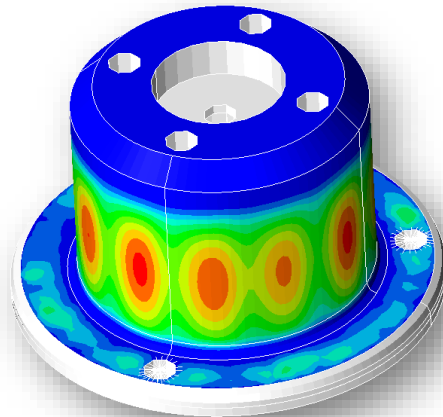
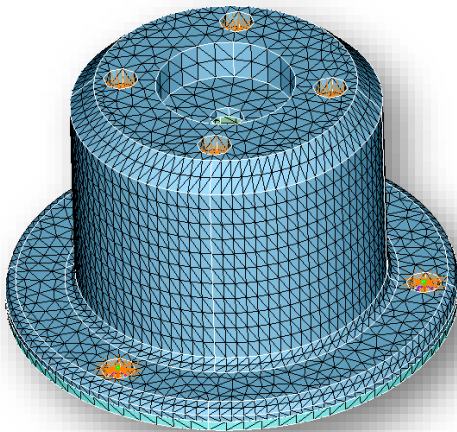


Modal frequency response analysis using the electromagnetic vibration loads from Flux

- This tutorial is based on the **SimLab** version **v2019.2**. Other software versions may have inconsistent interfaces or some compatibility issues.
- In the following tutorial, the user will be able to define 1D bolts, contacts, loads and constraints and setup the solver execution in order to perform a modal frequency response analysis in SimLab.
- This tutorial starts from the assembly created in "SimLab: mesh export"

In this lesson you will learn how to:

- Define materials and properties
- Define 1D bolts
- Defining loads and constraints
- Define contacts
- Create node sets
- Run a modal frequency response analysis with OptiStruct

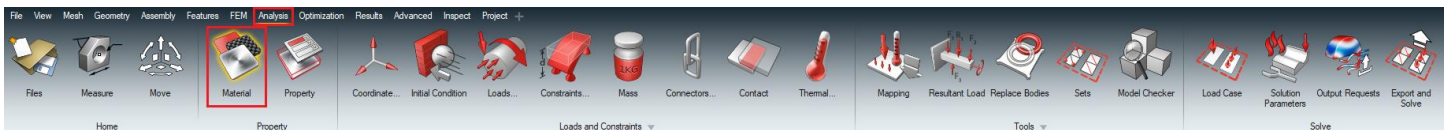


Step1: Open a SimLab file.

This tutorial is started from the meshed model created in the previous tutorial: "SimLab: mesh export".

Step2: Create Materials for the motor bodies

1. Head to the **Analysis** menu and then click on **Material**.



As several materials are already created in the project, two materials will be updated for the modeling, steel and aluminum.

2. Double click on the materials Steel and Aluminium, update the properties of, click on **OK**.

Material: Steel

Material ID: 3

Category: Solid

Class: Metal

Model: Isotropic

Name	Value	Table
Mechanical Properties		
Elastic		
Density	7.8E-09	none
Youngs_modulus	210000	none
Poissons_ratio	0.3	none
Shear_modulus		
Thermal_Expansion	1.13E-05	none
Reference_Temperature		
Damping_coefficient		
Stress_Tension		
Stress_Compression		
Stress_Shear		
Mat_Cord_Sys		
Plastic		
Visco Plastic		
Thermal Properties		
Magnetic Properties		
Fatigue Properties		

Description:

OK Cancel

Material: Aluminium

Material ID: 1

Category: Solid

Class: Metal

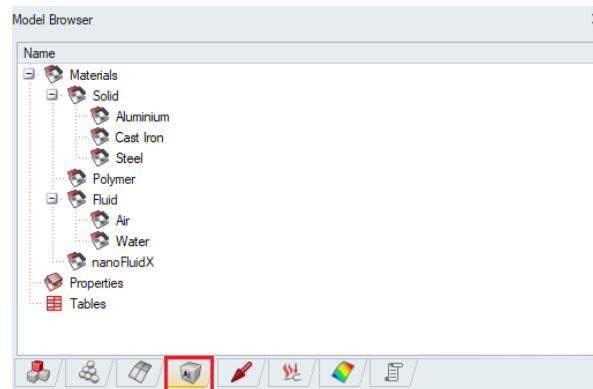
Model: Isotropic

Name	Value	Table
Mechanical Properties		
Elastic		
Density	2.77E-09	none
Youngs_modulus	70000	none
Poissons_ratio	0.33	none
Shear_modulus		
Thermal_Expansion	2.28E-05	none
Reference_Temperature		
Damping_coefficient		
Stress_Tension		
Stress_Compression		
Stress_Shear		
Mat_Cord_Sys		
Plastic		
Visco Plastic		
Thermal Properties		
Magnetic Properties		
Fatigue Properties		

Description:

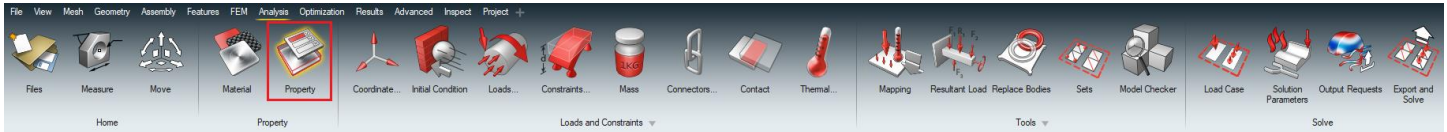
OK Cancel

3. To review the materials we just created, you can click on their names in the *Property* Tab of the Model Browser.

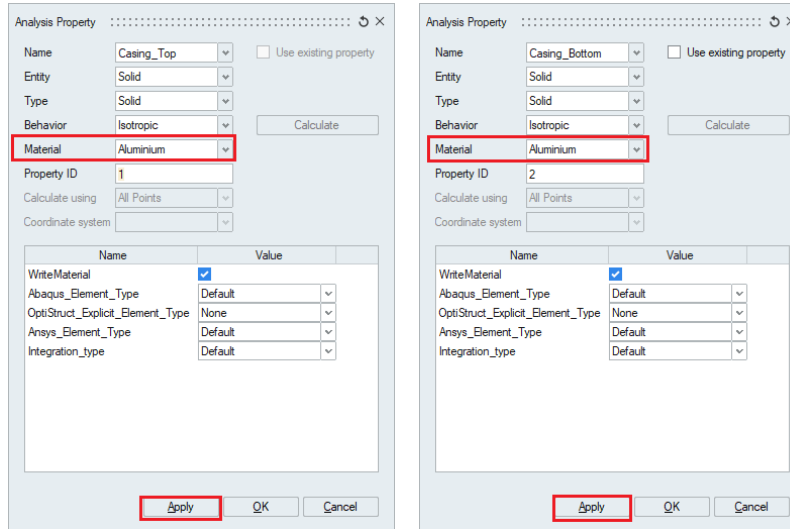


Step3: Define Properties

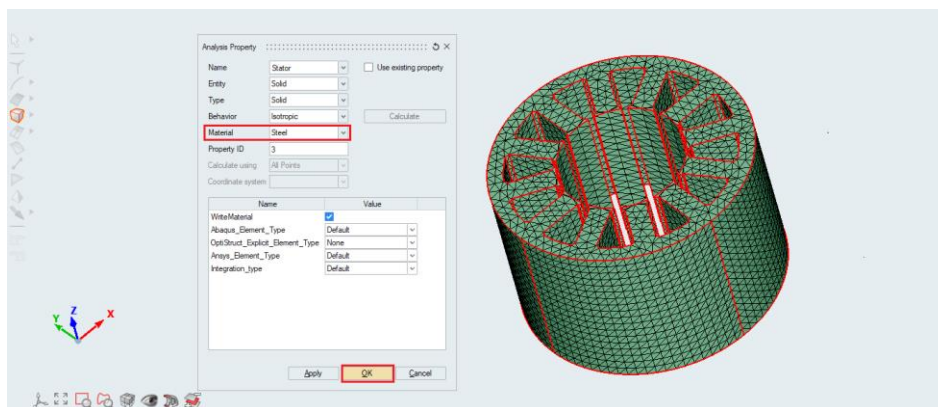
- Next, the materials will be assigned to the bodies through Properties. Open the **Analysis | Property** menu.



