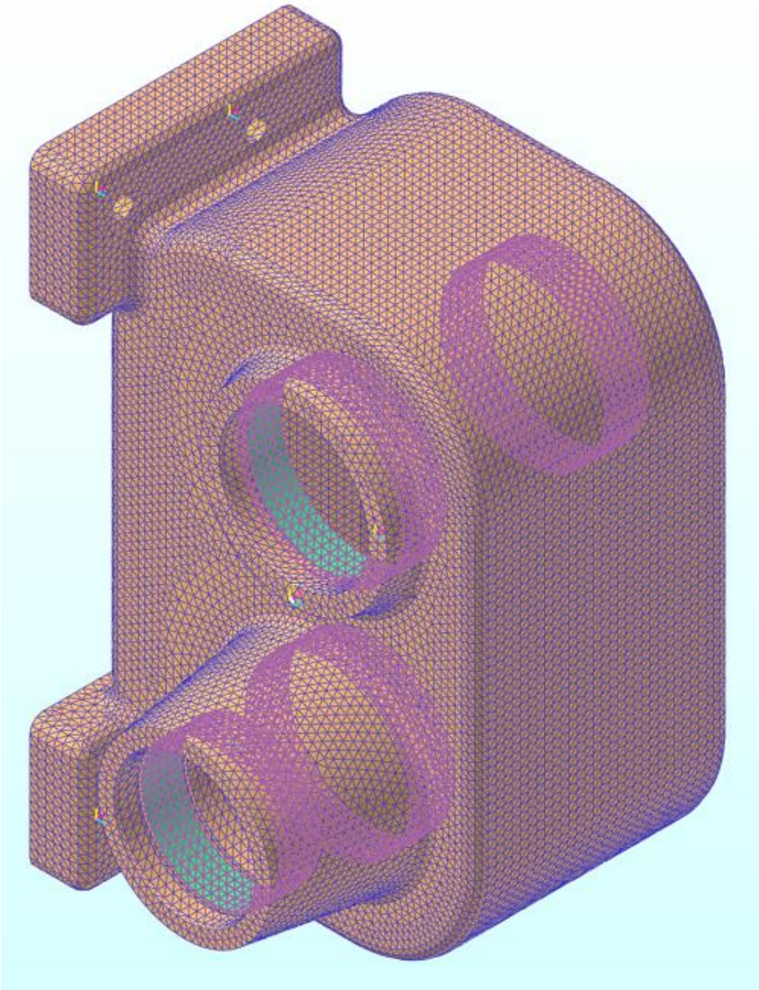


13. Repeat steps **3 to 10** to create a patch set in other bearing positions as shown to the right.



14. When the four patch sets are created, click **Cancel Masking** on the **Select Toolbar**.

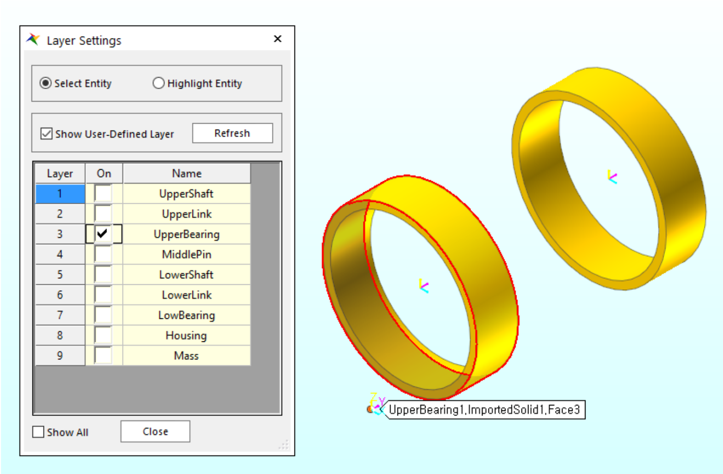
15. Click **Exit** to exit the **Edit Mode**.



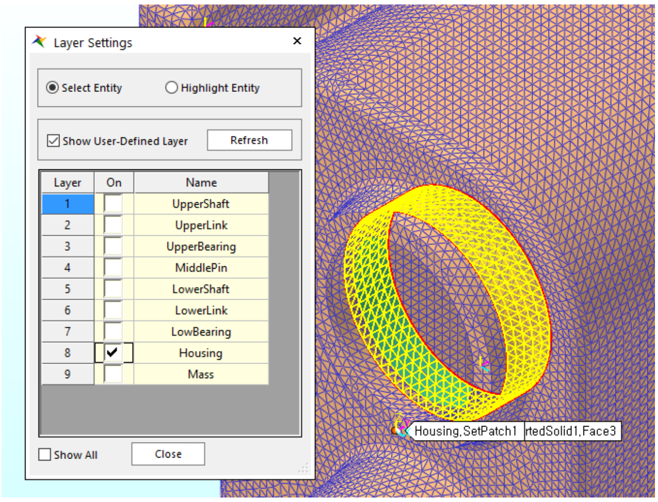
Define the contact between the patch set and the bearing.

To create Geo Surface Contact:

1. On the **Flexible** tab, in the **RFlex** group, click **GeoSur**.
2. Change the Creation Method to **Surface(PatchSet), Surface(PatchSet)**.
3. Click **Layer Settings** in the **Render Toolbar**.
4. In the Layer Settings dialog window, check **On** only for the **UpperBearing**.
5. In the working window, click **UpperBearing1.ImportedSolid1.Face3**.



6. In the Layer Settings dialog window, turn off **UpperBearing** and ensure that only the **housing** is visible.
7. Click the patch set of the **housing** suitable for the position of bearing selected in **5**.



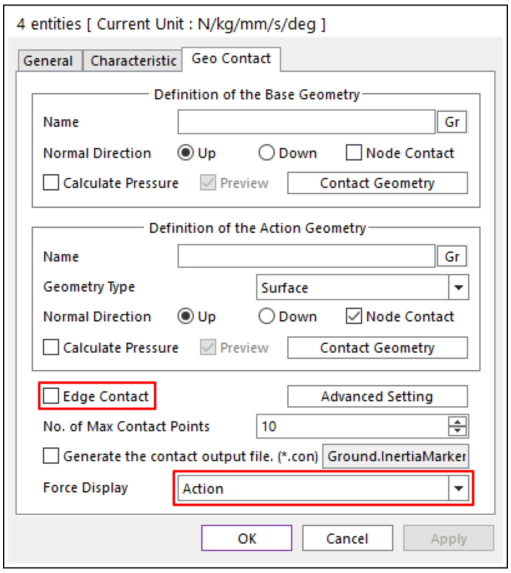
GeoSurContact1 is created.

8. Repeat steps **1 to 7** to create a contact in the remaining bearing positions.

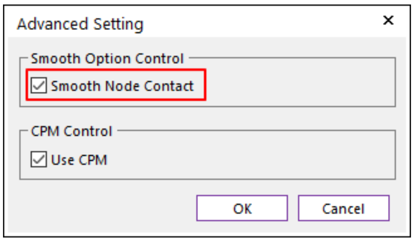
Modify the Contact property to the same state as in the initial model.

