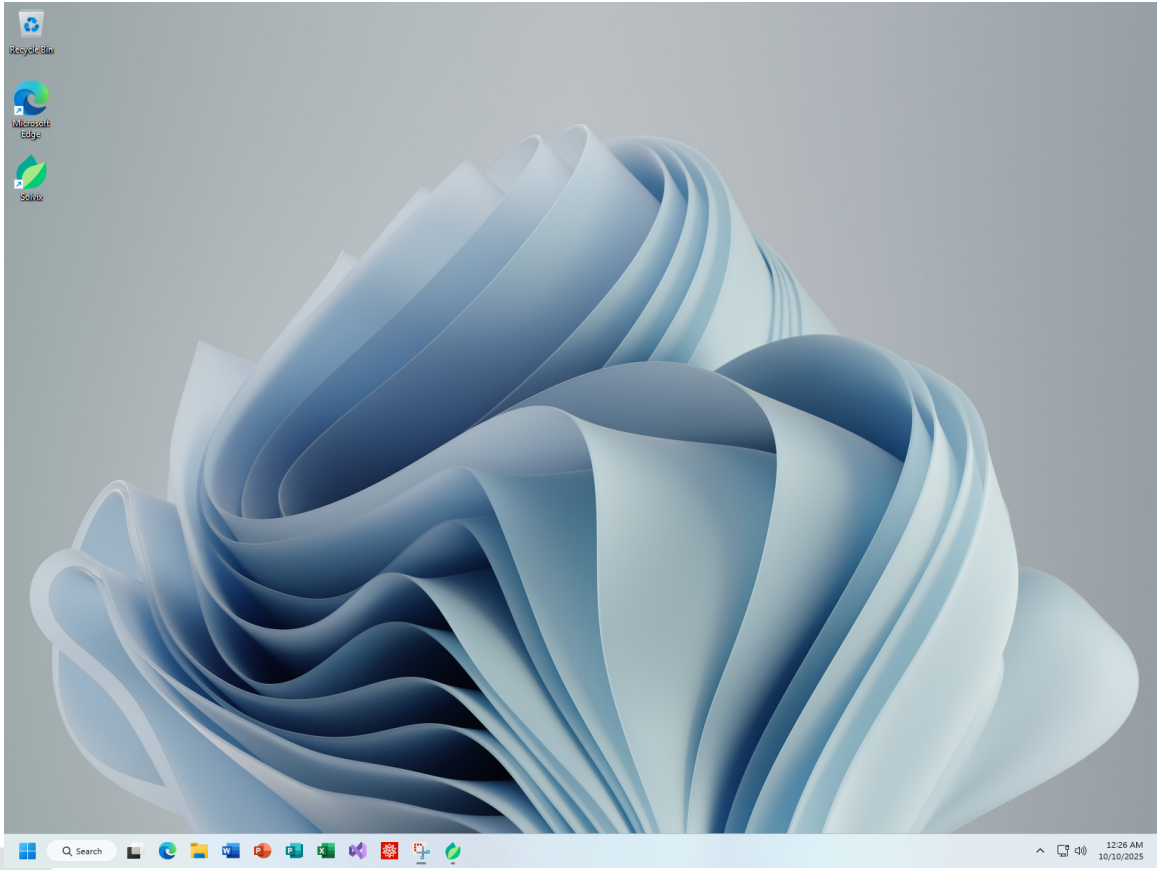
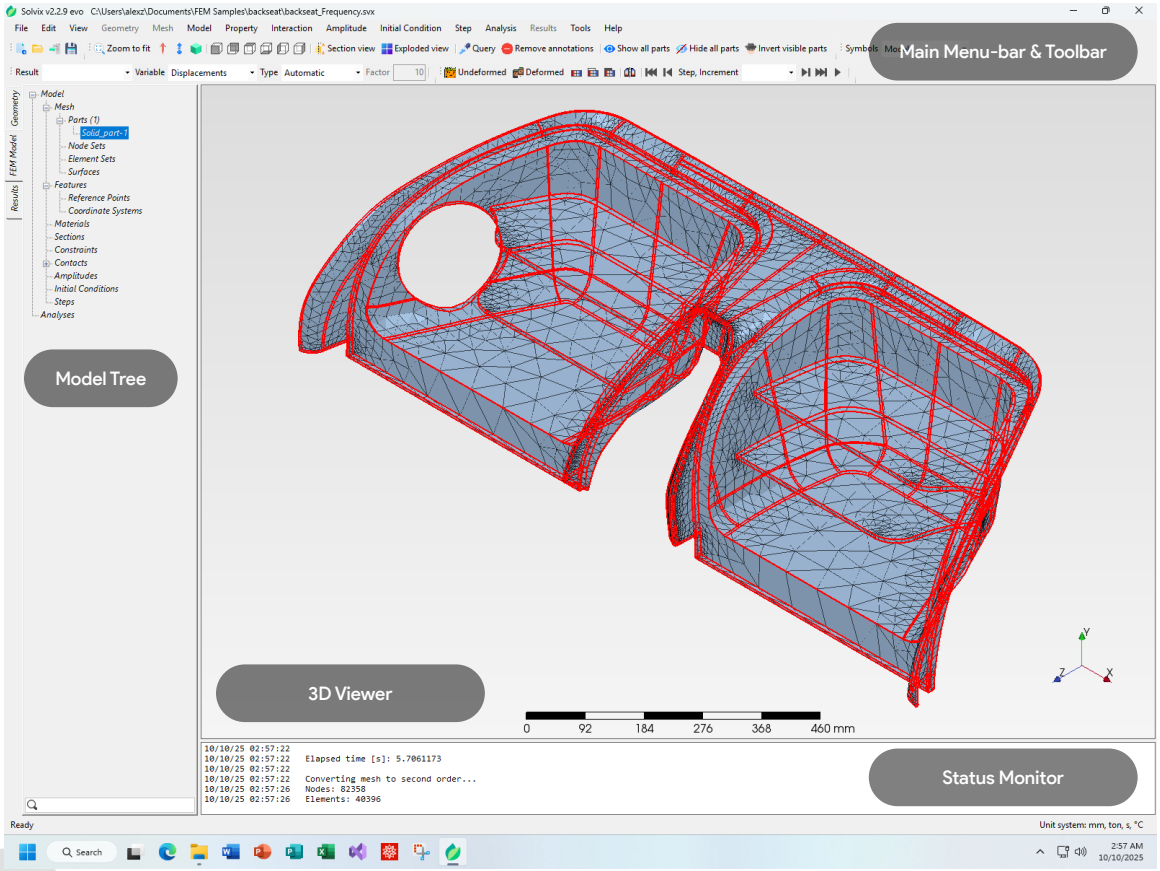
Starc Solvix Quick Start
Frequency (Modal) Analysis



Solvix is a pre- and post-processor works the Solvix FEA solver. Solvix offers a user-friendly GUI and supports many features provided by the solver.