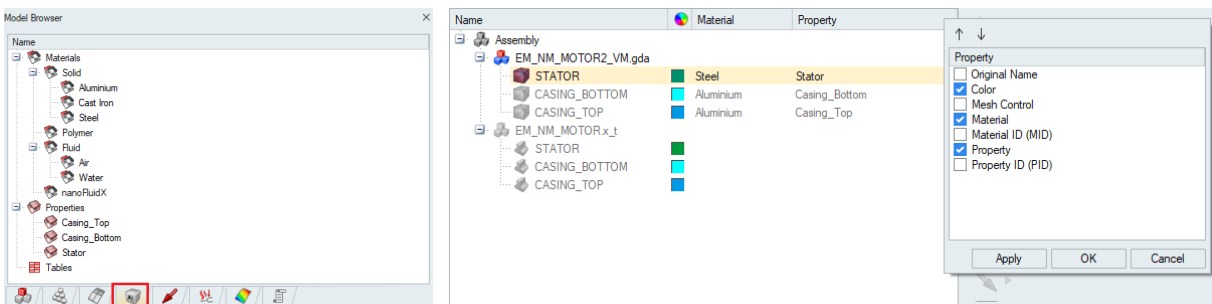
- In the Property dialog, enter Name as *Casing_Top*. Verify that the Entity is Solid, Type is Solid and Behavior is Isotropic. Choose Material as **Aluminum**, select the *CASING_TOP* body in the display area and click on **Apply**. Next in the same dialog, enter Name as *Casing_Bottom*. Verify that the Entity is Solid, Type is Solid and Behavior is Isotropic. Choose Material as **Aluminum**, select the *CASING_BOTTOM* body in the display area and click on **Apply**.



- Next in the same dialog, enter Name as *Stator*. Verify that the Entity is Solid, Type is Solid and Behavior is Isotropic. Choose Material as **Steel**, select the *STATOR* body in the display area and click on **OK**.







- The materials and properties will be listed in the Property tab of the Model Browser, and optionally in the Assembly tab next to the bodies (turn on the option by right-clicking on the upper bar).



Step4: Define 1D bolts

SimLab creates 3 (or 4) different LBC entities for each 1D pretension Bolt, as the following manners:

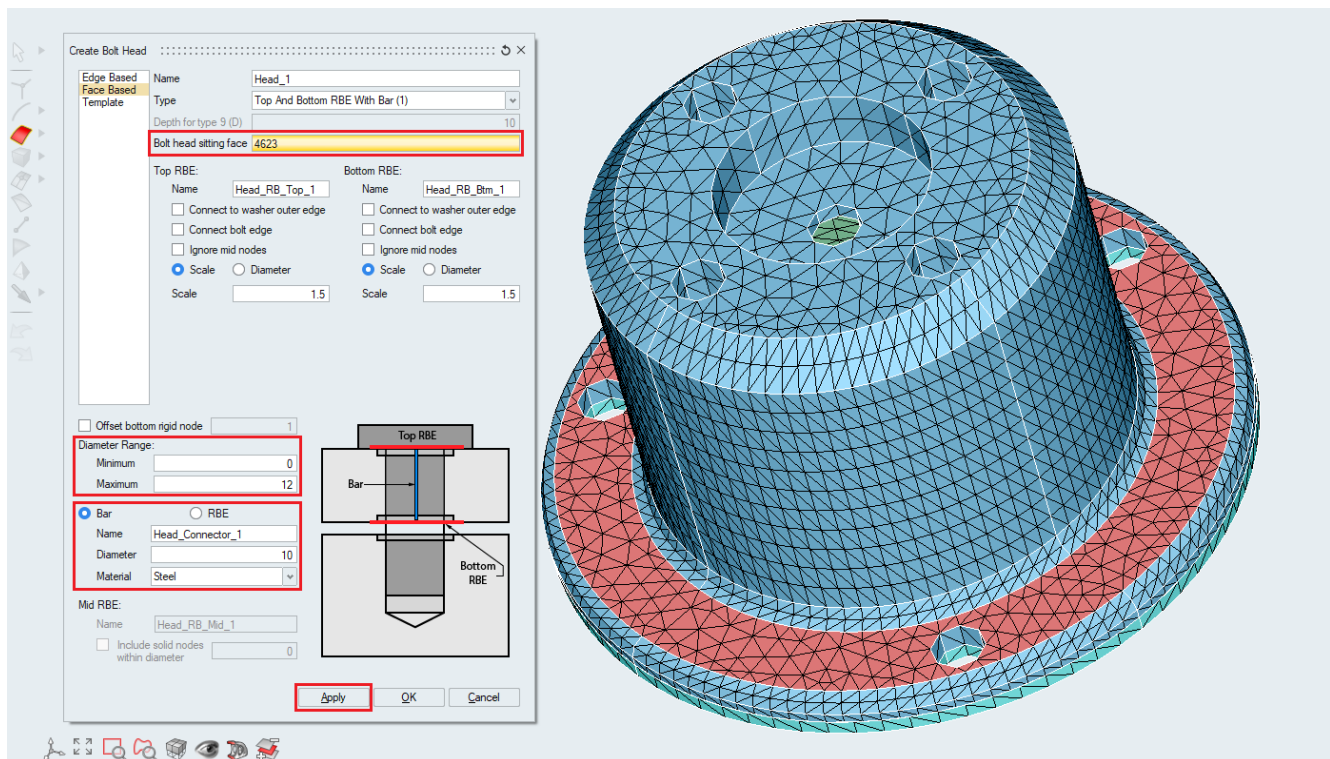
-  An **Automation Object** ("BPT1"): allows to edit all parameters of the 1D Bolt. It generates also a...
 -  **Pretension Object** ("BarPretension1"): it shows only the parameter related to the pretension force. It generates finally the LBCs which have to be actually included in the LoadCases, which are...
 -  A **Pretension Force** ("BarPretension1_Force")
 -  A **Pretension Constraint** ("BarPretension1_Const") would be created only if the **Lock Option** is activated; in our case we have no Lock defined, that's why the constraint is not defined.

The different objects are linked to each other. Once one of them is edited, all parent and children entities will be updated automatically. The most common way is to edit the "highest" entity (Automation Object, BPT1).

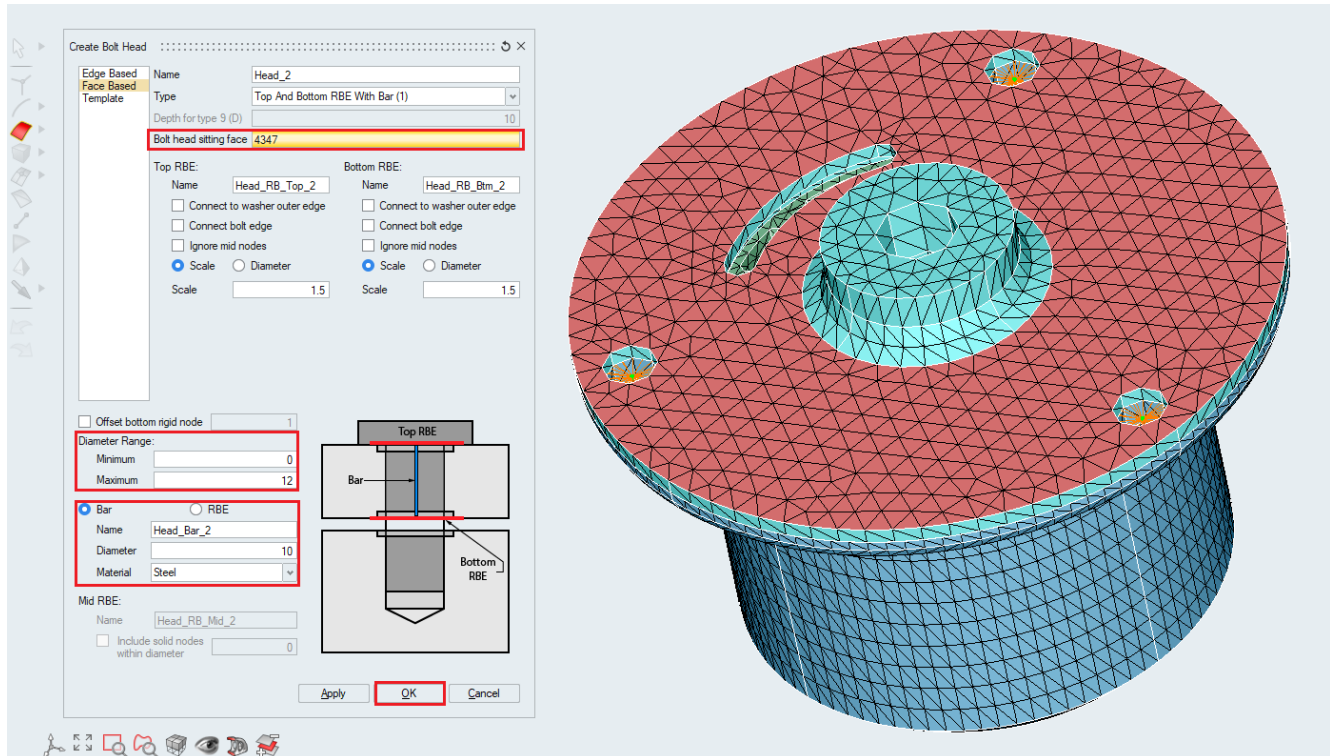
- Go to the **Advanced** tab, located in the menu bar, then click on **Bolt Modeling** to open the bolt modeling tools. Next, click on the *Bolt Head* icons



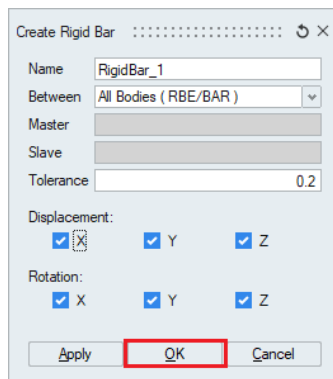
- Select the Bolt head sitting face on the CASING_TOP body as shown in the image. An *RBE* with the Radius Scale of 1.5 for both Top and Bottom RBE is predefined. For the diameter range enter minimum as 0 and maximum as 12. Enter Bar diameter of 10 [mm] and select material as *Steel*. Click on **Apply**.