Modifying the property of GeoSurContact

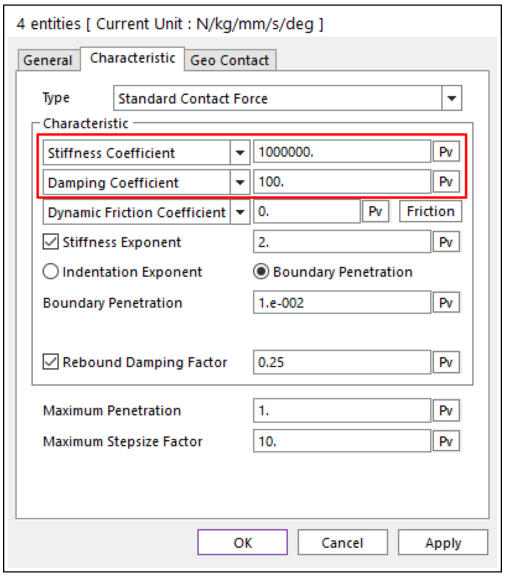
1. In the Database panel, select **GeoSurContact1, GeoSurContact2, GeoSurContact3,** and **GeoSurContact4** and open the Multi Property dialog window.



2. Turn off the **Edge Contact** option.
3. Set the **Force Display** as **Action**.
4. Click **Advanced Setting**.
5. In the Advanced Setting dialog window, turn on **Smooth Node Contact** and press **OK**.



6. Move to the **Characteristic** tab on the Multi Property dialog window.
7. Modify **Stiffness Coefficient** to **1000000**.
8. Modify **Damping Coefficient** to **100**.



To save the model:

Save the model as VibratingTransmission_RFlex.rdyn.

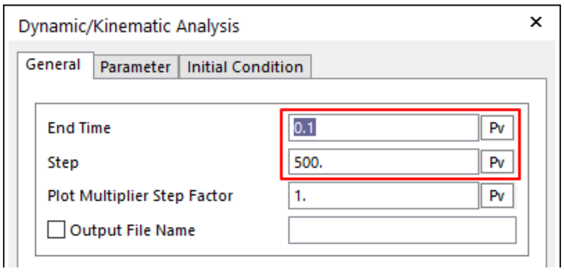
Performing Simulation

Run simulation to see if the model has been properly converted to a RFlex body.

To run the simulation:

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

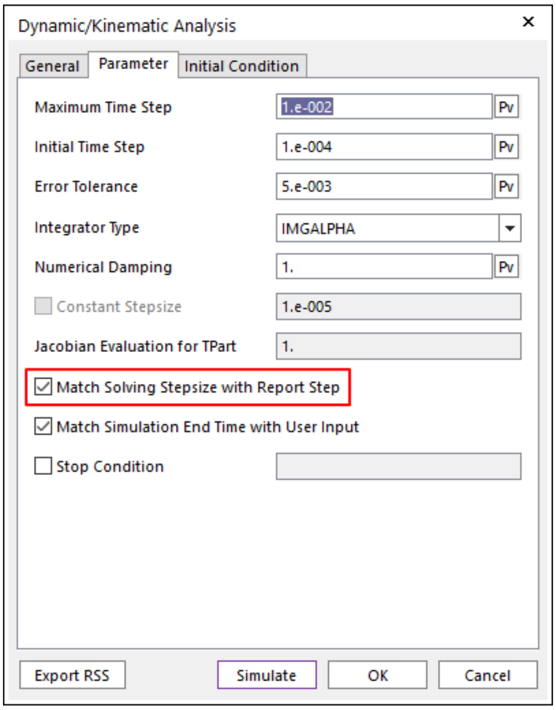
   The **Dynamic/Kinematic Analysis** dialog window appears.



2. Confirm the settings.
3. On the **General** page, set as follows:
   - **End Time**: 0.1
   - **Step**: 500

4. On the **Parameter** page, enable the option **Match Solving Stepsize with Report Step**.

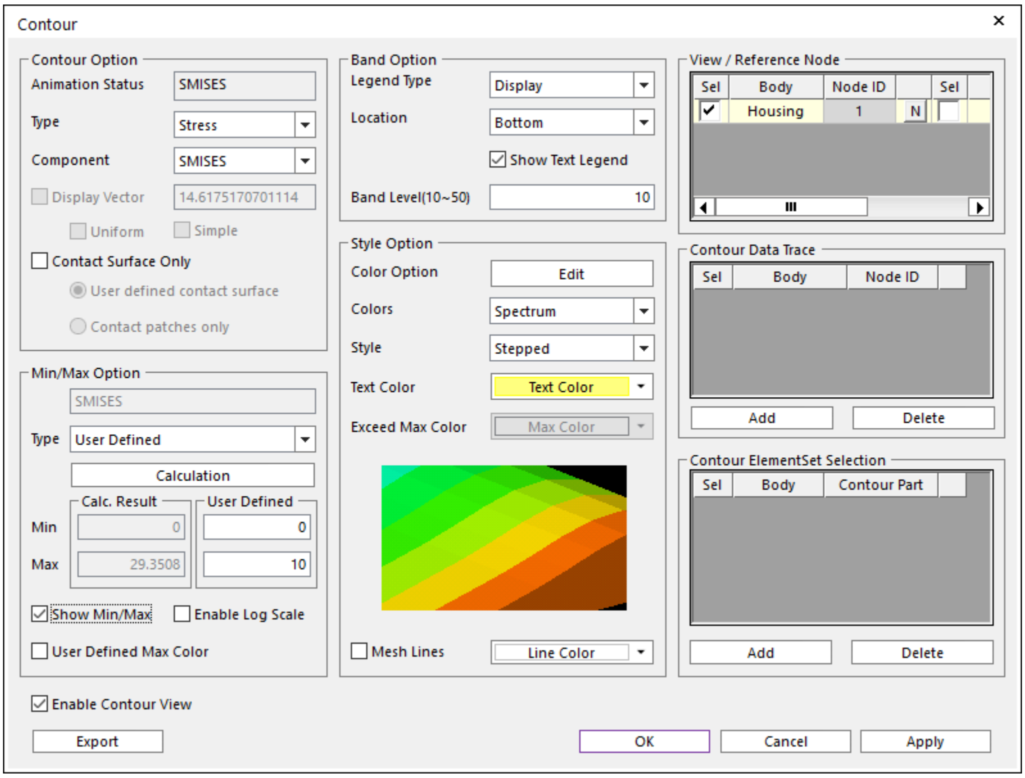
   In the next chapter, when calculating the FFT of the ERP, the data is divided at an equal interval. So you can use this option to produce better results.



