

PrePoMax 1.4.1 Manual

Jakub Michalski, Ihor Rokach and Matej Borovinšek

October 3, 2023

Contents

1	What is PrePoMax?	3
2	Currently available analysis features	3
3	Workflow, general options	5
3.1	File Menu	7
3.2	Edit Menu	9
3.3	View Menu	9
3.4	Tools Menu	11
3.5	Help Menu	13
4	Geometry and mesh options	14
4.1	Geometry Menu	14
4.2	Mesh Menu	15
5	FE Model options	16
5.1	Model Menu	16
5.2	Property Menu	19
5.2.1	Libraries	20
5.2.2	Library materials section	20
5.2.3	FE model materials section	20
5.2.4	Using the material library	20
5.2.5	Exchanging material data between users	21
5.3	Interaction Menu	24
5.4	Amplitude menu	27
5.5	Initial Condition Menu	28
5.6	Step Menu	29
5.6.1	Step	30
5.6.2	History Output	35
5.6.3	Field Output	37
5.6.4	Boundary condition	38
5.6.5	Load	40
5.6.6	Defined Field	45
5.7	Analysis Menu	46
6	Results options	46
6.1	Results Menu	47

1 What is PrePoMax?

PrePoMax is an open-source pre- and postprocessor for also open-source Finite Element Analysis solver **CalculiX CrunchiX**. PrePoMax is being developed by Dr. Matej Borovinšek from University of Maribor, Slovenia. The current stable version is 1.4.1. Source code of the development version of the program is available at [GitLab](https://github.com/PrePoMax). PrePoMax offers a very user-friendly GUI and supports many features provided by the CalculiX solver. It can be downloaded from the following website: <https://prepomax.fs.um.si/>. User forum is available here: <https://prepomax.discourse.group/>.