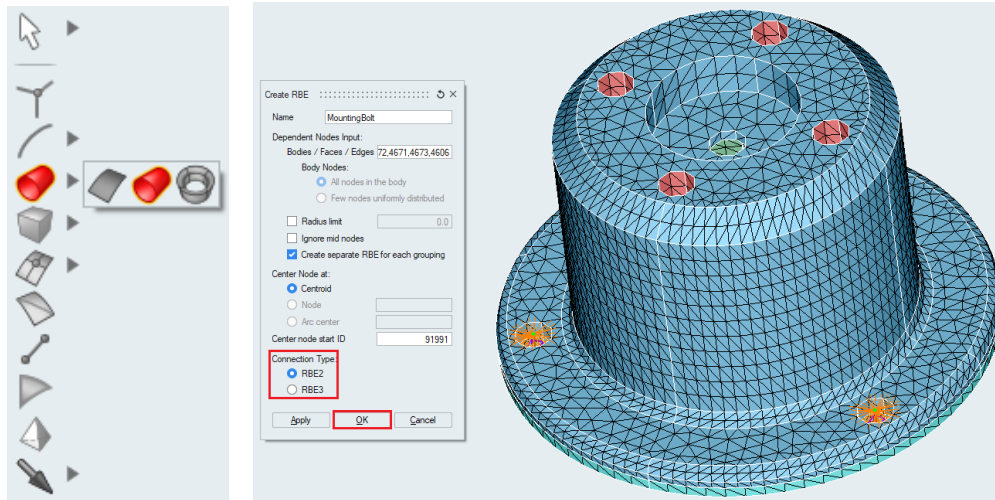
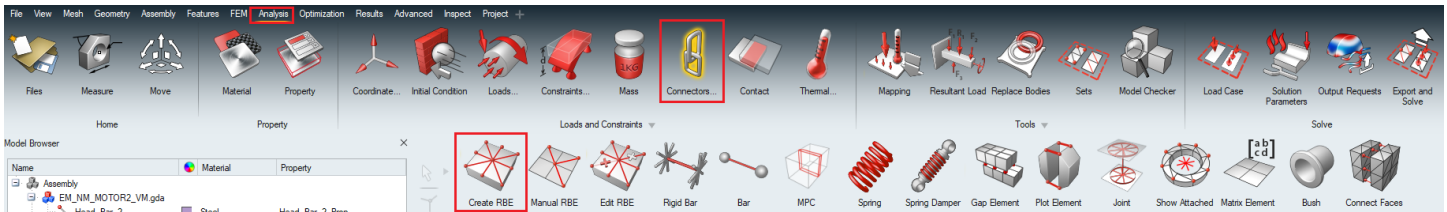
3. Next, select the Bolt head sitting face on the CASING_BOTTOM body as shown in the image. An RBE with the Radius Scale of 1.5 for both Top and Bottom RBE is predefined. For the diameter range enter minimum as 0 and maximum as 12. Enter Bar diameter of 10 [mm] and select material as *Steel*. Click on **OK**.



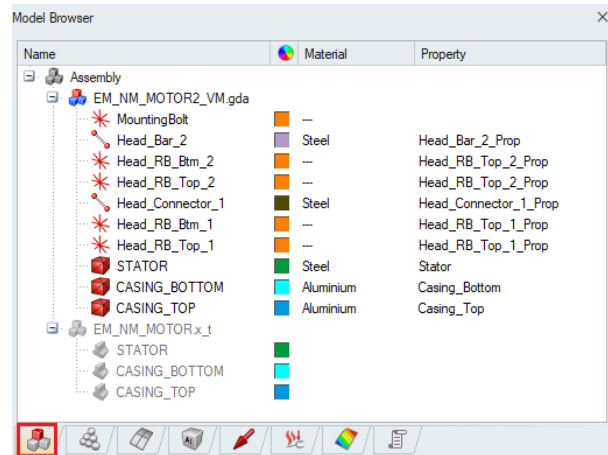
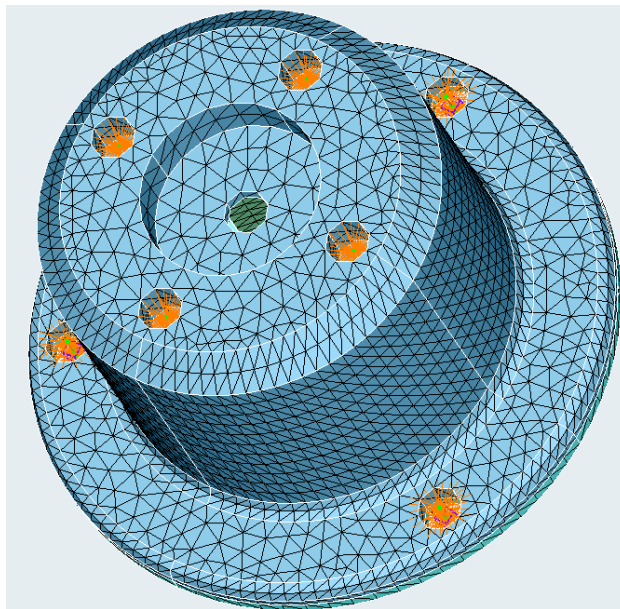
4. To connect the two bolt heads, a rigid connection will be created. To do this, go to **Advanced| Bolt Modelling| Connections| Rigid Bar**. Choose *All Bodies (RBE/RBAR)* for Between option, enter Tolerance as 0.2, select all check boxes to restrict displacement and rotation in all direction and click on OK.



5. Next, RBE2 will be created for the bolt holes on the top of the CASING_TOP body. Head to **Analysis | Connectors | Create RBE**. In the Create RBE dialog, enter name as *MountingBolt*. Switch from Face selection to Cylinder (Face) selection in the entity selection toolbar. Select the four mounting bolts on the top of the CASING_TOP body. Choose the connection type as RBE2 and click on **OK**.

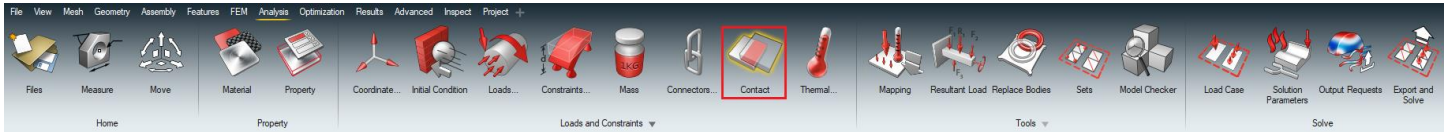


6. So far, all the RBEs have been create now. The Assembly tree should look like below.



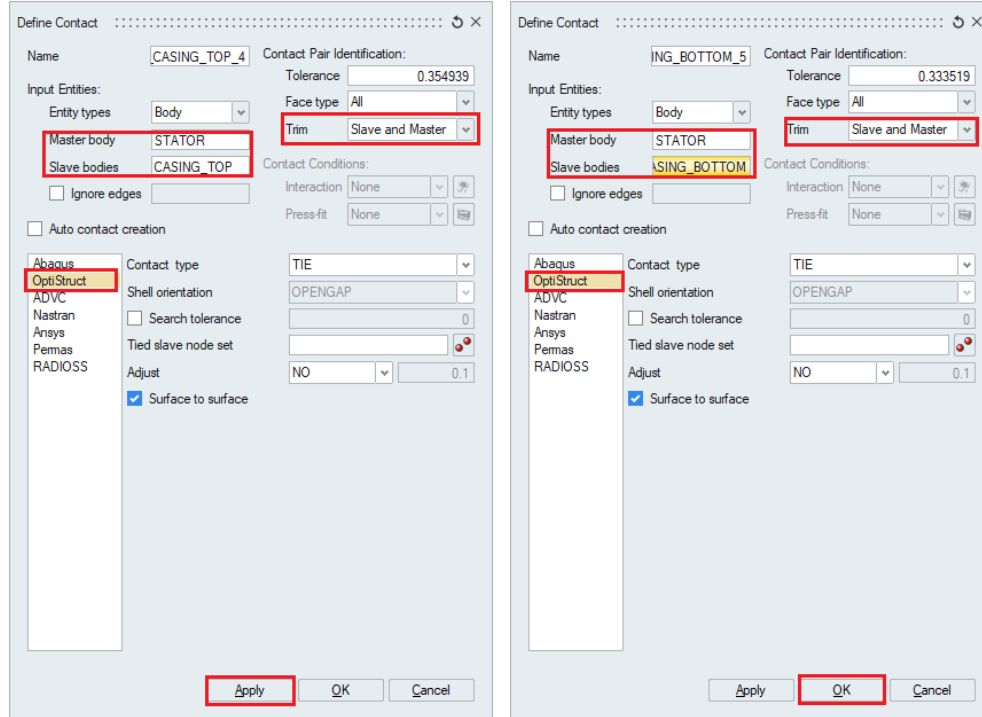
Step5: Define contacts.