5. Click the **Simulate** button.

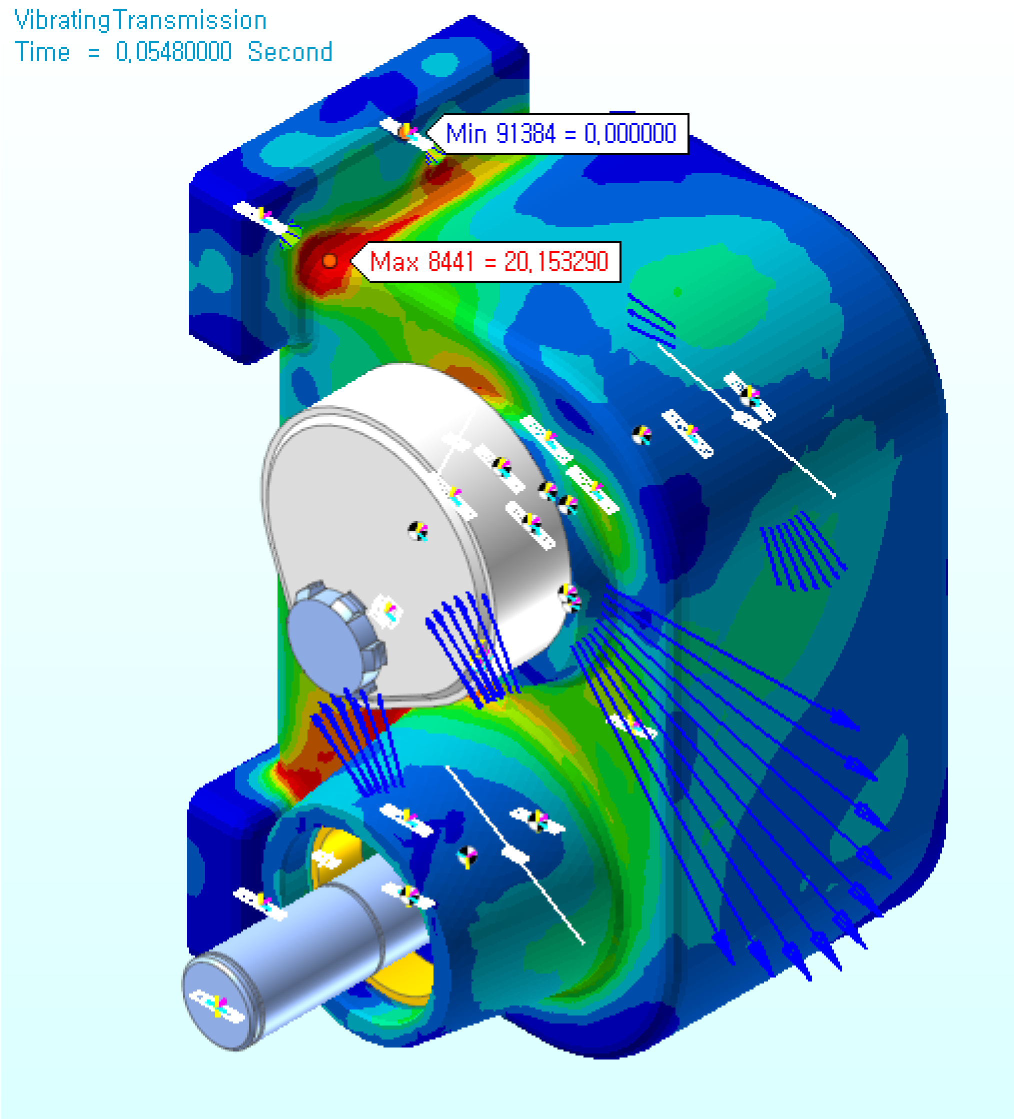
To view the stress contour result:

1. On the **Flexible** tab, in the **RFlex** group, click **Contour**.
2. In Contour Option, set the **Type** to **Stress**.
3. In Contour Option, set the **Component** to **SMISES**.
4. In Min/Max Option, set the **Type** to **User Defined**.
5. Click the **Calculation** button.
6. Change Max to **10**.
7. Select the **Show Min/Max** checkbox.
8. Click **OK**.



To view animation:

- On the **Analysis** tab, in the **Animation Control** group, click the **Play** button to check the result of stress. At **0.0548 seconds**, you can see that the result of **Maximum Von-Mises Stress** is approximately **20 Mpa**.



Chapter 4

Calculating Equivalent Radiated Power

The Acoustics analysis is a post tool and can be performed at any time with the analysis result of a flexible body (FFlex, RFlex). In the case of RFlex body, ERP can be additionally calculated for each mode, so you can analyze which mode affects it the most.

Task Objectives

In this chapter, you will learn how to calculate the ERP (equivalent radiated power) results using the functions provided by RecurDyn / Acoustics, and to derive the results through Scope and Contour.

Estimated Time to Complete This Task

15 minutes

Viewing Acoustics Result

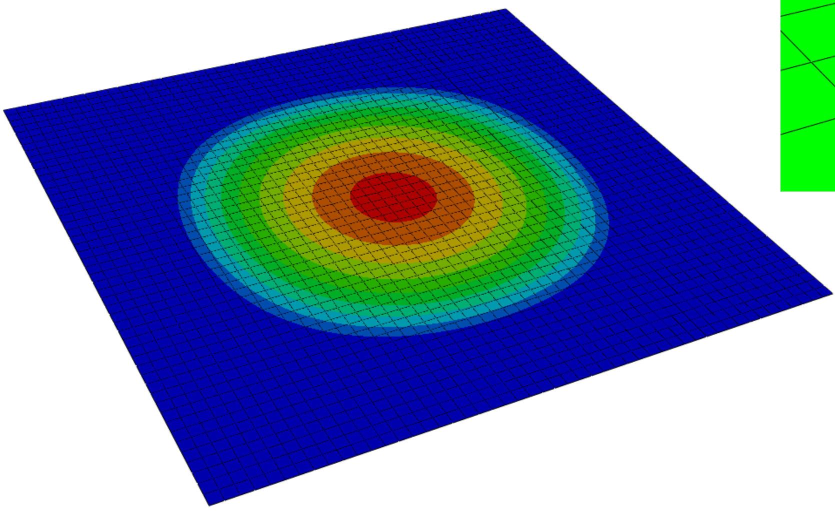
Acoustics analysis is carried out using the result simulated in Chapter 3. Before proceeding with the analysis, you should set the range for calculating ERP using a patch set.

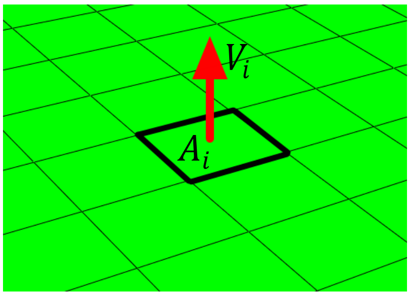
What is ERP (equivalent radiated power)?

One of the methods to analyze the frequency response of surface vibrations in a flexible body is to analyze the ERP results.
ERP is the sum of vibration energy generated from the surface as shown in the following equation.

$$\mathrm{ERP} = \mathrm{RLF} * \left(\frac{1}{2}\right) * \mathrm{C} * \mathrm{RHO} * \sum \left(A_i * V_i^{\,2}\right)$$

RLF: Radiation Loss Factor
C: Sound Velocity
RHO: Air Density
A_i: Surface Area of Each Element
V_i: Normal Velocity of Each Element.