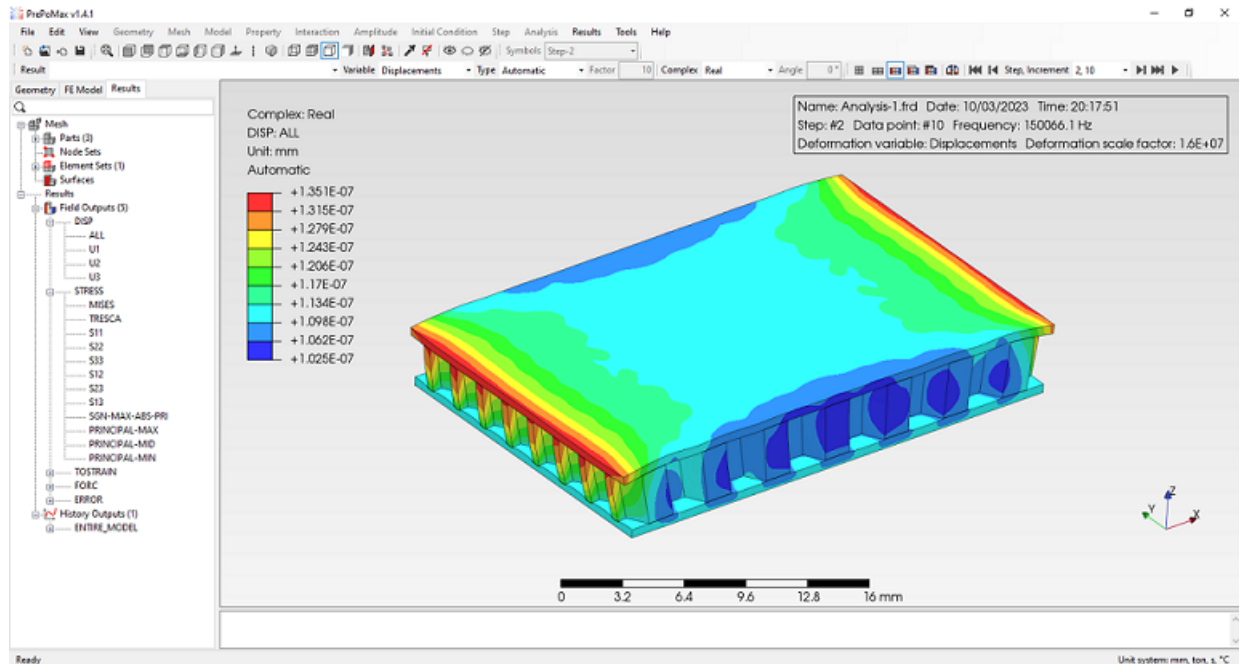


Figure 1: PrePoMax user interface

2 Currently available analysis features

- Analysis step types:
 - Static
 - Natural frequency extraction
 - Steady state dynamics
 - Modal dynamics
 - Dynamic (implicit and explicit)
 - Linear buckling
 - Heat transfer
 - Uncoupled temperature-displacement
 - Coupled temperature-displacement
 - Slip wear (based on the Archard model)
- Finite element types:
 - Solid tetrahedrons (first and second order)

- Shell triangles or quadrilaterals (first and second order)
- Membrane triangles or quadrilaterals (first and second order)
- Plane stress triangles or quadrilaterals (first and second order)
- Plane strain triangles or quadrilaterals (first and second order)
- Axisymmetric triangles or quadrilaterals (first and second order)
- Import of solid wedges (first and second order)
- Import of solid hexahedrons (first and second order)
- Material models (isotropic, with optional temperature dependency):
 - Linear elasticity
 - Plasticity
 - Thermal
 - Slip wear
- Constraints:
 - Point spring
 - Surface spring
 - Rigid body
 - Tie
- Interactions:
 - Contact
 - * Friction
 - * Gap conductance
- Initial conditions and defined fields
 - Temperature (initial condition and defined field)
 - Velocity (initial condition)
- Boundary conditions:
 - Fixed
 - Displacement/Rotation
 - Submodel
 - Temperature
- Loads:
 - Concentrated force
 - Moment
 - Uniform pressure
 - Hydrostatic pressure
 - Imported pressure
 - Surface traction

- Normal shell edge load
- Gravity
- Centrifugal load
- Pre-tension
- Concentrated flux
- Surface flux
- Body flux
- Convective film
- Radiation