1. To define contacts, click on **Contact** in the **Analysis** ribbon.



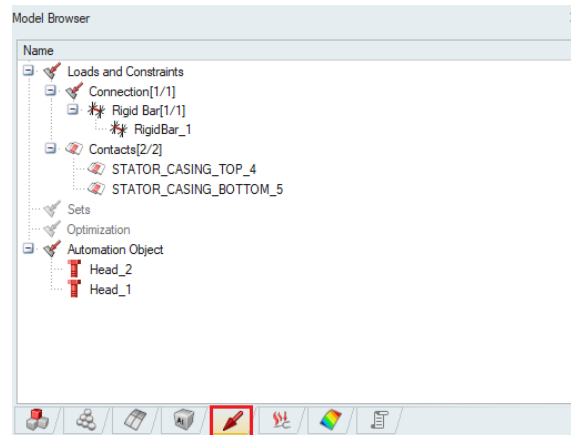
2. Select OptiStruct as the solver.
3. Bodies will be used as *Entity types*. Select *STATOR* as the *Master* and *CASING_TOP* as the *Slave*.
4. Choose All as the *Contact Face type*.
5. Choose TIE as the *Contact type* and click on **Apply**. The contact is created and appears in the LBC Browser.

Use the image below as reference for the rest of the parameters (tolerance is calculated automatically).



6. In the same dialog, select OptiStruct as the solver.
7. Bodies will be used as *Entity types*. Select *STATOR* as the *Master* and *CASING_BOTTOM* as the *Slave*.
8. Choose All as the *Contact Face type*.
9. Choose TIE as the *Contact type* and click on **OK**. The contact is created and appears in the LBC Browser.

Use the image above as reference for the rest of the parameters (tolerance is calculated automatically).
Once the definition is finished, switch to the Loads and Constraints tab to verify the contacts.

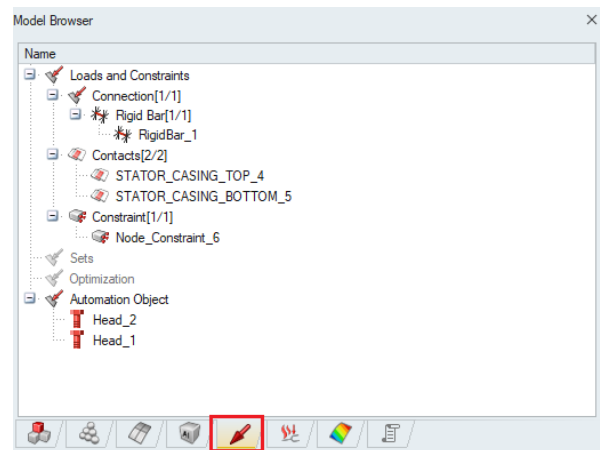
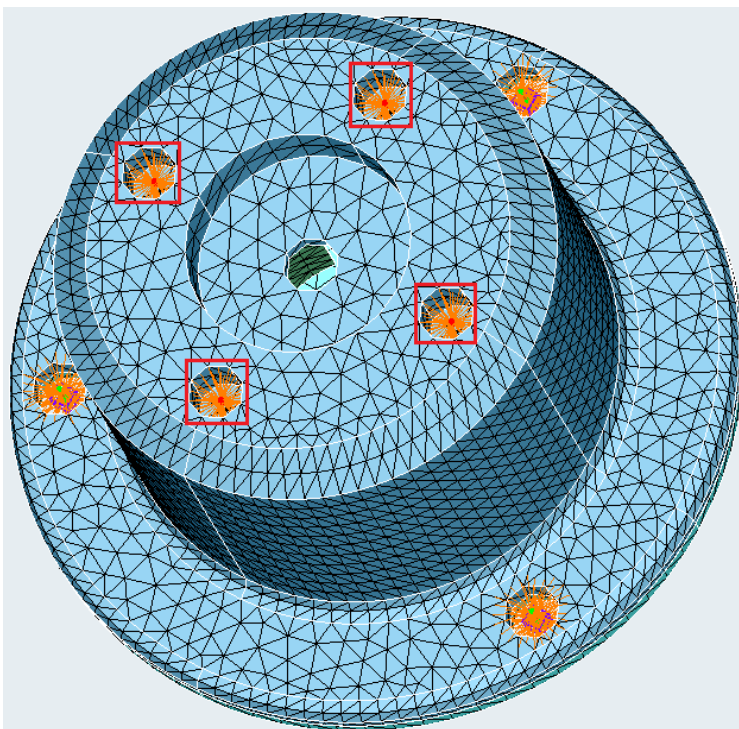
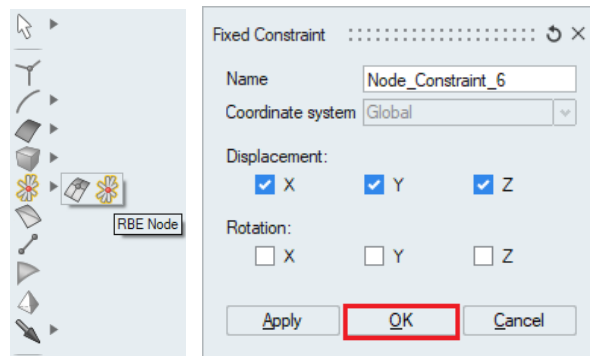


Step6: Establish constraints

1. To establish constraints, click on the **Analysis | Constraints** icon, then on **Fixed** to create a fixed constraint on the *MountingBolt* RBE center nodes.



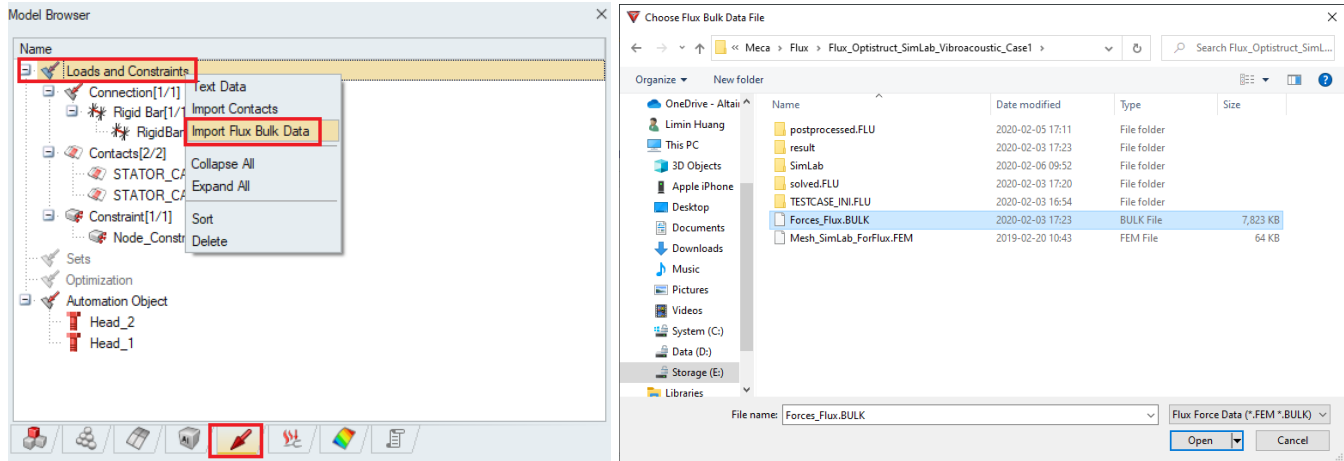
2. Switch from Face selection to RBE node selection in the entity selection toolbar. Select the RBE center nodes of the *MountingBolt* RBE. In the Fixed Constraint dialog, enter name as *Constraint*, select the *Displacement* for all axes, then click on **OK**. A *Constraint* object appears in the *LBC Tree* of the Browser.