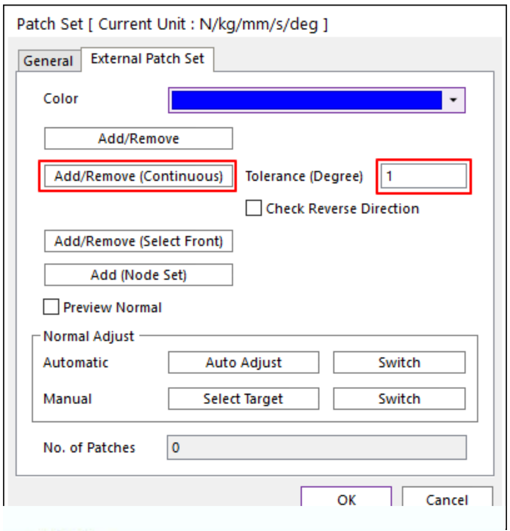




RecurDyn calculates the ERP by calculating the area and vertical velocity of each element for a specific patch set defined by the user. The ERP results for time domain and the results of using FFT (Fast Fourier Transform) as frequency domain are provided through Acoustic Scope. You can also check the ERP results using Contour animation.

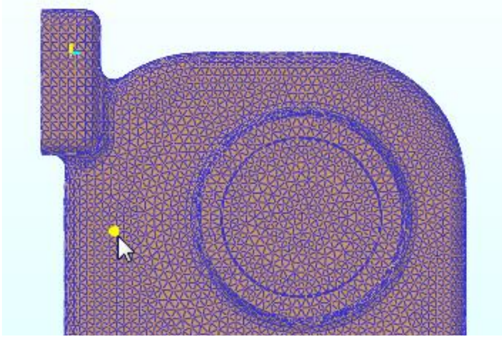
To create a patch set for Acoustics calculation:

1. Enter the RFlex Edit mode of the **Housing**.
2. In the **Set** group of the FR**Flex Edit** tab, click **Patch**.
3. In **Tolerance (Degree)**, enter **1**.
4. Click **Add/Remove (Continuous)** in the Patch Set dialog window.



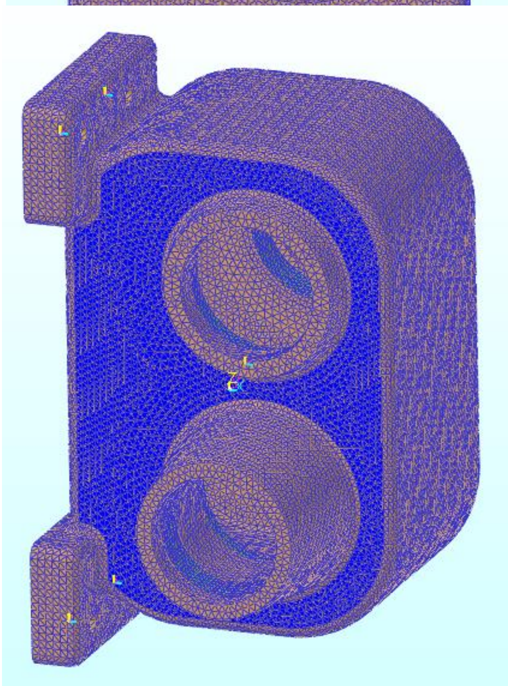
5. Click the side of the **housing** as shown on the right.

   The entire side is selected.
6. Right-click the working window and click **Finish Operation** in the pop-up menu.



7. Click **OK** in the Patch Set dialog window.

   SetPatch5 is created.



8. Click **Exit** to exit the **Edit Mode**.

To reload an animation:

In the process of creating the patch set, the relationship between the model and the animation is lost. Click **Reload the Last Animation file** button to retrieve the animation data to the model again.

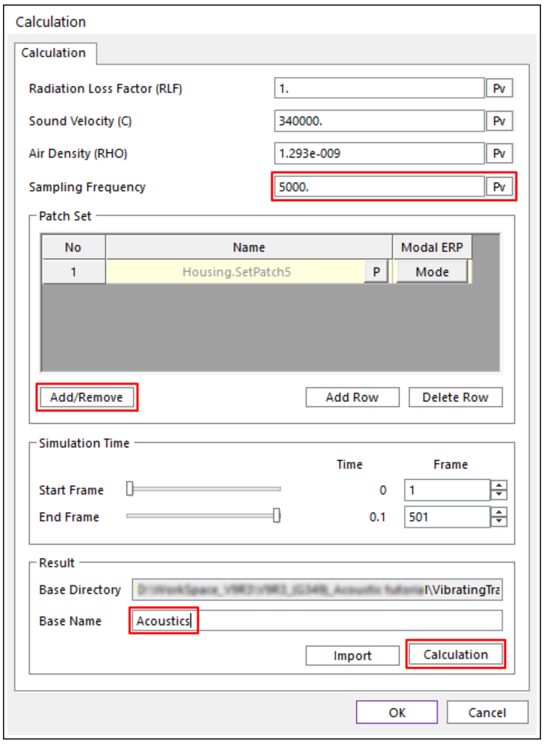
(If the reload is not possible, import the RAD or RAN file directly from the analysis result folder.)

To calculate Acoustics:

1. On the **Post Analysis** tab, in the **Acoustics** group, click **Calculation**.

   The **Calculation** dialog window appears.

   Since the simulation was performed at 500 steps per 0.1 second, you can extract the 5000 Hz result.



2. Modify **Sampling Frequency** to **5000**.

   Sampling frequency is used in the process of performing FFT for ERP.

3. In order to add a patch set, click the **Add/Remove** button.
4. In the working window, select **Housing.SetPatch5** created right before.
5. Right-click the working window and click **Finish Operation** in the pop-up menu.
6. Set the **Start Frame** to **1** and **End Frame** to **501**, which is the endmost value.
7. Set **Base Name** as **Acoustics**.
8. Click the **Calculation** button.
9. When the calculations are complete, click **OK** to close the dialog window.