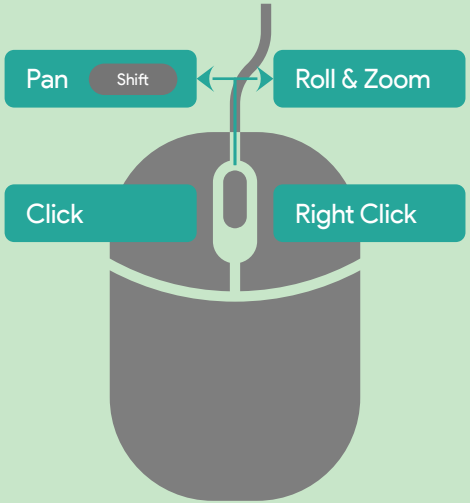
Click the icon to launch **Solvix**

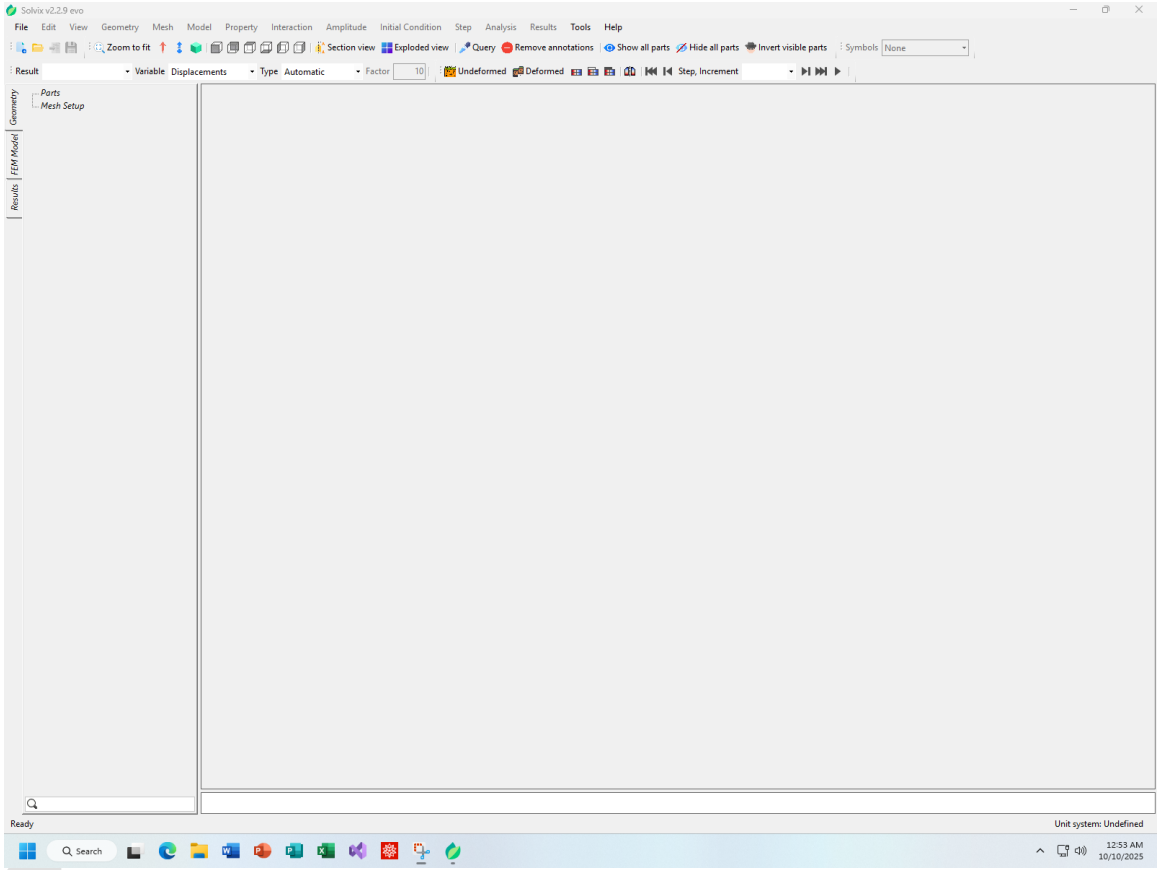
Solvix requires Microsoft .NET Framework 4.8.1 or higher to run, it is compatible with Microsoft Windows 10 or newer.



Solvix GUI is a Microsoft Windows Form application, it has four major parts in the main window including the “Main Menu-bar & Toolbar”, “Model Tree”, “3D Viewer” and the “Status Monitor”.

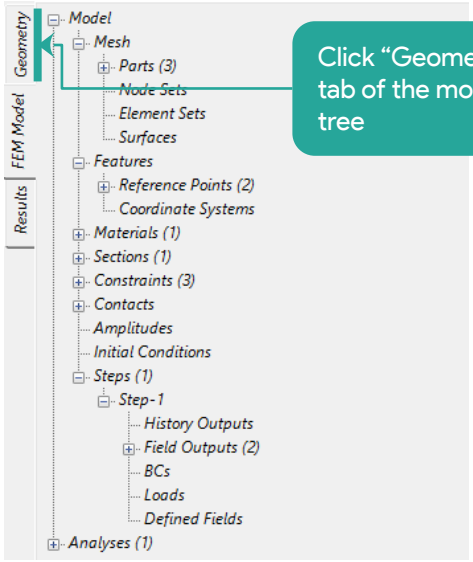
For the “Pan & Rotate” operation in the 3D viewer, please use a 3-key mouse.