Step7: Import load distribution from Flux

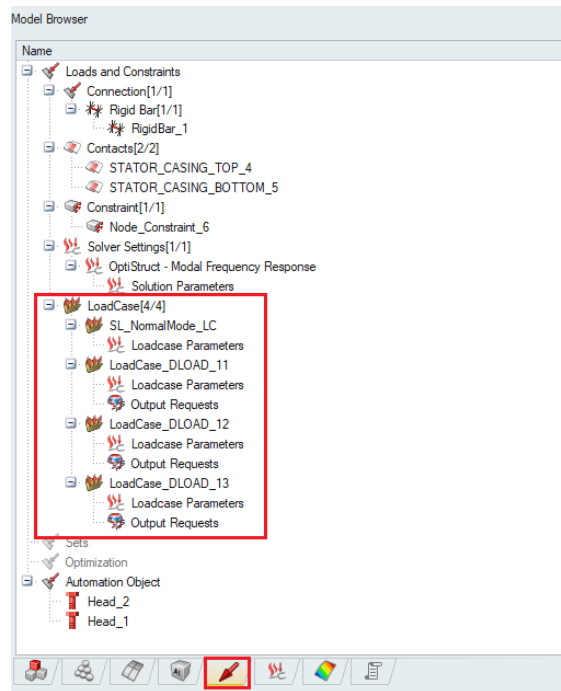
Electromagnetic analysis is done in Flux and loads were mapped on the mesh of the Stator tooth faces of the motor exported from SimLab. Exercise "SimLab: mesh export" refers to the creation of stator tooth faces. Note that the stator tooth face nodes are having the same IDs as that of the existing mesh.

1. Head to the Loads and Constraints tab in the Model browser.
2. Right-click on Loads and Constraints and choose *Import Flux Bulk Data* from the sub menu.



Navigate to the Tutorial-folder and open the file **Forces_Flux.BULK**.

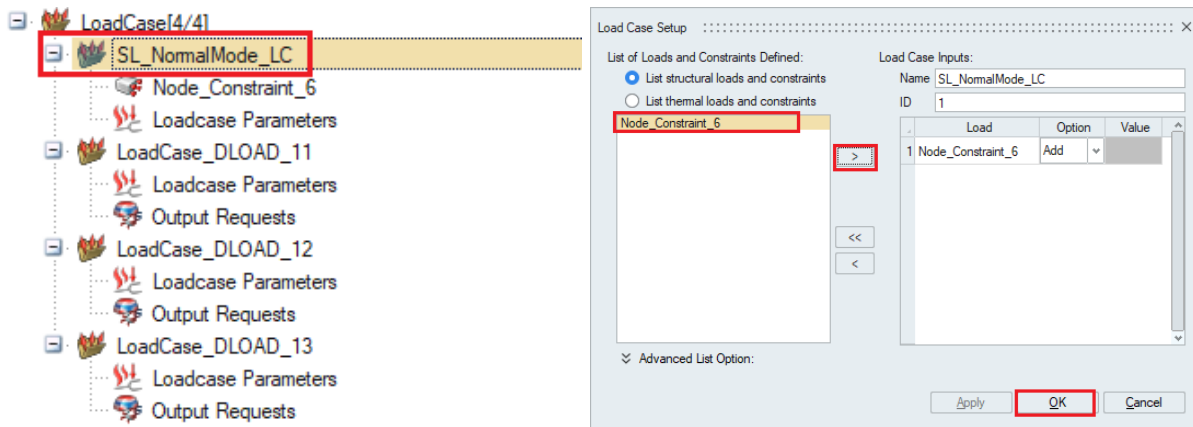
On importing the **Forces_Flux.BULK** file, four load cases are created automatically.



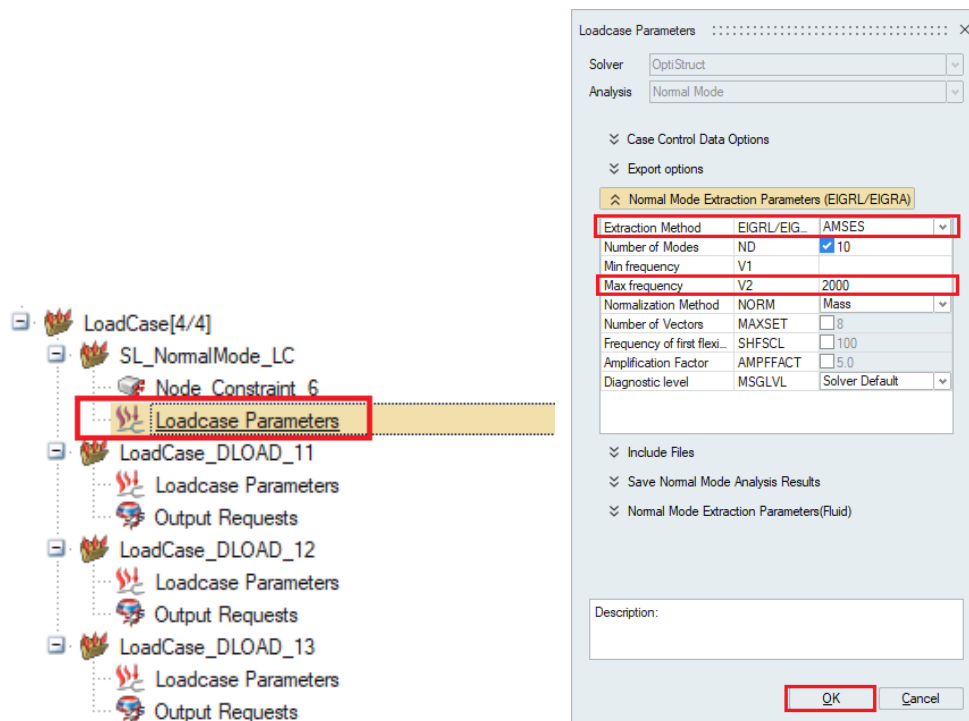
This is an automated step in which 4 load cases are created.

- 1st Load case = SL_NormalMode_LC is for running a normal modes analysis
- 2nd Load case = Loadcase_DLOAD_11 is for running Modal Frequency response analysis using the Radial loads acting on the Stator Tooth face exported from Flux
- 3rd Load case = Loadcase_DLOAD_12 is for running Modal Frequency response analysis using the Tangential loads acting on the Stator Tooth face exported from Flux
- 4th Load case = Loadcase_DLOAD_13 is for running Modal Frequency response analysis using the Global loads (Radial+Tangential) acting on the Stator Tooth face exported from Flux

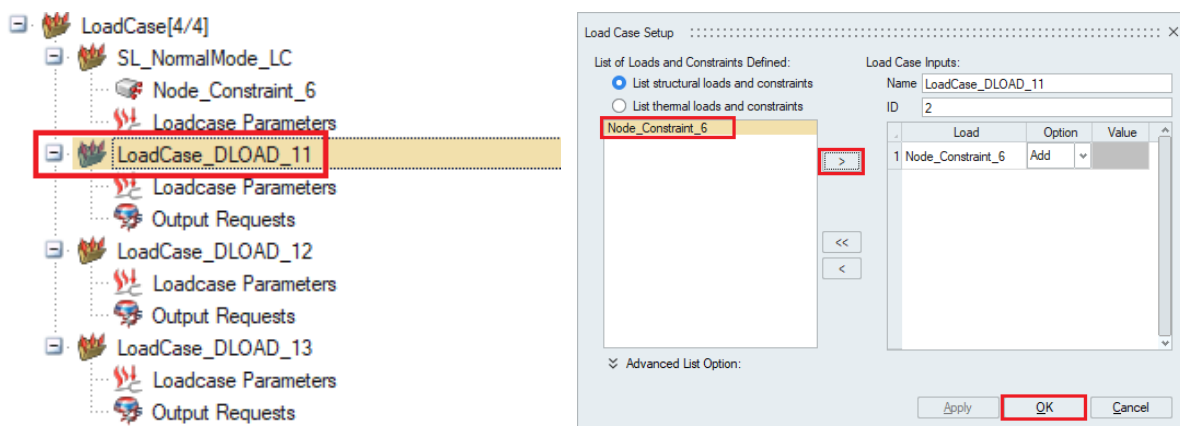
3. In the LBC tree, double click on the load case *SL_NormalMode_LC*. This will open the Load Case Setup Dialog. Choose *Constraints* and add it to the Load Case Inputs. Click **OK**.