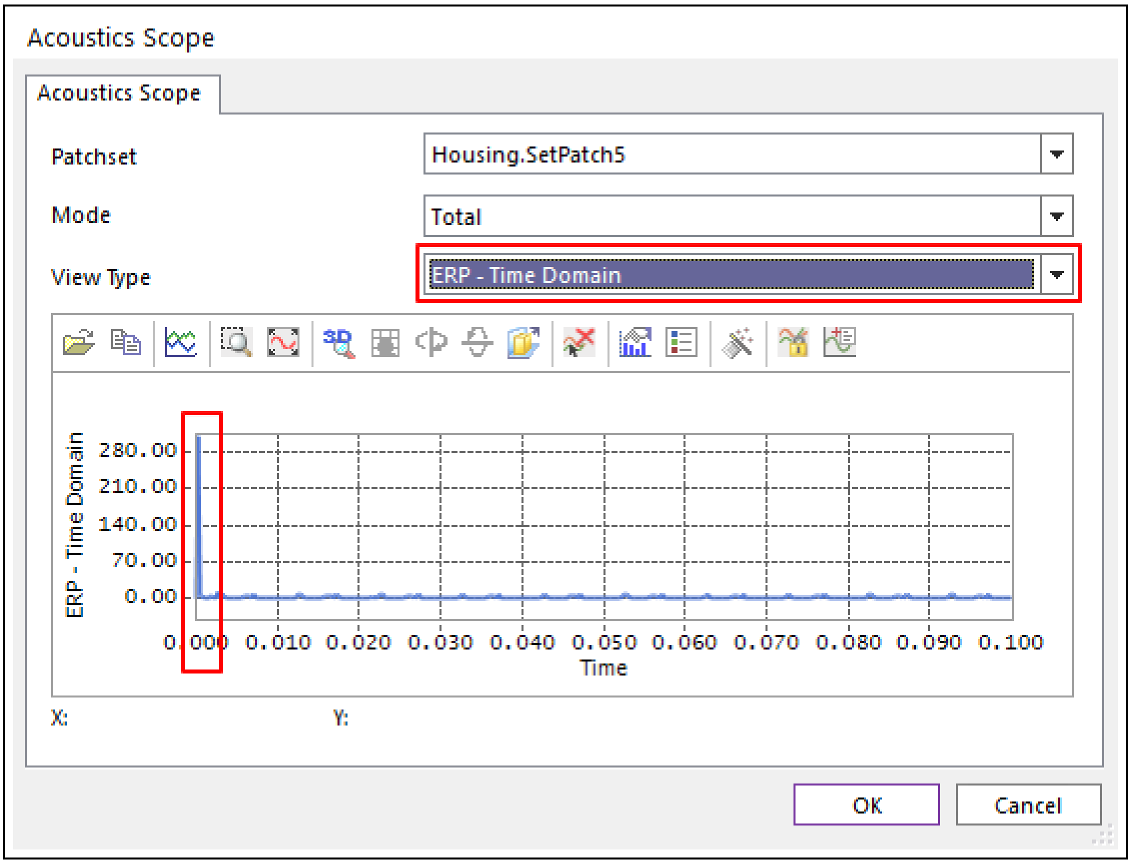
To view Acoustics scope:

1. On the **Post Analysis** tab, in the **Acoustics** group, click **Scope**.
2. Change the View Type to **ERP-Time Domain**.

   Scope shows the ERP calculation result of time domain for the whole housing.

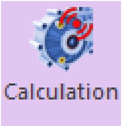
   There is a section that is not related to the periodic motion in the stabilizing phase between 0 and 0.005 seconds. In order to proceed with the FFT clearly, the ERP must be calculated again after removing the corresponding interval.

3. Click **Cancel** to close the Acoustics Scope dialog window.



Viewing Acoustics Results for Modal ERP

We will calculate the ERP for each mode of a RFlex body and analyze which mode has the greatest effect on the total ERP.

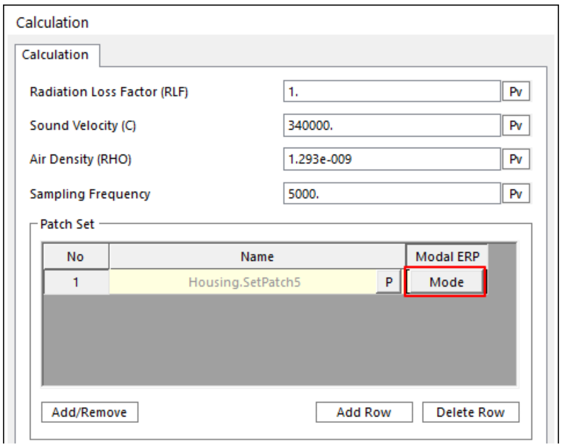


To recalculate Acoustics considering the modal ERP:

1. On the **Post Analysis** tab, in **Acoustics** group, click **Calculation**.

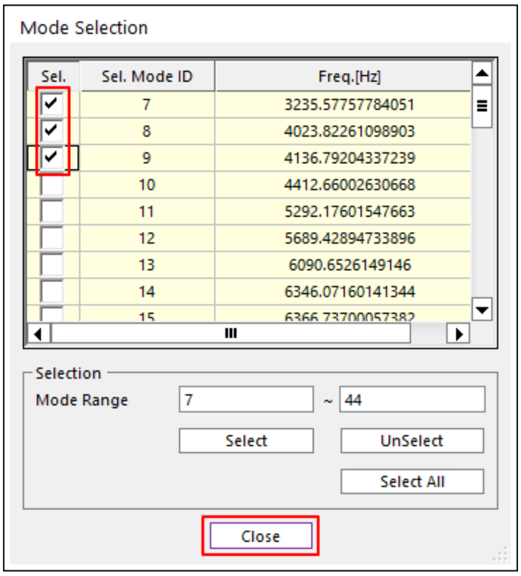
   The Calculation dialog window appears.

2. Click the **Mode** button of the Housing**.SetPatch5** previously selected.



   The Mode Selection dialog window appears.

3. Select the **Mode** numbers **7**, **8**, and **9** and click **Close**.

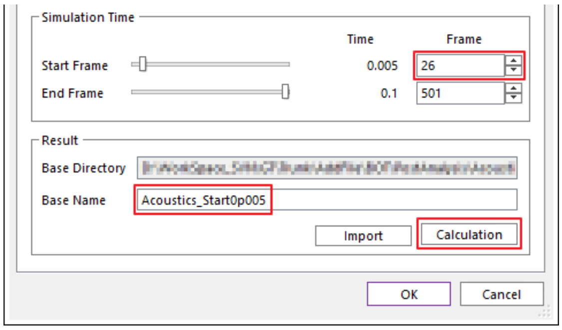


4. Set **Start Frame** to **26** to calculate starting from **0.005 second** to eliminate the interval that seems not related to the periodic motion between 0 second and 0.005 second.

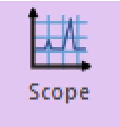
5. Change Base Name to **Acoustics_Start0p005**.

6. Click the **Calculation** button.

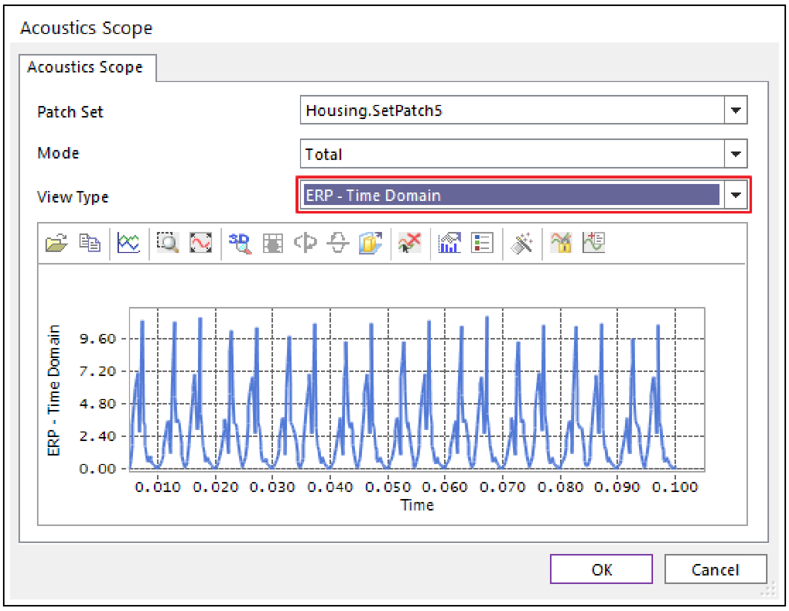
7. When the calculations are complete, click **OK** to close the dialog window.