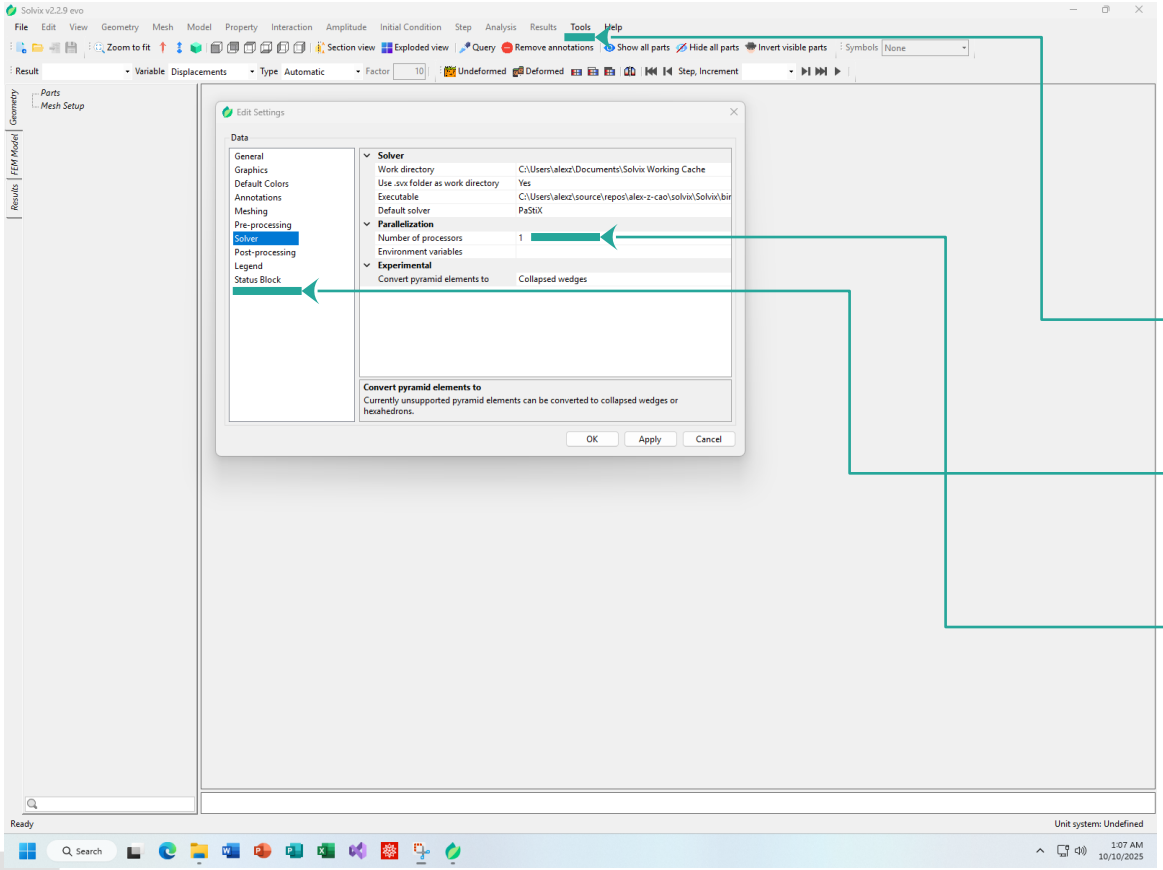


In Solvix, three tabs of the model tree dictate the workflow in this software:

- Geometry - geometry editing and mesh generation,
- FEM Model - analysis set up (pre-processing),
- Results - post-processing.



Click “Geometry” tab of the model tree



First we need to set up the work directory of Solvix, it is our first priority to put project files in order, usually we use the .svx folder as the work directory.

1 Click “Tools” in the main menu-bar, then click “Settings”, then “Solver”

2 Make changes to these options to fit our requirements

3 Remember to change the “Number of processors” if using parallel computing

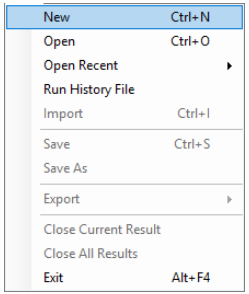
For AMD CPUs, please choose PaStiX, for Intel CPUs, please choose Pardiso, since the Pardiso solver is compiled with Intel MKL, which may not be supported by some AMD CPUs under Microsoft Windows.

Let's create our first project, the model space can be 3D, 2D Plane Stress, 2D Plane Strain or 2D Axisymmetric.

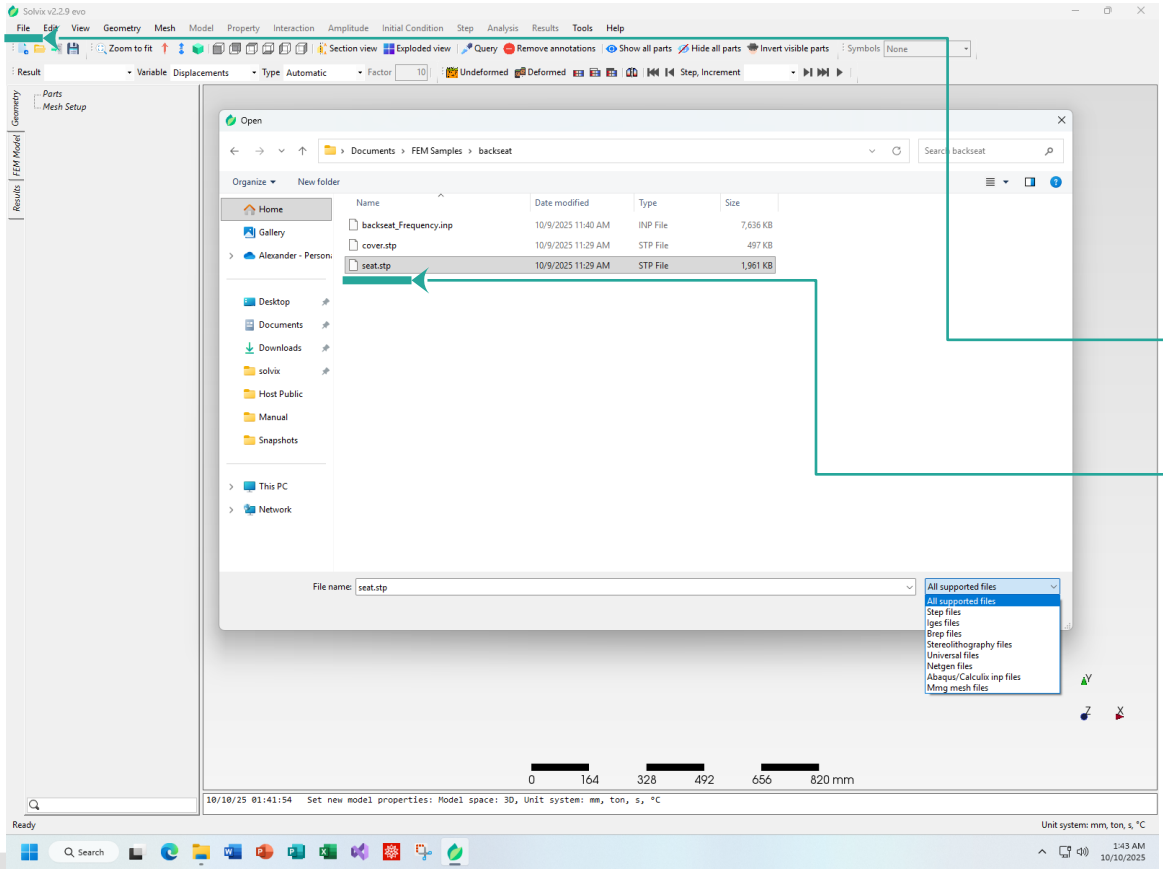
1 Click "File" in the main menu-bar, then click "New"

2 Choose the Model Space and the Unit System

3 Click "OK" to create a new model



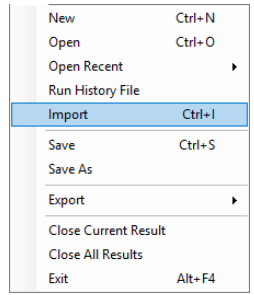
For the Model Space, please take a quick look at https://en.wikipedia.org/wiki/Plane_stress and https://en.wikipedia.org/wiki/Infinitesimal_strain_theory#Plane_strain