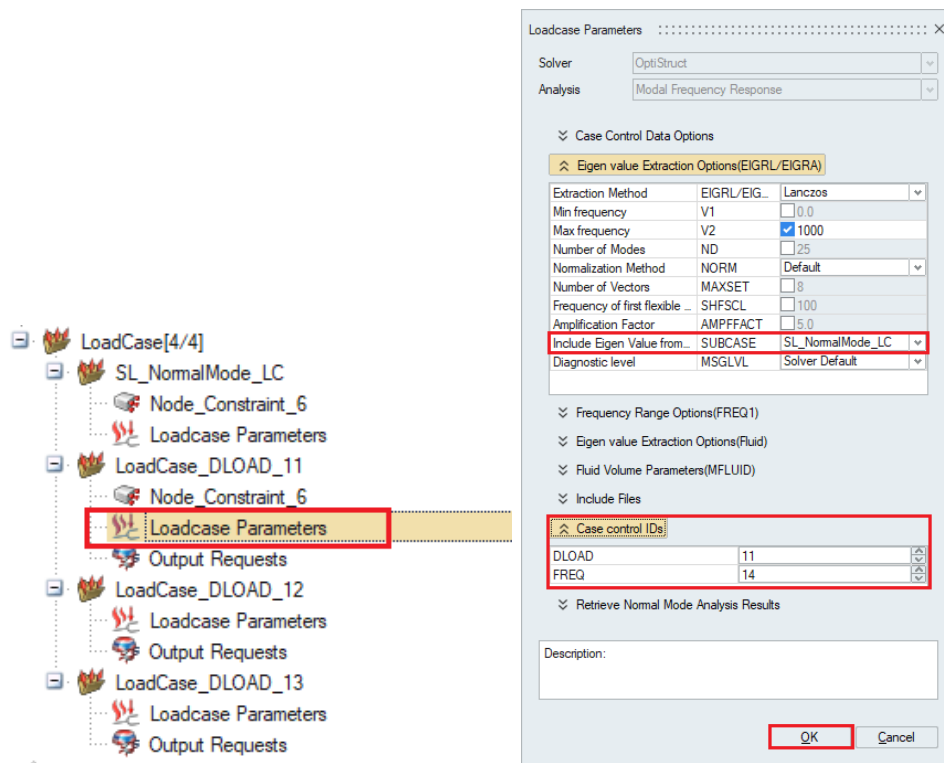
4. Next, double click on Solution Parameters in the LBC tree under *SL_NormalMode_LC* loadcase. In the Solution Parameters dialog, expand *Normal Mode Extraction Parameters*. Choose *Extraction method* as AMSES. Uncheck *Number of Modes* and enter *Max Frequency* as 2000. And click on **OK**.



5. Again, in the LBC tree, double click on the load case *LoadCase_DLOAD_11*. This will open the Load Case Setup Dialog. Choose *Constraints* and add it to the Load Case Inputs. Click **OK**.

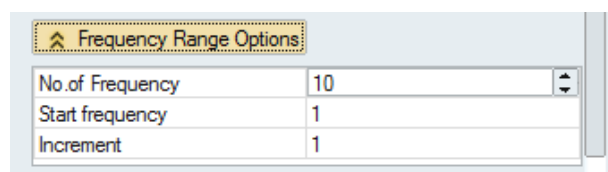


6. Next, double click on Solution Parameters in the LBC tree under *LoadCase_DLOAD_11* load case. In the Solution Parameters dialog, expand the Eigen value Extraction Options and verify that *SL_NormalMode_LC* is chosen for Include Eigen Value from Subcase. Expand Case control IDs and verify that DLOAD has ID *11* and FREQ has ID *14*. Click **OK**.

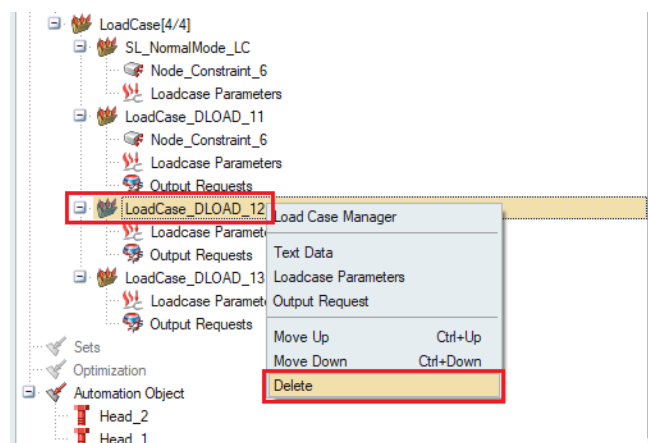


In the above case, eigen values from the SL_NormalMode_LC subcase will be used for calculating the modal frequency response. Also, this subcase points to a DLOAD and FREQ card in the Force_Flux.BULK file. This card information is automatically taken while importing the Bulk Data file.

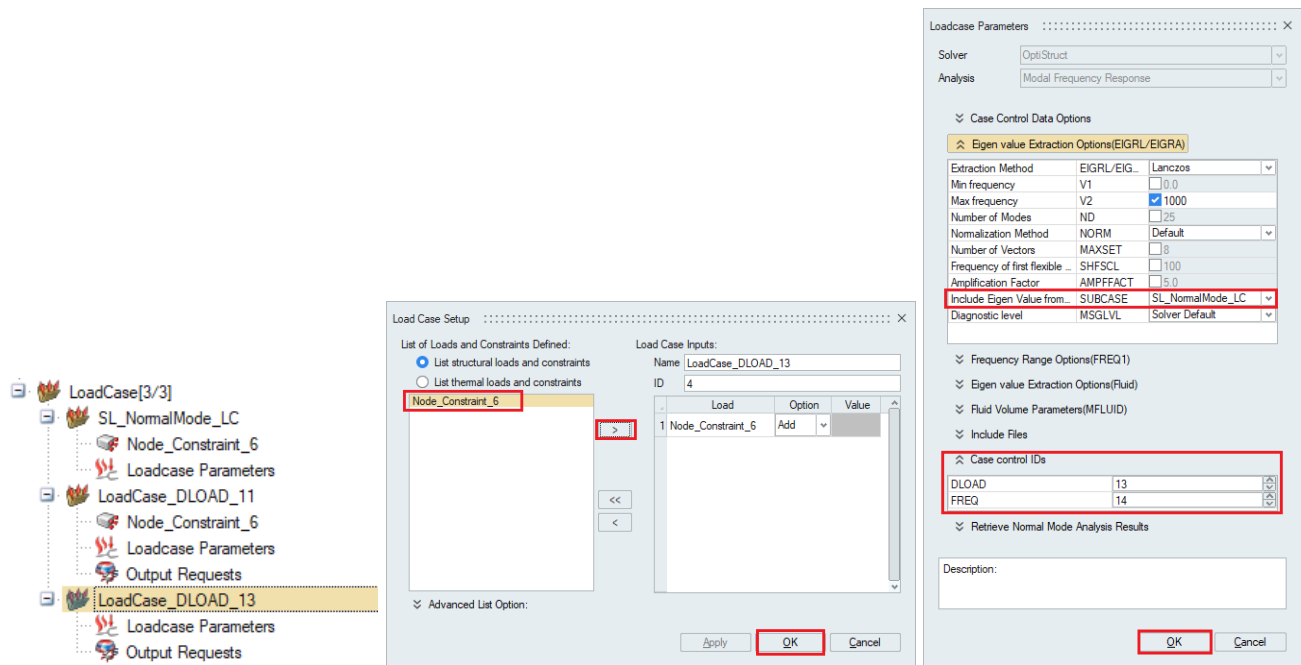
7. [OPTIONAL] You can define your own Frequency card for defining the loading frequencies by expanding the Frequency Range Options and entering the appropriate parameters. Then delete the ID associated with the FREQ card in the Case Control IDs.



8. To reduce the size of the problem, delete *LoadCase_DLOAD_12* loadcase. In the LBC tree, right click on the load case *LoadCase_DLOAD_12*, choose Delete from the sub menu.

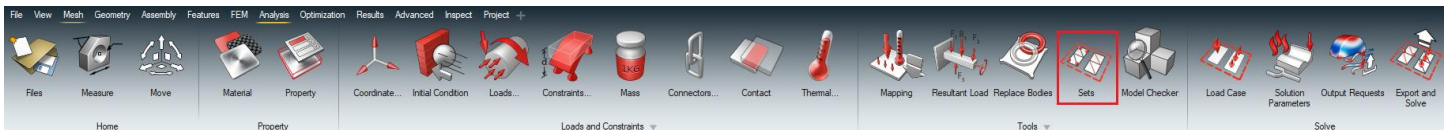


9. Repeat sub-steps 5 and 6 for *LoadCase_DLOAD_13* load case. The DLOAD ID for this load case will be 13.

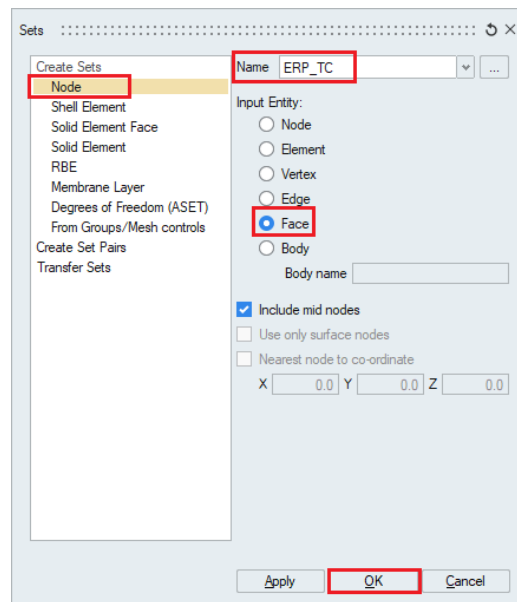
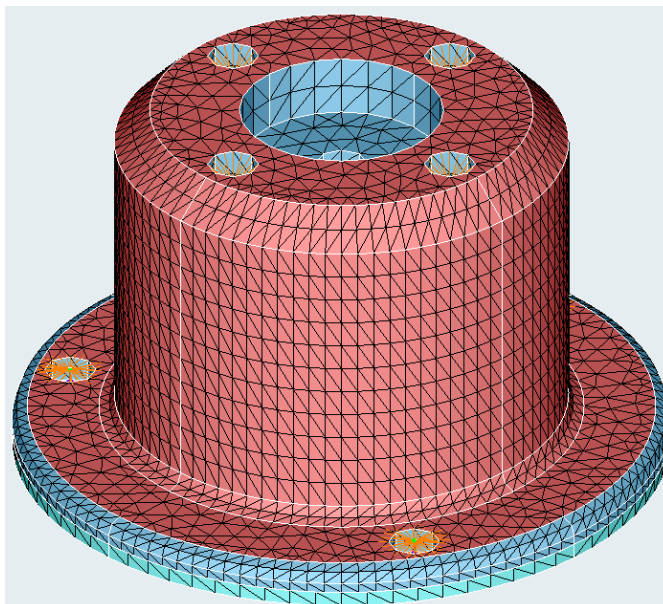


Step8: Define node set

1. In this step, a node set of the external face nodes of the CASING_TOP body will be created to calculate ERP output at these nodes. First, click on **Analysis | Tools | Sets**.

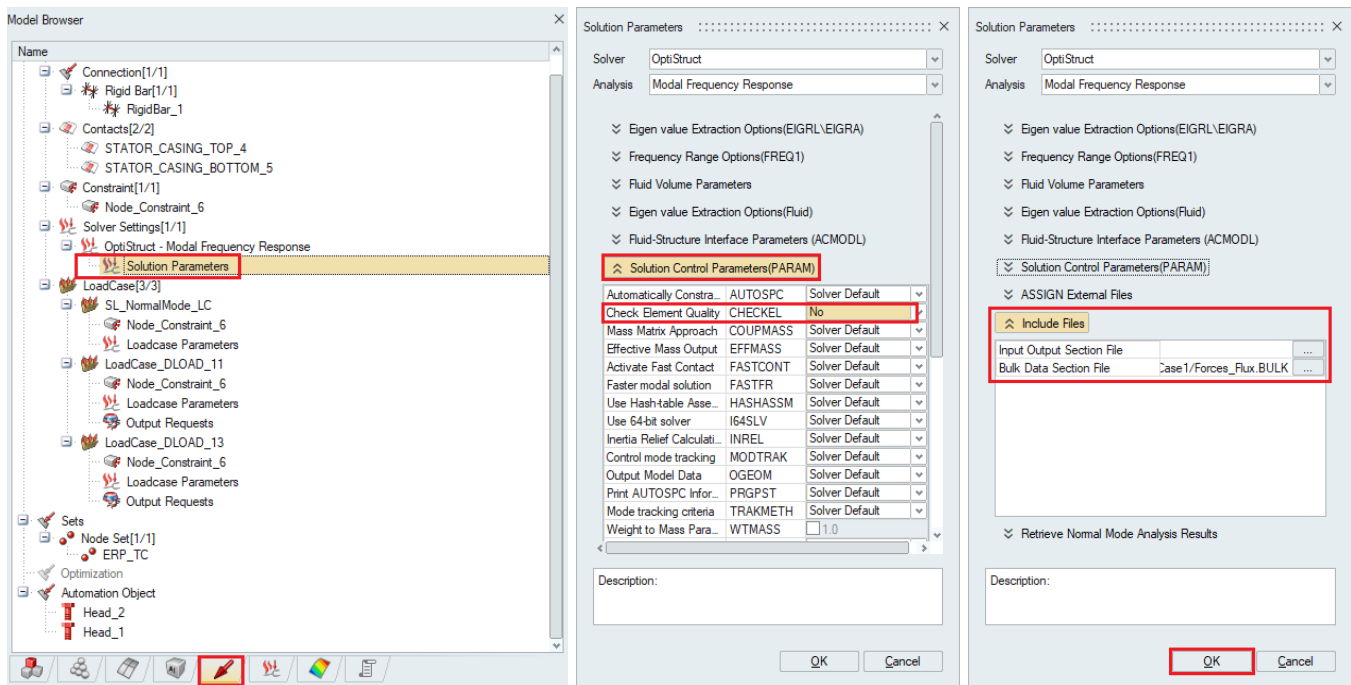


2. Choose the type as Node on the left side of the dialog. Enter name as *ERP_TC*. Choose Input entity as Face. Select the shown outer faces of the CASING_TOP body as shown in the below image. Click on **OK**.

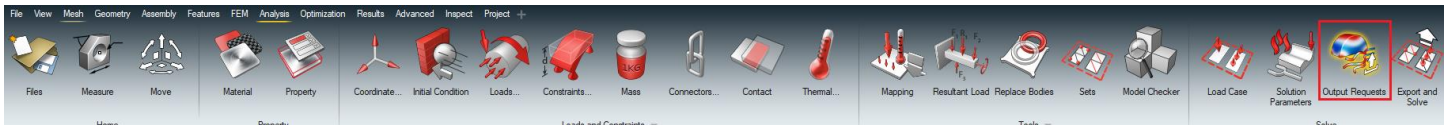


Step9: Define Solver settings

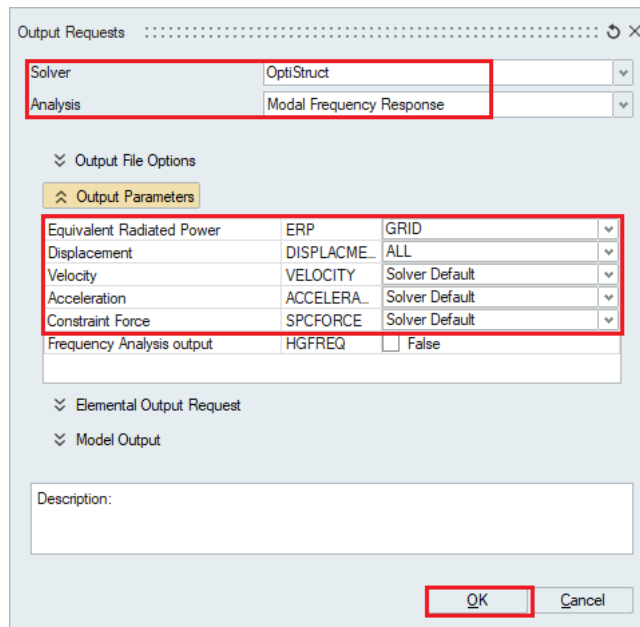
1. In the LBC tree, under Solver Settings, double click on Solution Parameters. Expand Solution Control Parameters (PARAM), choose NO for CHECKEL. Expand Include Files and verify that the Bulk Data Section File field is populated with path to the *EM_Flux_Forces.BULK* file. Click on **OK**.
In this model setup, Flux_Forces.BULK file is being used as an include file.