3 Workflow, general options

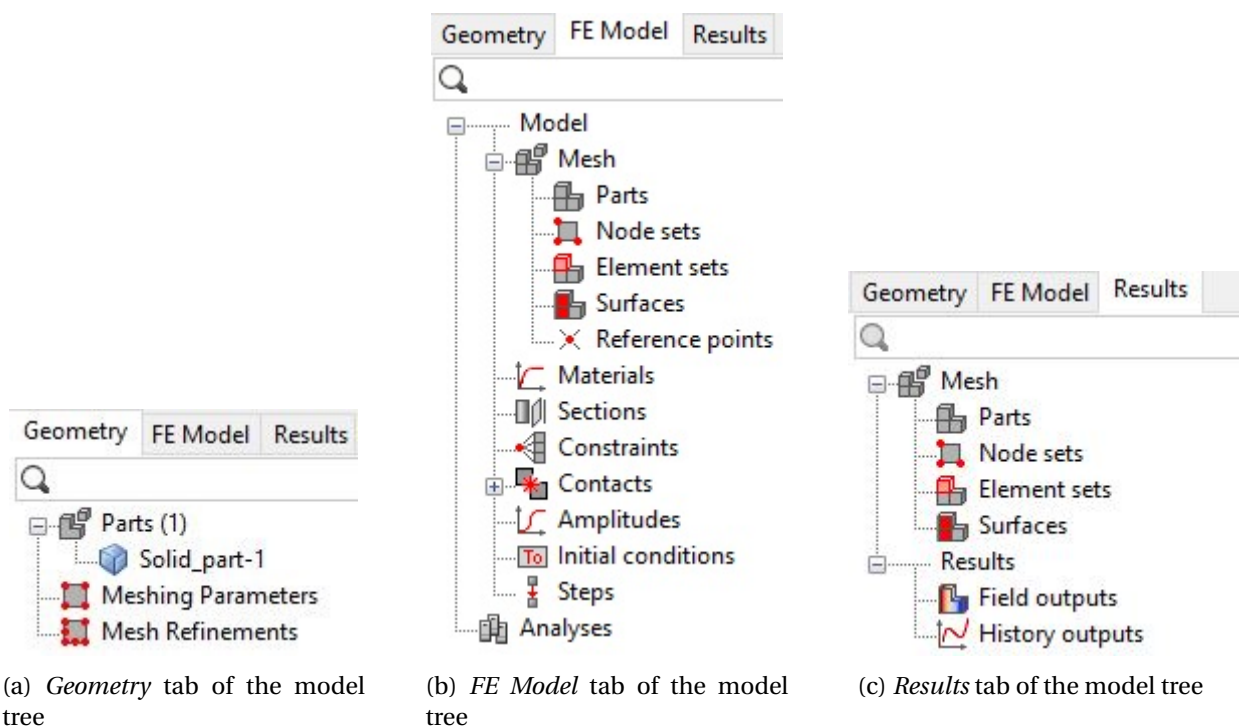


Figure 2: Three PrePoMax tabs

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

1. *Geometry* – geometry editing and mesh generation (Figure 2a).
2. *FE Model* – analysis set up (pre-processing), see Figure 2b.
3. *Results* – post-processing (Figure 2c).

Each tab also activates proper menus in the top toolbar.

The general steps that should be followed when working in PrePoMax are listed below:

1. Create a new model (**File** > **New**) and select a model space and unit system.
2. Import CAD/STL geometry or mesh for the analysis (**File** > **Import**). For 2D analyses, parts have to 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.

3. For CAD and STL geometry set mesh parameters (**Mesh** >> **Meshing Parameters** >> **Create**) and generate a mesh (**Mesh** >> **Create Mesh**).
4. When the mesh is successfully created, proceed to the *FE Model* part of the model tree.
5. Define material for the analysis (**Materials** >> **Create**), add proper material behaviors: Elasticity, Density (for performing a modal analysis or using gravity or centrifugal loads), Plasticity (to account for permanent deformations), Thermal or Slip wear.
6. Create a new section with the material defined before (**Section** >> **Create**). Section type can be Solid (for using 3D or 2D solid elements), Shell (for using shell elements) or Membrane (for using membrane elements). The thickness must also be specified if shell, membrane or plane stress/strain elements are used. Select the part to assign the section to it.
7. Create a new analysis step. Default step settings are sufficient in most cases.
8. Optional: create constraints (**Constraints** >> **Create**), define contact (expand **Contact** container, select **Surface Interactions** >> **Create** and then select **Contact pairs** >> **Create**), initial conditions (**Initial condition** >> **Create**), defined fields (**Defined fields** >> **Create**) and/or amplitudes (**Amplitude** >> **Create**).
9. Define boundary conditions (**BCs** >> **Create**) and loads (**Loads** >> **Create**). There are several types available, and they can be applied to the model in different ways (discussed later).
10. Submit the analysis (**Analysis-1** >> **Run**), when it finishes, the results can be viewed (**Analysis-1** >> **Results**). Various post-processing options will be explained later.

In PrePoMax, it's possible to work using node and element sets, as well as surfaces. They can be defined before any boundary condition, load, constraint or contact is created and then referenced in the definition of the selected feature. However, it's usually preferred to add features without creating sets/surfaces first. In such cases, the selection tool within the feature window will let the user pick entities based on geometry or mesh, depending on the type of feature.

There are two main ways to work in PrePoMax – use options available in menus (menu bar and top toolbar) or select them in the model tree. Some more advanced options can be accessed only from the menu bar.

View controls in PrePoMax:

- zoom: mouse scroll;
- rotate model: middle mouse button or keyboard arrows;
- pan (move model): **Shift** + middle mouse button.