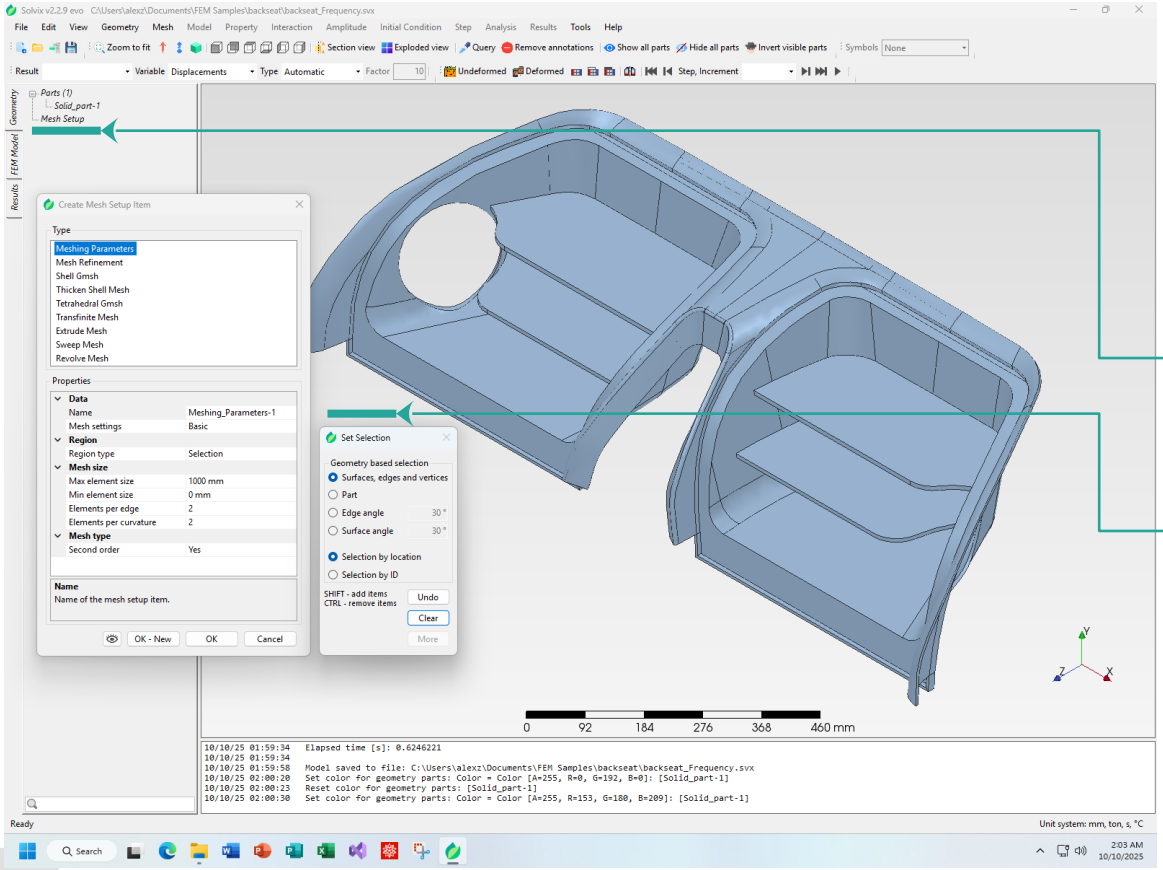
After the new empty model has been created, we need to import our geometry design model into Solvix. Solvix is compatible with .stp .iges and other neutral formats.

1 Click “File” in the main menu-bar, then click “Import”

2 Choose a geometry design file to import

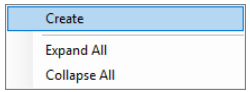


For 2D analyses, parts must be imported as faces lying entirely in the XY plane. Moreover, for axisymmetric analyses, faces representing parts must lie to the right of the Y axis.



After importing the geometry model, we need to create the FEM mesh. To control the quality of the mesh, we need to set the mesh parameters for Solvix to generate the mesh.

1 Right click “Mesh Setup” in the “Geometry” model tree, then “Create”

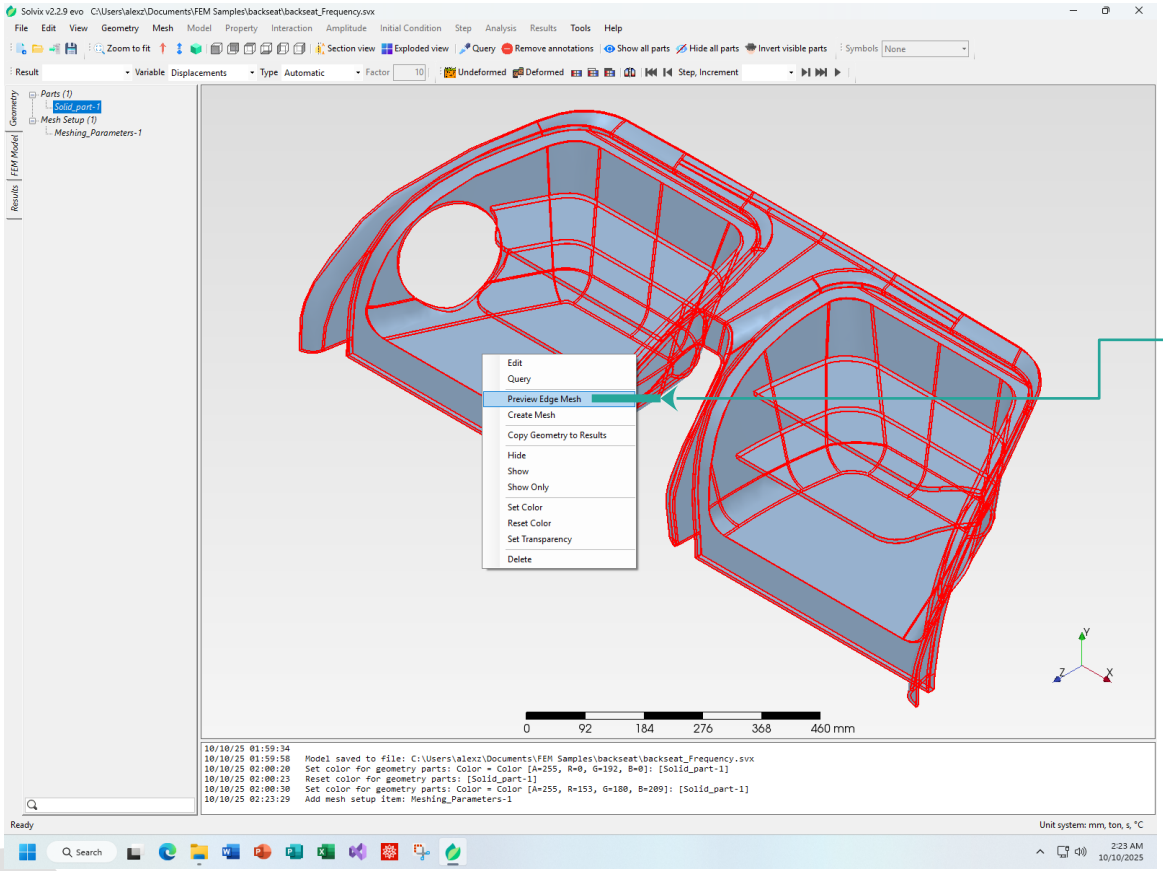


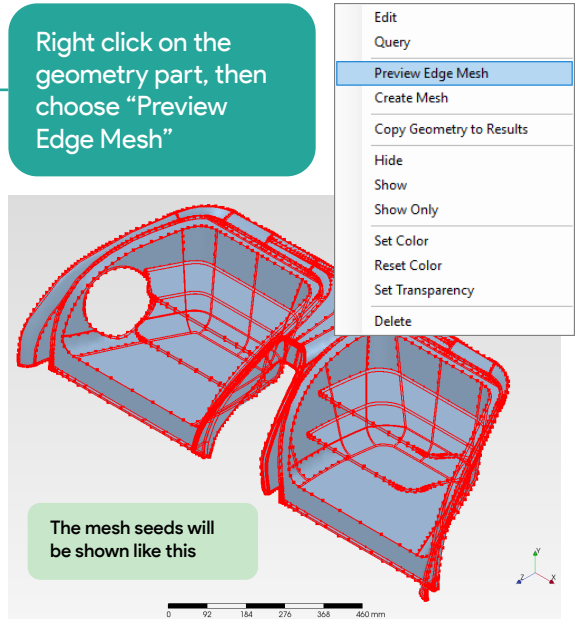
2 Define the mesh setup parameters and use the “Set Selection” tool to choose the section for the meshing setup

For the types and quality of mesh , please take a quick look at https://en.wikipedia.org/wiki/Types_of_mesh

Remember to preview the seeds of the mesh by right click on the geometry model before we do the mesh, this step sometimes can save a lot of time.

Right click on the geometry part, then choose “Preview Edge Mesh”

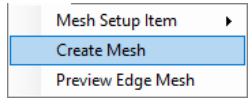




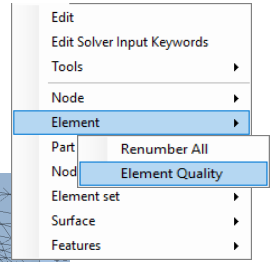
The mesh seeds will be shown like this

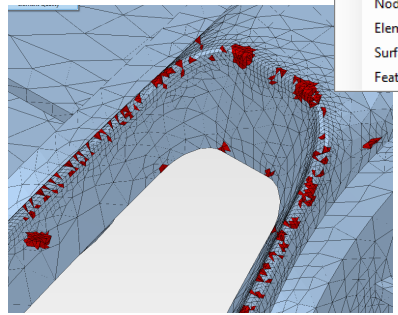
Then we can mesh the geometry model. Different geometry parts can use different mesh setups, defined by different mesh setup sections.

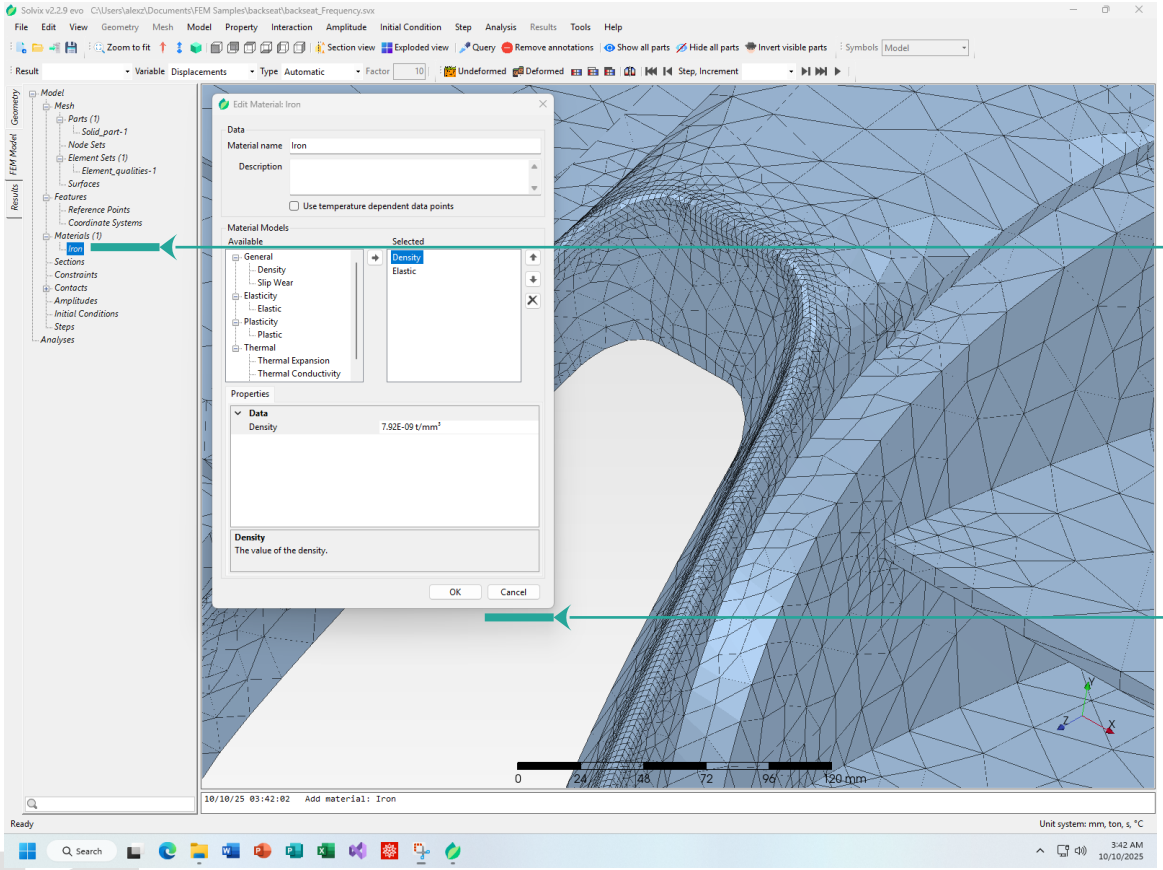
1 Click “Mesh” in the main menu-bar, then click “Create Mesh”



2 Click “Model” in the main menu-bar then “Element” then “Element Quality” to check the quality of the mesh

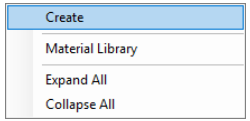






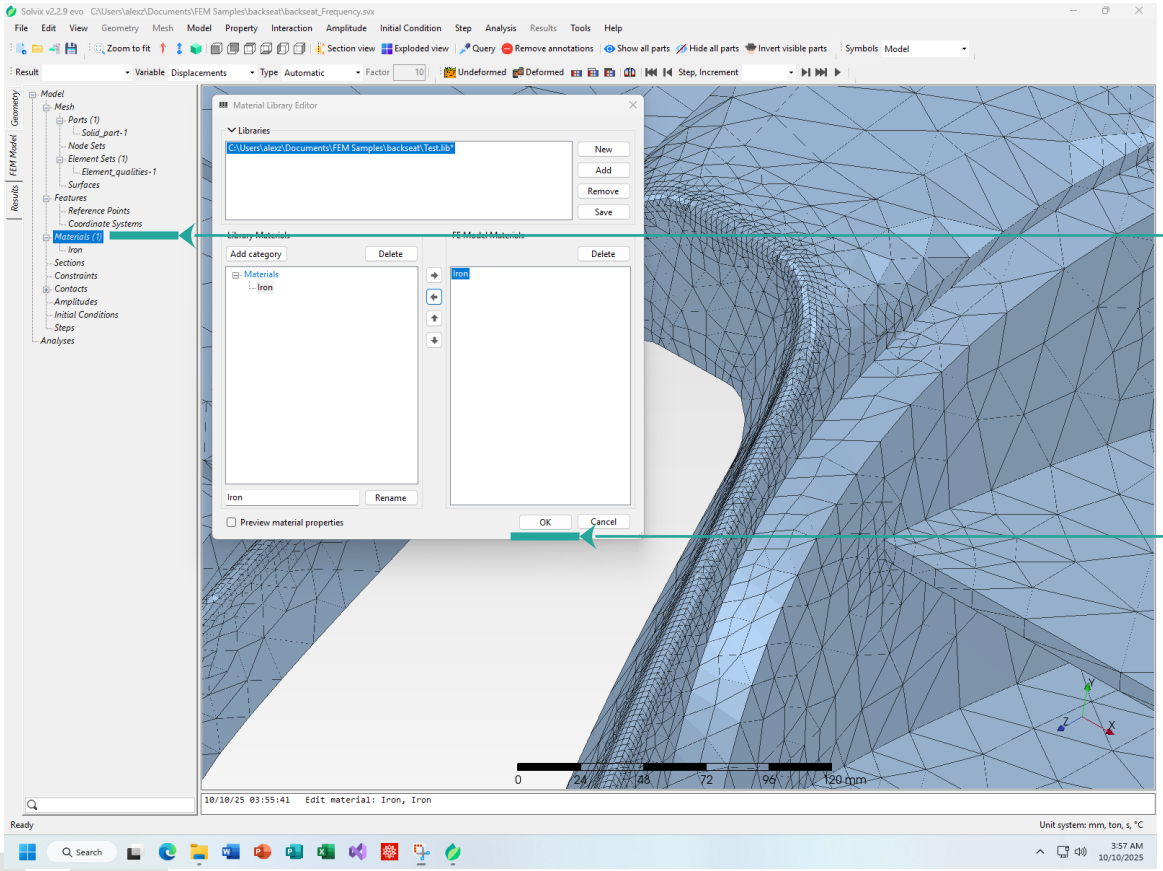
We need to define materials for the FEM analysis, different analysis types require different material parameters.

1 Right click “Materials” under “FEM Model” tab of the model tree, then click “Create”



2 Create a new material named “Iron” with necessary parameters then click “OK”

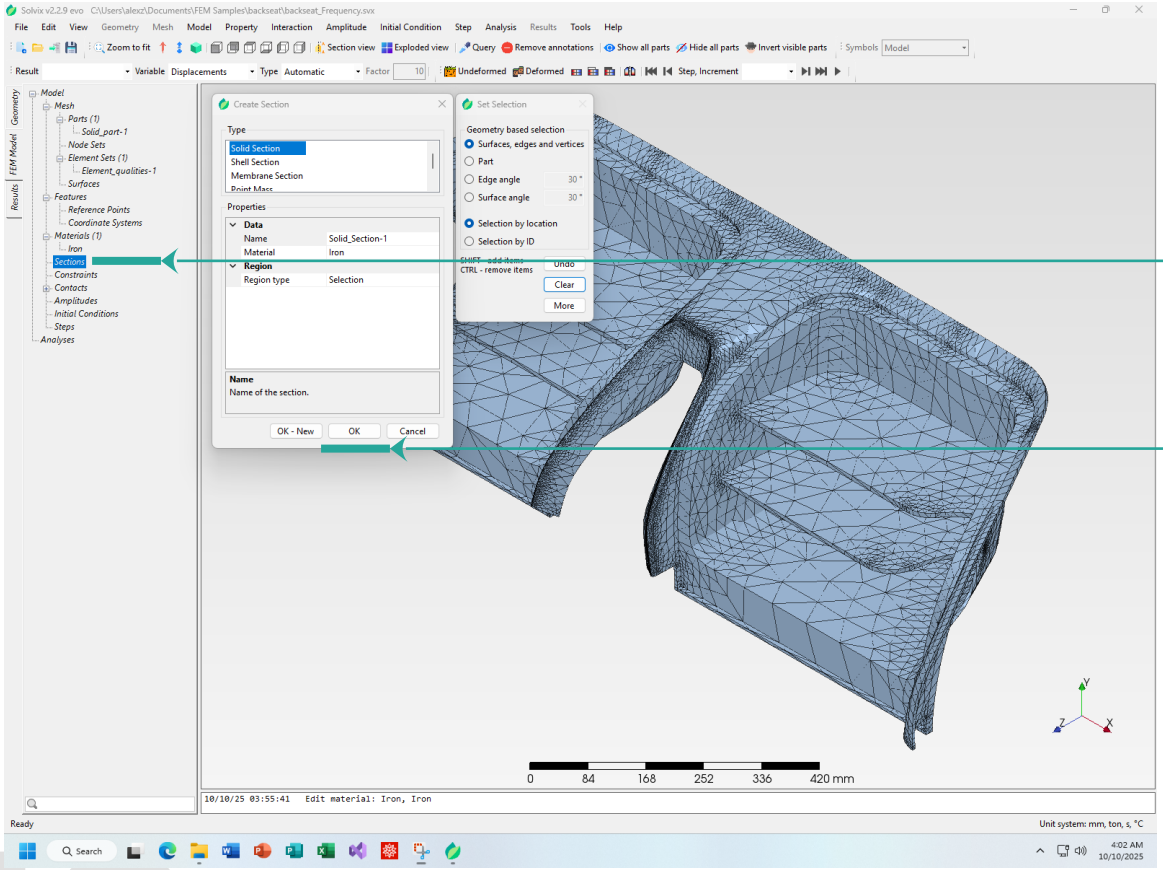
The Materials module can automatically manage the unit of the material parameters, for example, the density of the iron is around 7,920kg/m^3, we can directly input “7920kg/m^3” into the field, it can be converted into 7.92E-9 t/mm^3 after we hit Enter. For the “Elastic” parameters, we use “2.11E11 Pa” for the Young’s Modulus and “0.33” for the Poisson’s Ratio.



To define the material, building a customized material library is a forethoughtful way for further simulations

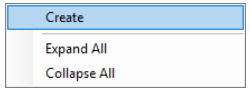
1 Right click “Materials” under “FEM Model” tab of the model tree, then click “Material Library”

2 Build the material library as your wish and save it to the safe place, click “OK” when finish

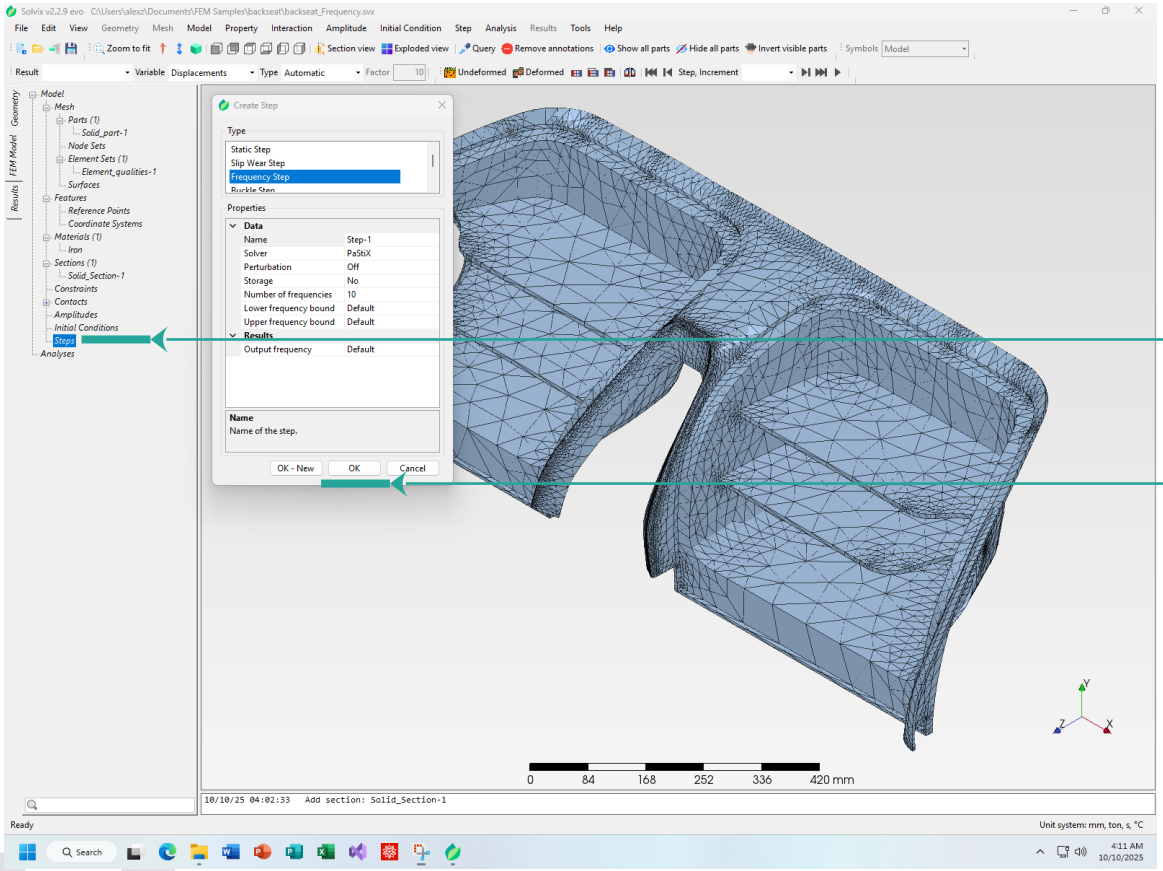


Now we apply the material data to the meshed part, different materials can by applied to different parts by using multiple sections.

1 Right click “Sections” under “FEM Model” tab of the model tree, then click “Create”

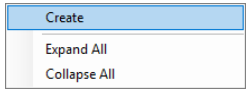


2 Define the section parameters by choosing different options and connect the parameters to the mesh by using the “Set Selection” tool, click “OK” when finish



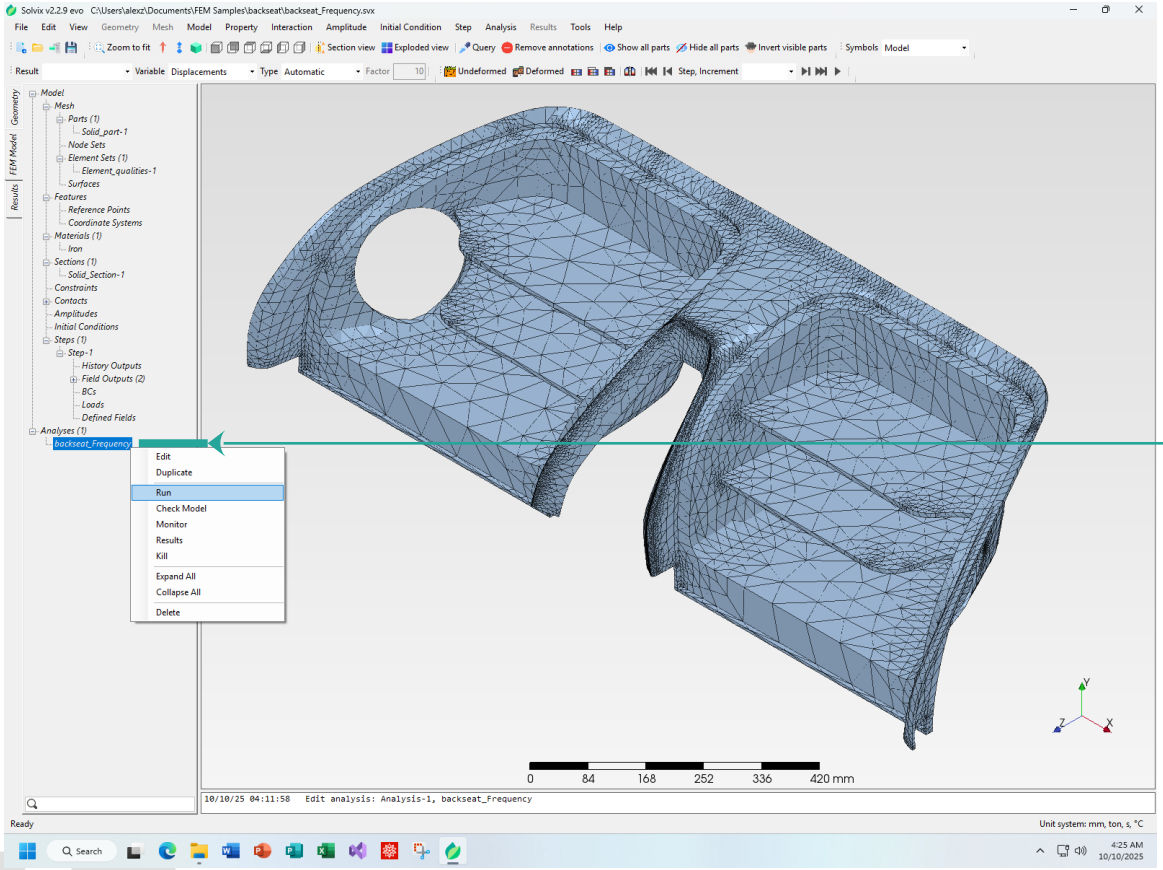
In Frequency analysis (Modal Analysis), we do not have to define the “Constraints”, “Boundary Conditions (BCs)” and “Loads”, we now jump to the “Steps” definition part

1 Right click “Steps” under “FEM Model” tab of the model tree, then click “Create”



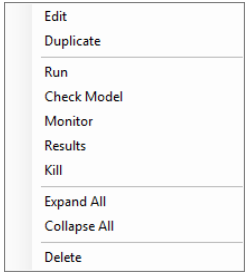
2 Choose “Frequency Step”, the define the parameters by choosing different options, click “OK” when finish

Since there’s no constraints in this analysis model, the first six eigenvalues of the frequency should be close to zero, they indicate six rigid-body modes of motion.

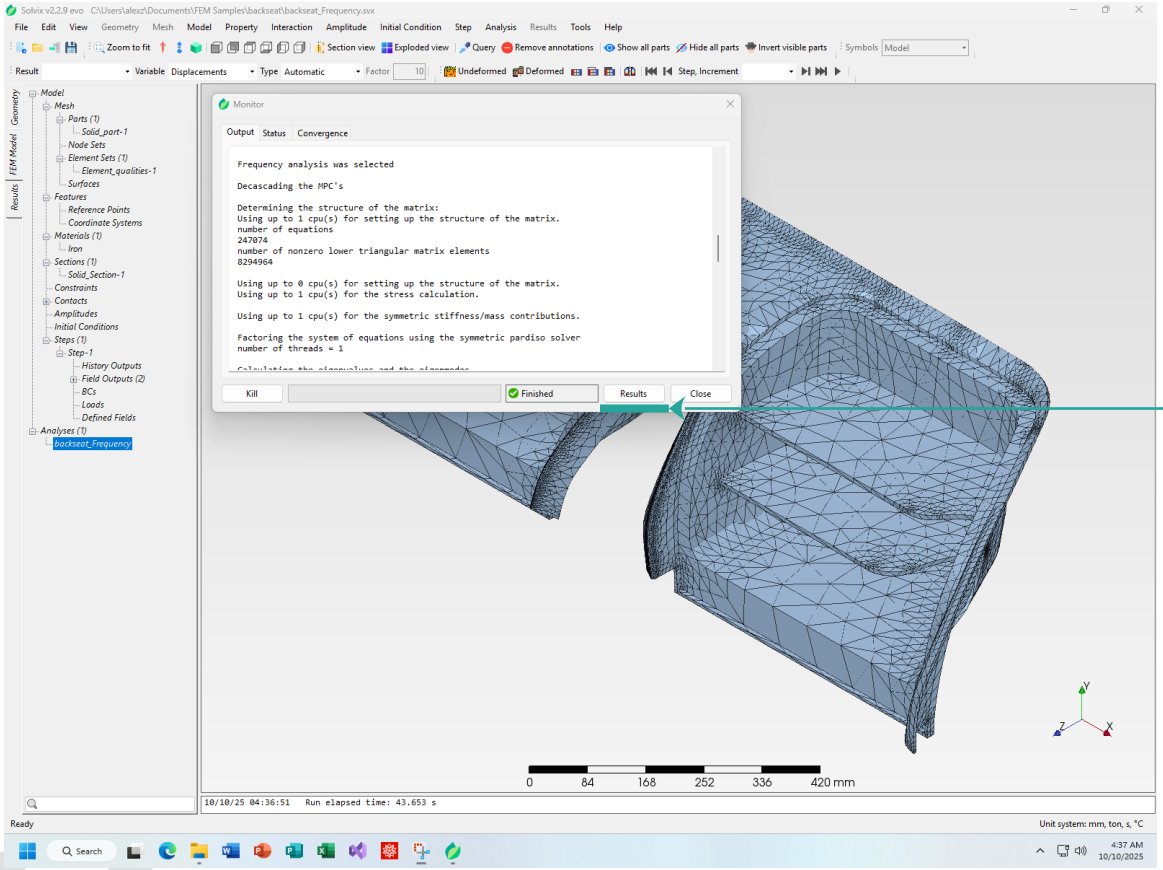


A new analysis task will be generated after we define the new step, we need to submit the task to run or we can use “Check Model” to generate the analysis deck file for further use.

Right click the analysis task, then click “Run”

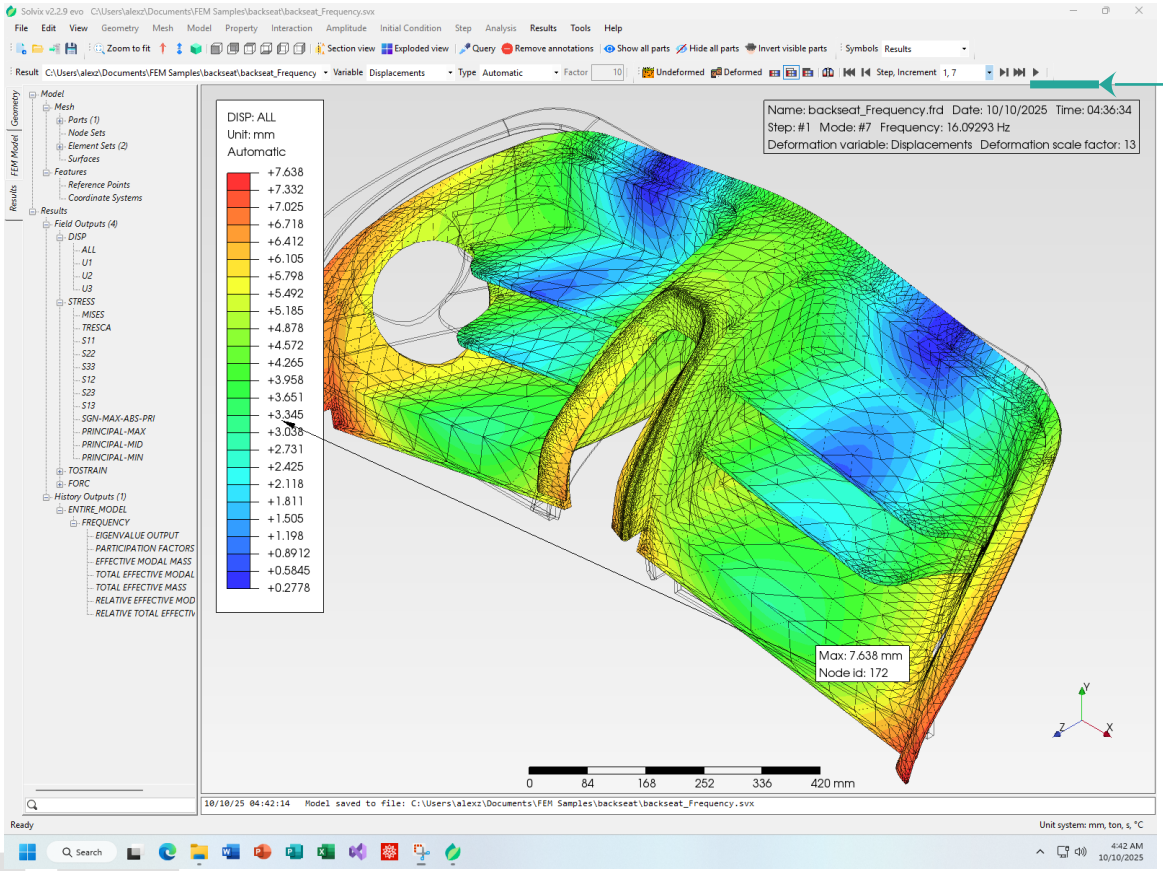


The analysis deck file will be generated into the work directory, we can copy the analysis deck file, which ends with the extension name of .inp to the high performance computer to run.



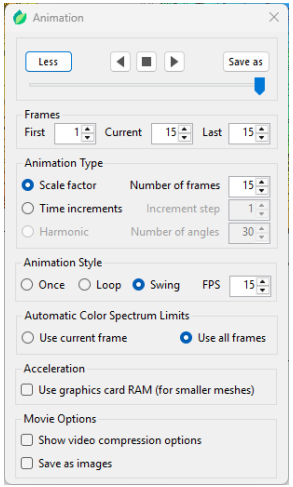
After we click “Run”, the analysis task will be submitted to the solver, the status can be seen through the Monitor window, when it’s done, click “Results” to read the result of the analysis.

Click “Results”



Finally we get the result from Solvix, use the result toolbar of the "Results" menu to enjoy it.

Click the Play symbol to create animation of the deformation



Starc Solvix Quick Start
Frequency (Modal) Analysis

Q&A