2. Head to **Analysis | Solve**. Click on **Output Requests**.

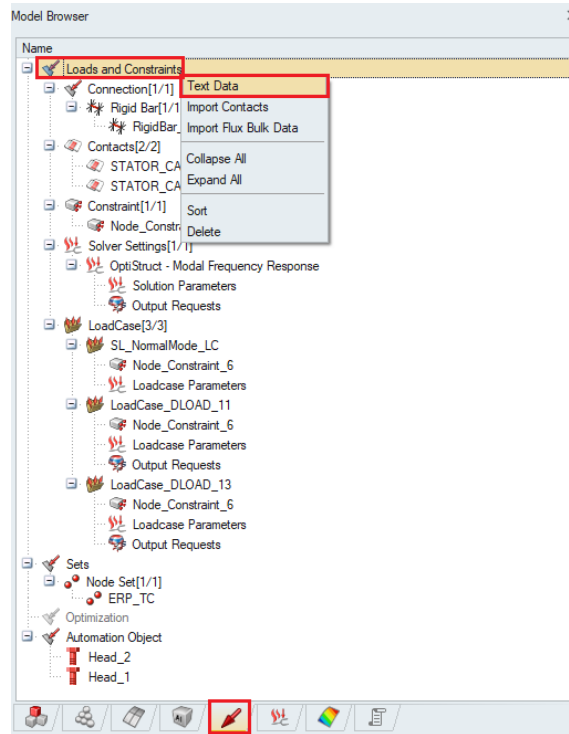


3. In the dialog, choose Solver as *OptiStruct* and Analysis as **Modal Frequency Response**. Expand *Output Parameters*. For Equivalent Radiated Power choose **GRID**. For Displacement choose **ALL**. For Velocity, Acceleration and Constraint Force choose **Solver Default**. Click on **OK**.



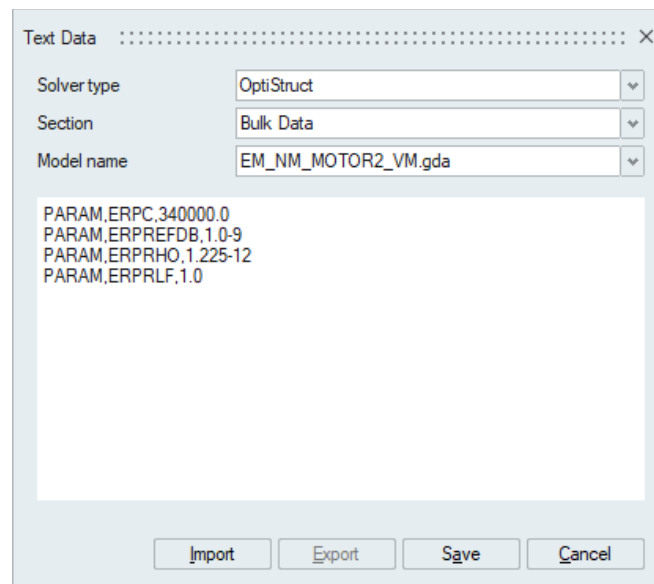
Step10: Define unsupported cards

1. ERP analysis requires some constant parameters to be defined. To define these, right click on *Loads and Constraints* in the LBC tree and click on *Text Data*.



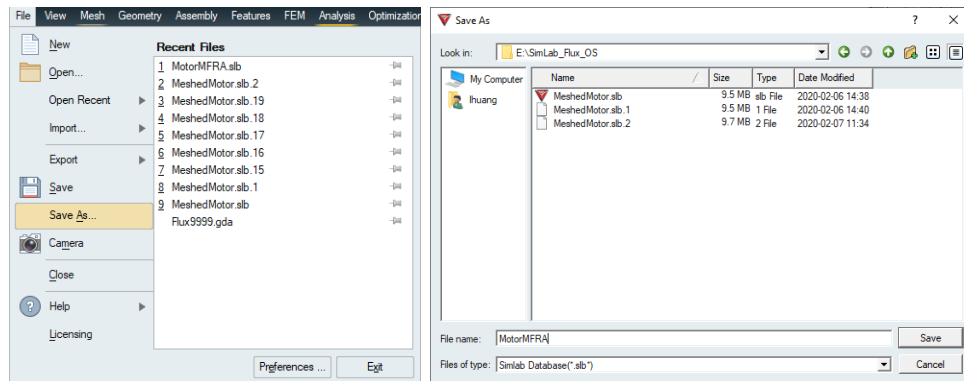
2. In the Create Text Input dialog, choose the solver type as *OptiStruct*, Section as *Bulk Data*. Leave the Model name as *EM_ NM_MOTOR2_VM.gda* (or depending on your model name). Copy and paste the below lines in the white space and click on **Save**.

```
PARAM,ERPC,340000.0
PARAM,ERPREFDB,1.0-9
PARAM,ERPRHO,1.225-12
PARAM,ERPRLF,1.0
```



Step11: Save

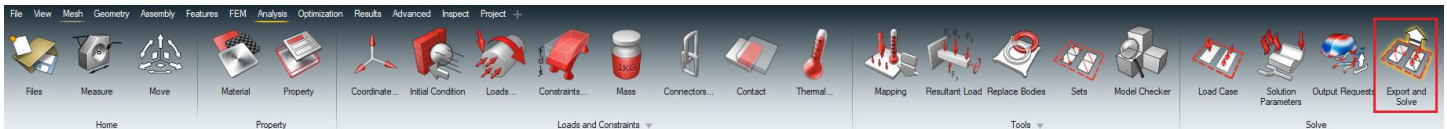
1. Open the **File** menu and click on **Save as**.
2. Enter a name *MotorMFRA.slb* and save the file in the folder of your preference.



Step12: Export and solve

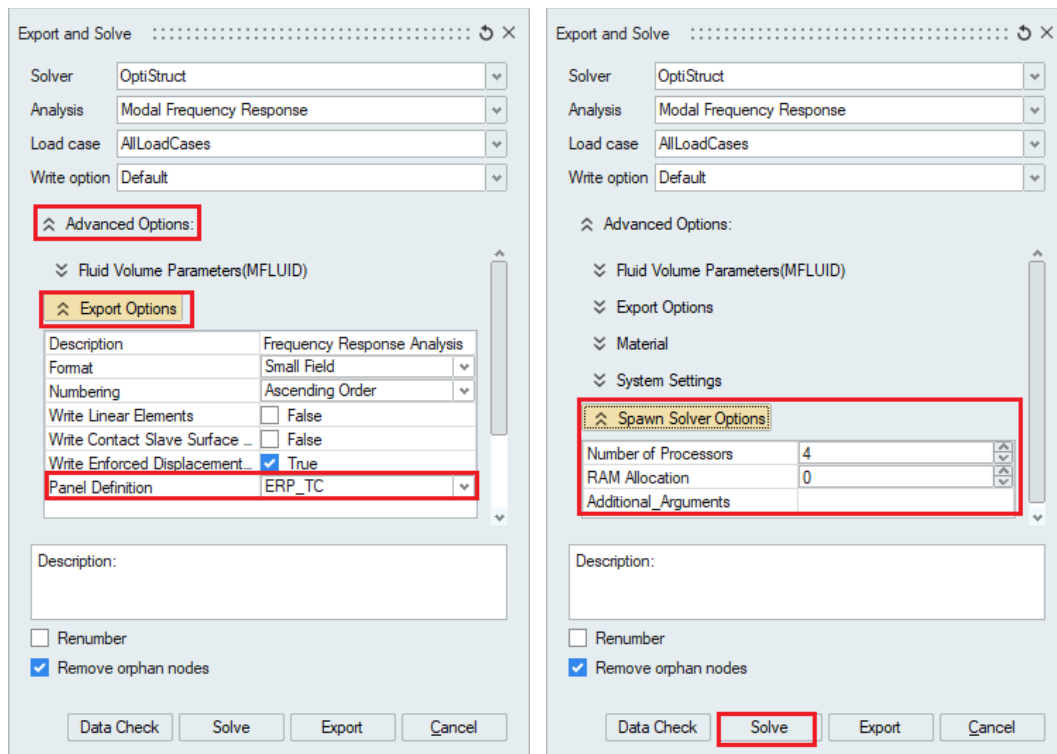
Although the setup is not totally completed, SimLab allows to complete the setup by adding the missing information, like creating the missing properties for the RBE bodies. SimLab will choose default parameters for missing options.

1. Click on **Analysis | Solve | Export** and Solve to run directly a Modal Frequency Response analysis in OptiStruct.

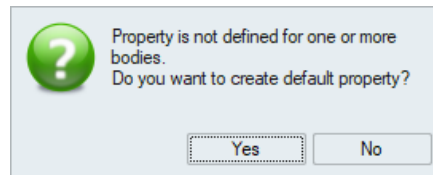


2. Then select **OptiStruct** as **Solver** and **Modal Frequency Response** as the **Analysis** type. Expand **Advanced Options** and then **Export Options**. For **Panel Definition** choose **ERP_TC**. Expand **Spawn Solver Options**. Enter the **Number of Processors** as 4 and click on Solve.

Note that: - Enter number of processor between 2 and 64 based on the number of processor you want to use for solving this problem and based on your computer capacity.



3. Confirm with Yes to the appeared dialog window.



4. Enter a name *MotorMFRA* and save the file at the location where you want to run it. The simulation is executed. **Notice that *EM_Flux_Forces.BULK* file must be copied to the folder where this file is saved.**

This simulation might take more than an hour to run. If you want to run this model externally, Export the file instead of clicking on Solve.