To view Acoustics scope:

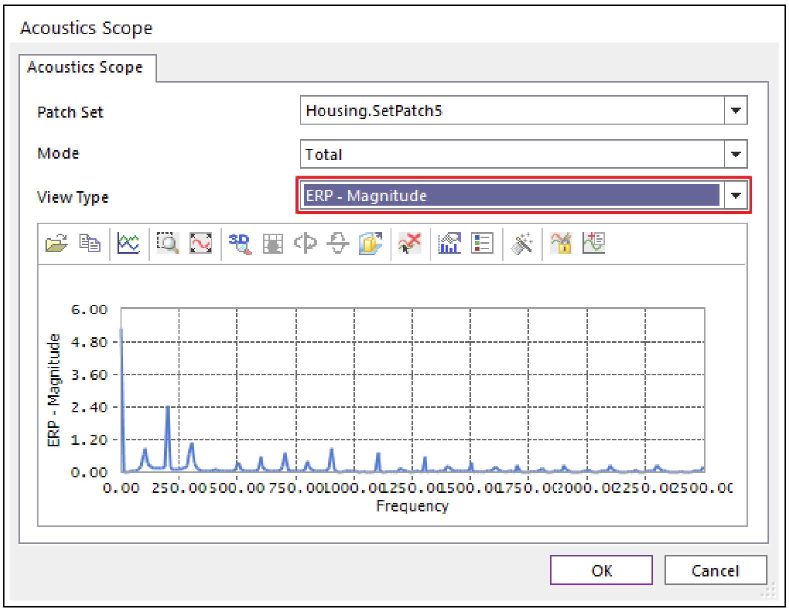


1. On the **Post Analysis** tab, in the **Acoustics** group, click **Scope**.
2. Change the View Type to **ERP-Time Domain**.



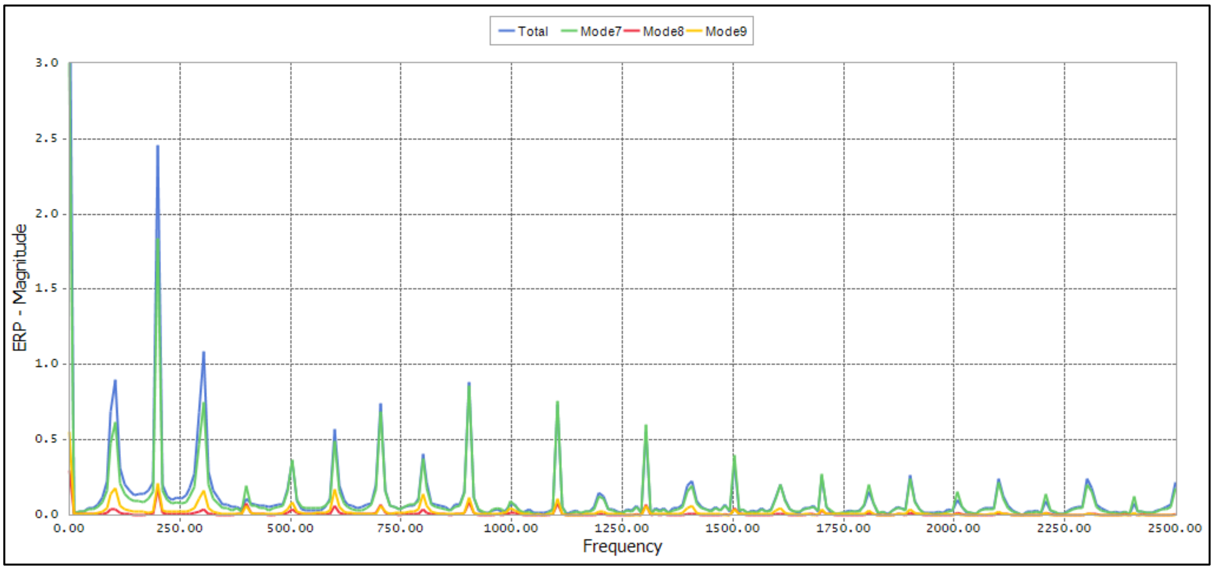
Time Domain ERP calculations are removed for up to 0.005 second, so they are well organized into the analytical data.

3. Change the View Type to **ERP-Magnitude**.



It can be seen that the harmonic frequency occurs for the 100Hz frequency.

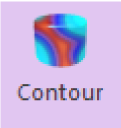
4. Change the mode to analyze the results for **Total**, **Mode7**, **Mode8**, and **Mode9**.



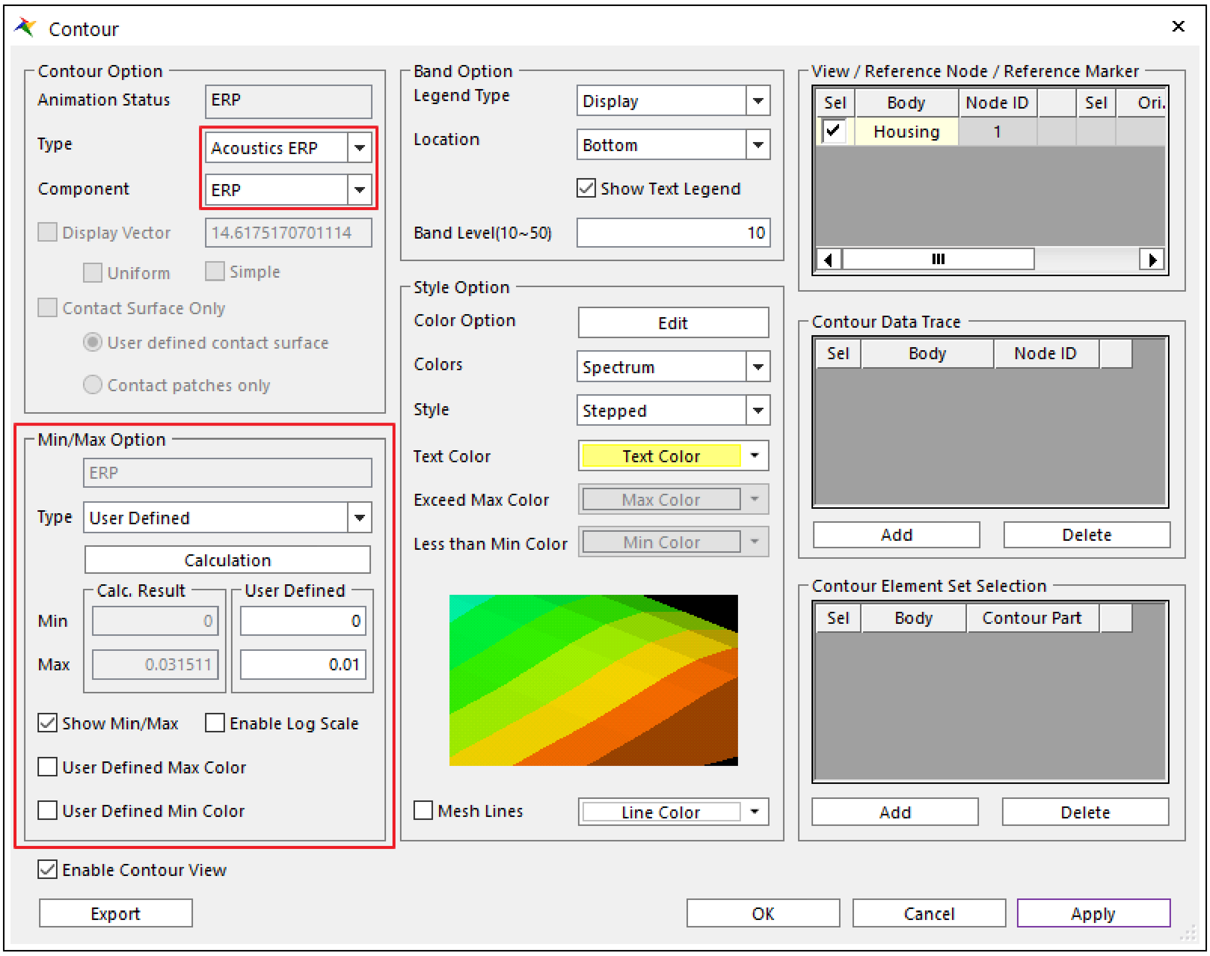
If **ERP-Magnitude** is plotted against **Total**, **Mode7**, **Mode8**, and **Mode9** at once, you can see that Mode7 has the greatest effect.

5. Click **Cancel** to close the Acoustics Scope dialog window.

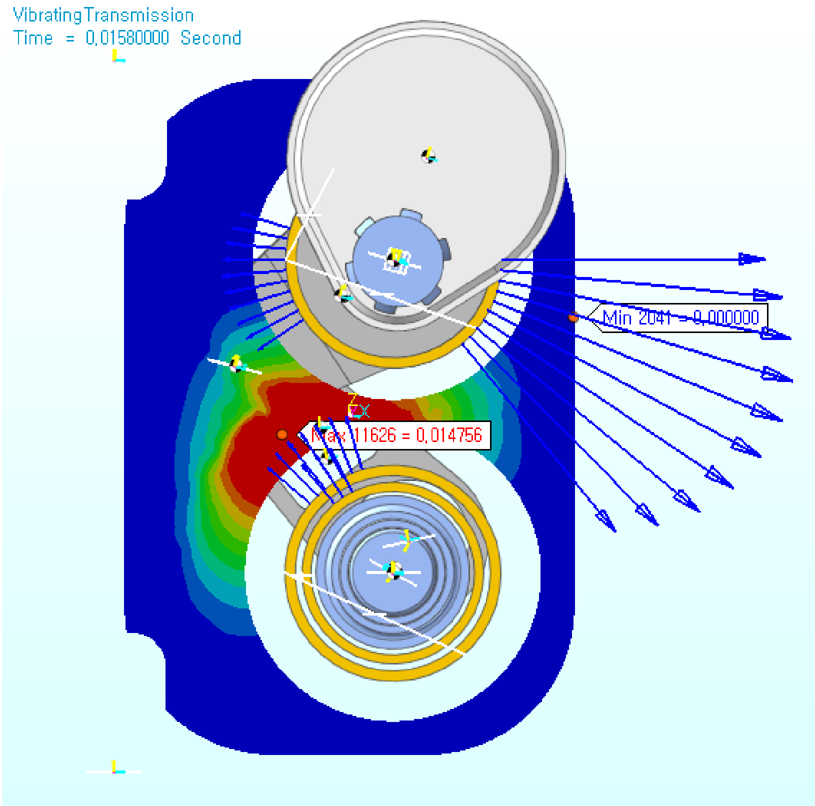
To view the Acoustics Contour result:



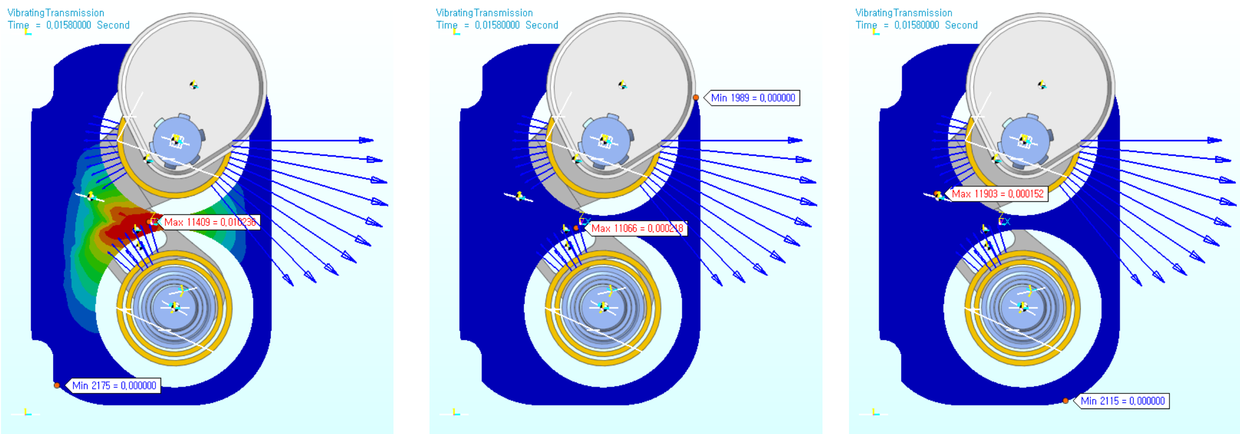
1. On the **Post Analysis** tab, in the **Acoustics** group, click **Contour**.
2. Set the Type to **Acoustics ERP**.
3. Set the Component to **ERP**.
4. In Min/Max Option, set the Type to **User Defined**.
5. Click the **Calculation** button.
6. Set Max to **0.01**.
7. Select the **Show Min/Max** checkbox.
8. Click **OK**.



9. On the **Analysis** tab, in the **Animation Control** group, click the **Play** button to check the result of ERP. The Contour result shows that the ERP value is biggest in the middle part.



10. Again, open the Contour dialog window and draw an ERP Contour for **other Modes**.



The ERP_HOUSING_MODE7 result is similar to the Total result, and the rest of the modes is considered to have no significant impact.

Chapter

5

Modifying and Analyzing the Model

Task Objectives

We will try to improve the model so that the noise vibration generated in the housing can be reduced.

Estimated Time to Complete This Task

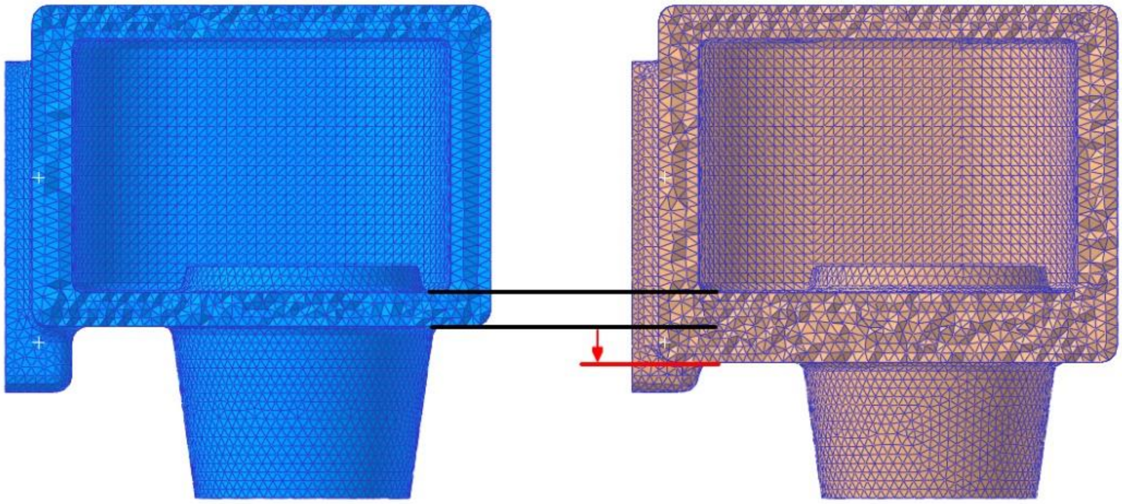
15 minutes

Replacing the Housing

To replace with a RFlex body:

Repeat the procedure for swapping with the RFlex body that you carried out in **Chapter 3**. For the RFlex body to replace the housing, use the file **Housing_Modified.rfi** in the folder copied in **Chapter 2**.

The thickness of the housing has been modified to reduce noise on the sides. Comparing it with the existing housing produces the following figure.



To redefine contact:

Repeat the procedure in **Chapter 3** to redefine contact.

To save the model:

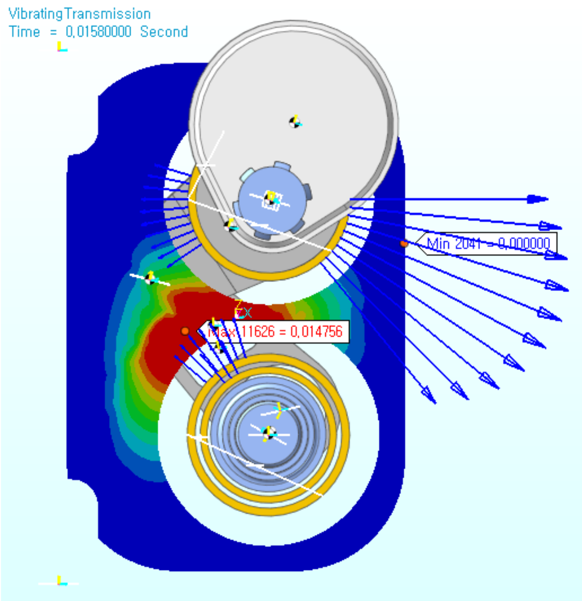
Save the model as **VibratingTransmission_RFlex_NewHousing.rdyn**.

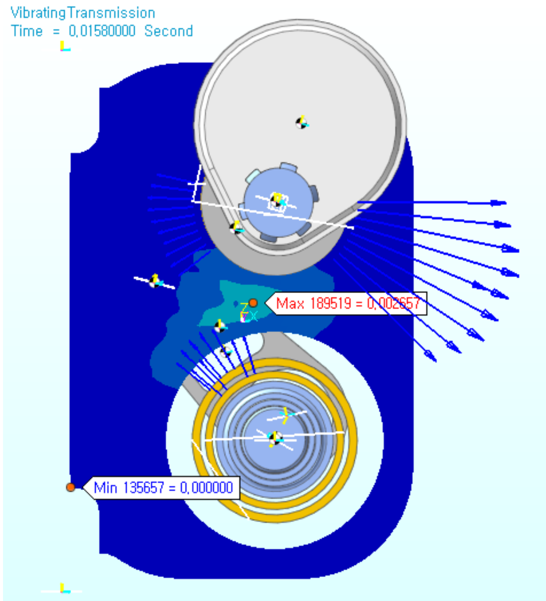
To perform simulation after saving the model:

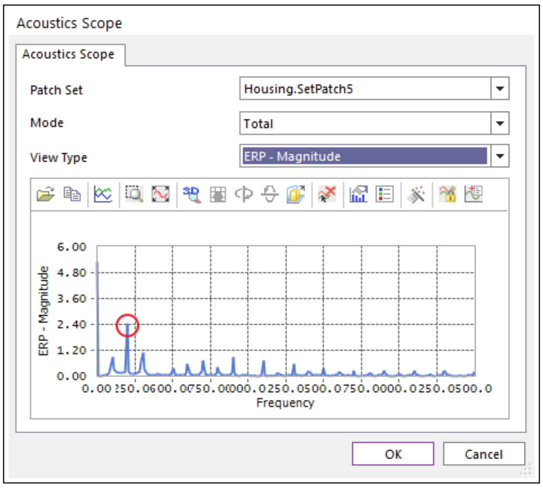
Execute the simulation in the same conditions as those simulated in **Chapter 4**.

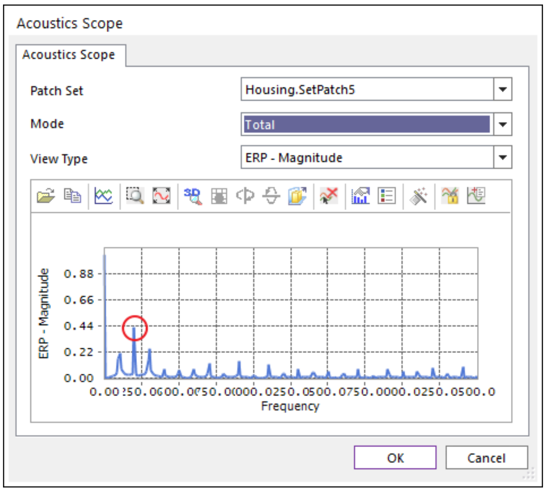
Viewing the Acoustics Result

Repeat the Acoustics calculation procedure in **Chapter 4** to calculate the ERP for the side of the housing again.









Note that the ERP-Magnitude value at 200 Hz is reduced from 2.4 to 0.42.

Thanks for participating in this tutorial!