Pressing middle mouse button anywhere on the part moves that location to the center of the screen and sets it as rotation center.

For selection, use the left mouse button (drag the mouse to use the selection box - only items entirely inside the box are selected). Holding **Shift** when selecting adds entities while holding **Ctrl** removes them.

A search bar is available on top of each of the three main tabs. It can be used to filter geometry, FE model and results trees.

Input fields that use units support their conversion. The user can enter a value, edit the displayed default unit (for example change *MPa* to *GPa*) and the conversion will be done after pressing **Enter** - PrePoMax will display the value in default units.

In the case of failure (when PrePoMax suddenly closes), the user is asked if a crash recovery file should be used to regenerate the previous session.

It is possible to open and import files supported by PrePoMax using the Windows command window. The syntax is as follows:

```
PrePoMax.exe "file_path\file_name.extension" -us unit_system
```

File path is not necessary if the file is located in the same location as PrePoMax executable. The unit system types are:

- *M_KG_S_C*
- *MM_TON_S_C*
- *M_TON_S_C*
- *IN_LB_S_F*
- *UNIT_LESS*

As an example, the following command can be used to open the *Assembly.step* file in the *Models* directory of PrePoMax catalogue:

```
PrePoMax.exe "C:\PrePoMax v1.3.0\Models\Assembly.step" -us M_KG_S_C
```

PrePoMax supports postprocessing of analyses with some features that are not yet implemented for preprocessing in GUI - namely, sensitivity analyses and simulations involving state variables (SDV) can be visualized.

3.1 File Menu

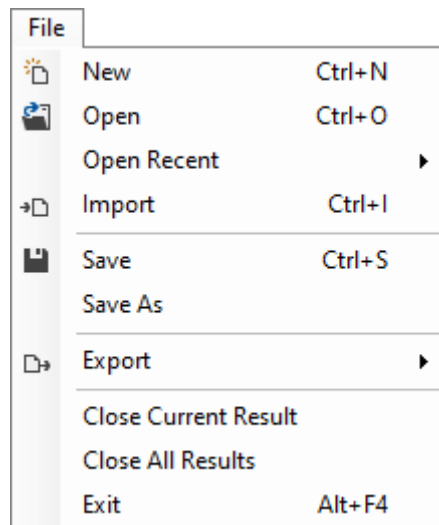


Figure 3: File menu

The **File** menu (Figure 3) contains the following options:

- New (**Ctrl** + **N**) – create a new model. The model space can be 3D, 2D Plane Stress, 2D Plane Strain or 2D Axisymmetric. The following unit systems are available (derived units are listed when a unit system is selected):
 - Unitless,
 - m, kg, s, °C – standard SI units,
 - mm, ton, s, °C – unit set typical for mechanical engineering,
 - m, ton, s, °C,

- in, lb, s, °C – imperial units.
- Open (Ctrl) + (O) – open a previously created model (.pmx), history (.pmh), CalculiX results (.frd) or OpenFOAM files (.foam)
- Open Recent – open one of the recently used models or clear the list of the recent files.
- Import (Ctrl) + (I) – import CAD geometry, mesh or model saved in one of the following formats:
 - STEP: .stp and .step (recommended in most cases),
 - IGES: .igs and .iges,
 - B-rep: .brep,
 - Stereolithography: .stl,
 - Universal: .unv,
 - Netgen: .vol,
 - Abaqus/CalculiX input files: .inp,
 - Mmg mesh: .mesh.
- Save (Ctrl) + (S) – save changes in the current model.
- Save As – save the model as .pmx file with a given name.
- Export – export part/model in one of the following formats:
 - STEP: .stp,
 - B-rep: .brep,
 - Stereolithography: .stl,
 - CalculiX input file: .inp,
 - Abaqus input file (experimental): .inp,
 - Mmg mesh: .mesh,
 - Deformed mesh: .inp (respects the current deformation scale factor, exports solid elements if output is set to 3D for analysis with shell elements),
 - Deformed visualization: .stl (respects the current deformation scale factor).
- Close Current Result – close the currently displayed results from the post-processing part of the model tree (saving the pmx model without the results reduces the file size).
- Close All Results – close all the results from the post-processing part of the model tree (saving the pmx model without the results reduces the file size).
- Exit (Ctrl) + (F4) – close PrePoMax (the user will be asked if the model should be saved first).

3.2 Edit Menu

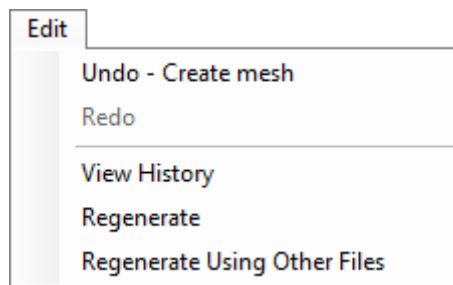


Figure 4: Edit menu

The **Edit** menu (Figure 4) contains the following options:

- Undo – undo the last action.
- Redo – redo the action that was undone previously.
- View History – open text file with history of user actions.
- Regenerate – regenerate the model in case of errors.
- Regenerate Using Other Files – swap the model's geometry with a similar one from another file so that it is not necessary to redefine the model.

3.3 View Menu

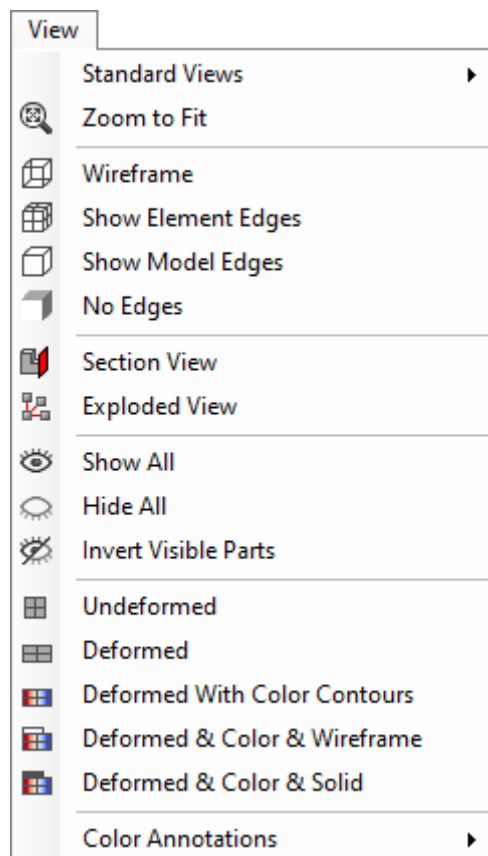


Figure 5: View menu

The **View** menu (Figure 5) contains the following options:

- Standard Views – apply one of the predefined views:
 - Front or Back,
 - Top or Bottom,
 - Left or Right,
 - Normal,
 - Vertical,
 - Isometric.
- Zoom to Fit – zoom the model in such a way that it fits in the window.
- Wireframe – apply a view showing only model edges.
- Show Element Edges – apply a view showing the model with the finite element edges.
- Show Model Edges – apply a view showing the model and its feature edges.
- No Edges – apply a view showing the model with no edges.
- Section View – cut the model using a plane defined by a point and a normal or disable the section view (this option can be used in post-processing as well).
- Exploded View – show assembly parts in an exploded view - 2 methods available:
 - Default: Exploded view direction, Magnification, Scale factor,
 - Center point: X/Y/Z coordinates, Exploded view direction, Magnification, Scale factor.
- Show All – show (unhide) all parts of the model.
- Hide All – hide all parts of the model.
- Invert Visible Parts – invert the visibility of parts.
- Undeformed – show undeformed model in results.
- Deformed – show deformed model in results.
- Deformed With Color Contours – show deformed model with contour plot in results.
- Deformed & Color & Wireframe – show deformed model with contour plot and wireframe representation of undeformed model in results.
- Deformed & Color & Solid – show deformed model with contour plot and solid representation of undeformed model in results.
- Color Annotations – use colors to distinguish items of the model based on various criteria: face orientations, parts, materials, sections, section thicknesses, all symbols, reference points, constraints, contact pairs, BCs, loads.



Figure 6: File and View toolbar

Selected options from the **File** and **View** menus can be accessed directly using a toolbar menu shown in Figure 6. This toolbar also contains two buttons for queries and annotations which will be discussed later.

3.4 Tools Menu

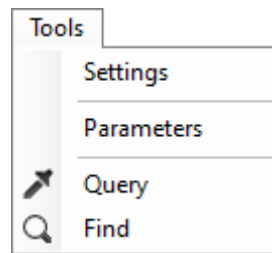


Figure 7: Tools menu

The **Tools** menu (Figure 7) contains the following options:

- Settings - to reset all settings in any of the tabs listed below to defaults, right-click in the empty space and select Reset all
 - General
 - * Open last file, last file name.
 - * Save results in .pmx files.
 - * Default unit system.
 - * Import mesh → Edge angle.
 - Graphics
 - * Background: Type, Top color, Bottom color.
 - * Geometry: CAD deflection.
 - * Lighting: Ambient component, Diffuse component.
 - * Smoothing: Point smoothing, Line smoothing.
 - * Widgets: Coordinate system visibility, Scale widget visibility.
 - Default Colors
 - * Face orientation: Front face color, Back face color.
 - * General.
 - Annotations
 - * Design: Background type, Draw a border rectangle, Number format, Number of significant digits
 - * Edge/Surface annotation: Show edge/surface id, Show edge/surface size, Show maximum value, Show minimum value, Show sum value
 - * Node annotation: Show node id, Show coordinates
 - * Part annotation: Show part name, Show part id, Show part type, Show number of elements, Show number of nodes
 - Meshing – default settings for meshing
 - * Mesh size: Mesh size definition, Max element factor, min element factor, Grading, Elements per edge, Elements per curvature, Hausdorff factor,
 - * Mesh optimization: Optimize steps 2D, Optimize steps 3D,
 - * Mesh type: Second order, Midside nodes on geometry, Quad-dominated mesh,
 - * Mesh operations: Split compound mesh, Keep model edges.
 - Pre-processing
 - * Color bar: Background type, Draw a border rectangle.

- * Selection: Primary/secondary/mouse highlight color.
- * Symbols: Constraints/Boundary condition/Loads color, Symbol size, Node symbol size, Draw symbol edges.
- Calculix
 - * Calculix: Work directory, Use .pmx folder as work directory, Executable, Default solver.
 - * Parallelization: Number of processors, Environment variables.
- Post-processing
 - * Undeformed model
 - Draw undeformed model: None/Wireframe body/Solid body,
 - Undeformed model color.
 - * Limit values
 - Show min value location: Yes/No,
 - Show max value location: Yes/No.
 - * History output
 - Max history output entries.
- Legend
 - * Color spectrum settings
 - Color spectrum type: Cool-warm, Rainbow, Warm, Cool, Cividis, Viridis, Plasma, Black body, Inferno, Kindlmann, Grayscale,
 - Brightness,
 - Reverse colors: Yes/No,
 - Number of discrete colors – in the range of 2-24.
 - * Color spectrum values
 - Number format: Scientific/General,
 - Number of significant digits – in the range of 2-8,
 - Min/Max limit type: Automatic/Manual.
 - * Design
 - Background type: None/White,
 - Draw a border rectangle: Yes/No.
- Status Block
 - * Design
 - Background type: None/White,
 - Draw a border rectangle: Yes/No.
- Parameters
 - Name
 - Value/Equation
 - Evaluates to
- Query
 - Vertex/Node – check the vertex/node ID, coordinates and the value of the currently displayed field output in this node (in Results).
 - Facet/Element – check the facet/element ID.

- Edge – check the edge ID and its length as well as maximum, minimum and total value of the currently displayed field output along this edge (in Results).
 - Surface – check the surface ID and its area as well as maximum, minimum and total value of the currently displayed field output on this surface (in Results)
 - Part – check part information (name, id, type, number of elements and nodes).
 - Assembly – check the number of parts, elements and nodes.
 - Bounding box size – check the size of the model bounding box.
 - Distance – measure the distance between two nodes.
 - Angle – measure the angle using three points.
 - Circle – measure the radius of the circle passing through three selected points.
- Find
 - Item type: Vertex/Node, Facet/Element, Edge, Surface, Part.
 - Data: Add annotation, Item id.

Currently, most of the annotations created using the `Query` menu display output not only in the message area at the bottom of the PrePoMax window but also in the form of labels attached to the model. Right clicking on them shows the options to `Edit` (manually change the displayed content), `Reset` and `Delete` them. This way it is also possible to access the settings menu where annotations can be further adjusted. Clicking the `Clear` button at the bottom of the `Query` window, deletes all the visible annotations (it is possible to display several annotations of different type at the same time). There is also a `Remove annotations` button on the File and View toolbar (Figure 6). Next to it, there is a `Query` button - a shortcut for `Tools >> Query`. Moreover, annotations are saved in .pmx files.

The Find tool can be used to locate (highlight) geometric and mesh entities with a specified ID. It can be useful for example when looking for nodes or elements reported as problematic by the solver.

Unitless parameters can be defined and then used in equations for reference point, material, section, constraint, load and boundary condition definitions. To add an equation, one should start the entry field with the equal sign = and delete the unit from the field. The equations use the open-source Ncalc library and thus its functions are supported.

3.5 Help Menu

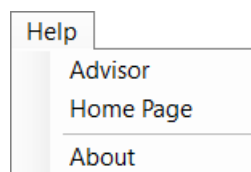


Figure 8: Help menu

The `Help` menu (Figure 8) contains the following options:

- Advisor – opens an interactive helper that will guide the user through the main steps to set up the analysis in PrePoMax.
- Home Page – opens the PrePoMax website in the default browser.
- About – displays the title card.

4 Geometry and mesh options

The *Geometry* tab of the model tree features three main containers – *Parts*, *Meshing parameters* and *Mesh Refinements*. The first one stores all parts in the model, and the following options are available when right-clicking the part:

- Edit – allows editing of the name and the color of the part (can be accessed also by double-clicking the part).
- Query – enables the Query tool.
- Preview Edge Mesh – displays points representing nodes in the mesh that will be generated (this way, it can be determined whether the size of the mesh is sufficient or not before generating it).
- Create Mesh – starts the mesher and creates the finite element mesh for the part.
- Copy Geometry to Results – allows to display geometry parts in the post-processing.
- Hide/Show/Show Only – options to hide and show the selected part, Spacebar can also be used to hide and show parts and other objects.
- Set Transparency – setting translucency color component of the part.
- Expand All/Collapse All – expands or collapses all objects listed in the Part container
- Delete – deletes the selected part.

4.1 Geometry Menu

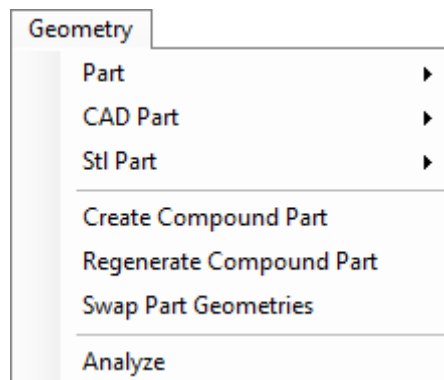


Figure 9: Geometry menu

The Geometry menu (Figure 9) contains the following options:

- Part – general part geometry options:
 - Edit: Name, Part type, Mesh, Appearance,
 - Transform,
 - * Scale.
 - Copy Geometry to Results,
 - Hide/Show/Show Only,
 - Set Transparency,
 - Delete.

- CAD Part - CAD part geometry options:
 - Flip Face Normal – flips the shell face orientation,
 - Split a Face Using Two Points – divides a face using two selected points by projection.
- Stl Part - STL part geometry options:
 - Find Model Edges by Angle – finds the model (feature) edges that fit into the specified angle criterion,
 - Flip Part Surfaces Normal – flips the orientation of the surfaces,
 - Smooth Part – applies smoothing to STL part.
- Create Compound Part – merges two or more parts into one part (body).
- Regenerate Compound Part – regenerates a compound part.
- Swap Part Geometries – swaps part geometries between two parts.
- Analyze – checks the geometry for the following:
 - Edges shorter than [specified value],
 - Edges closer than [specified value],
 - Faces smaller than [specified value],
 - Vertices with angle [specified value].

4.2 Mesh Menu

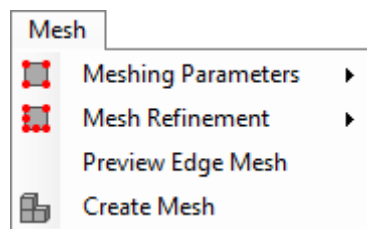


Figure 10: Mesh menu

The **Mesh** menu (Figure 10) contains the following options:

- Meshing Parameters: Create, Edit, Duplicate, Delete – settings for the mesher:
 - Data
 - * Name,
 - * Mesh settings: Basic, Advanced,
 - Mesh size
 - * Mesh size definition: Absolute, Relative,
 - * Max element size (for Absolute) or Max element factor (for Relative),
 - * Min element size (for Absolute) or Min element factor (for Relative),
 - * Grading: 0 – uniform mesh, 1 – aggressive local grading,
 - * Elements per edge – number of elements per edge of the geometry,
 - * Elements per curvature – number of elements per curvature radius.
 - Mesh optimization

- * Optimize steps 2D – number of optimization steps to use for the 2D mesh,
- * Optimize steps 3D – number of optimization steps to use for the 3D mesh.
- Mesh type
 - * Second order: Yes/No,
 - * Midside nodes on geometry – project the midside nodes on the geometry, options: Yes/No,
 - * Quad-dominated mesh: Yes/No (for shell and 2D parts only).
- Mesh operations
 - * Split compound mesh: Yes/No (splits compound part mesh to unconnected part meshes).
- Mesh Refinement: Create, Edit, Duplicate, Delete – settings for local mesh refinement:
 - Name,
 - Element size.
- Preview Edge Mesh
- Create Mesh

The relative mesh size specification was introduced for cases when the mesh should be denser only for the selected parts of the assembly and those parts are of different size so there's no easy way to achieve this with a single absolute setting. Relative settings enable the creation of equally denser meshes on all parts in such cases.

The Mesh refinement option can be used for local mesh controls. A surface, edge or vertex can be selected, and local mesh size can be specified. The mesh refinement can be previewed before generating the mesh.

5 FE Model options

Switching to FE Model tab activates the following menus: **Model**, **Property**, **Interaction**, **Amplitude**, **Initial condition**, **Step** and **Analysis**. They can be used to define various FEM features. The same features can also be created using the model tree.

5.1 Model Menu

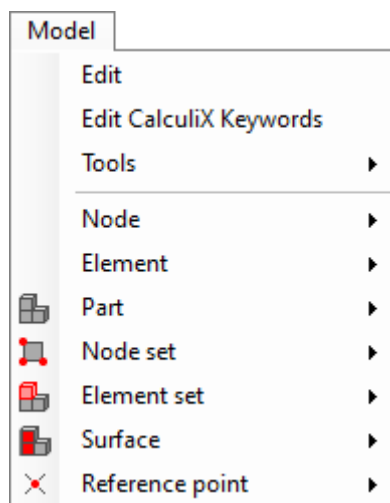


Figure 11: Model menu

The Model menu (Figure 11) contains the following options:

- Edit
 - Name,
 - Model space:
 - * for 3D models: 3D,
 - * for 2D models: 2D plane stress, 2D plane strain or 2D axisymmetric,
 - