# Introduction

This advisor helps you to build a simple finite element (FE) model. It limits the FE model to a single component and a linear elastic material model.

No constraints or contacts are currently supported by the advisor.

Three different types of FE analysis can be performed using the advisor: static linear analysis, frequency analysis and buckling analysis.

To create a new FE model select the option 1 and choose the desired system of units.

After the unit system is created, select Next to continue.

# Import Geometry

The geometry can be imported into PrePoMax from other CAD programs using .stp/.step or .stl file formats.

To import the geometry, select the option 1 and browse for the geometry file to import. Wait until the geometry is imported and then check the model tree on the left that only one part was imported. To rename the part select the option 2 and change the Name field.

Select Next to continue.

# Prepare the FE Mesh

The geometry must be discretized into finite elements, to perform any type of analysis in the PrePoMax. Smaller finite elements result in better accuracy but require longer computational time.

Select the option 1 to use the default finite element size and create the FE mesh. To change the size of the largest allowable finite element in the mesh, choose the option 2 and change the Max element size field.

After the mesh is generated the size of the finite elements can be adjusted by repeating the step 2.

Select Next to continue.

# Apply Material Properties

Material properties are defined using a material model. A new material model can be created using the option 1. Material models can also be stored or retrieved from the material library using the option 2.

After the definition of the material model its properties must be assigned to the FE mesh. Use the option 3 to assign the created material model to the FE mesh. The FE mesh must be selected using the mouse.

Select Next to continue.

# Select the Analysis Type

Different analysis types can be performed using this advisor. For most cases, the default analysis settings can be used.

If displacements, stresses and strains are to be computed, create a static linear analysis using the option 1 (Static step).

To find the lowest eigenfrequencies of the FE model, select the option 2 for the frequency analysis (Frequency step).

When the FE model is loaded in compression, buckling might occur. To determine the buckling factor choose the option 3 (Buckle step).

# Create Boundary conditions

For a static and buckling analysis, the movement of the FE model must be prevented using a boundary condition. For a frequency analysis, the boundary condition is optional.

To add a boundary condition that prevents the movement in all directions, choose the option 1 and then select the region to constrain.

To individually specify the directions in which the movement is prescribed, select the option 2 and then choose the region to constrain.

# Define the Loads

For a static and buckling analysis the loads must be defined. For a frequency analysis, no loads are allowed.

To add a load defined by a force magnitude and direction, select the option 1 (Surface traction) and then choose the region to load.

If the FE model is loaded by a pressure, select the option 2 (Pressure) and then choose the region to load.

When the FE model is large and bulky, the gravity must be taken into account. To assign a gravity load, select the option 3 (Gravity load) and then choose the region to load.

# Run the Analysis

To run the analysis, select the option 1. A Monitor window will open where you can follow the analysis process. After the analysis completes, close the Monitor window.

After the analysis completes, the results must be loaded by using the option 2.

# Results

Depending on the analysis type, different results are of interest.

For static linear analysis, displacements (DISP) and stresses (STRESS) are usually inspected. Their components can be displayed by selecting them in the results tree.

Eigenfrequency and eigenshape are important for frequency analysis. The eigenshape is displayed as the deformation of the FE Model while the eigenfrequency is reported in the status block. By default, 10 eigenfrequencies are computed. The displayed eigenfrequency can be changed by using the Step, Increment drop-down menu in the results toolbar.

The buckling analysis reports the buckling factor in the status bar. Using the buckling factor, the limit load can be computed by multiplying it with the load defined on the FE Model. The limit load represents the load at which the FE Model buckles.

At the end of the analysis, you can save the model for later by using the option 1.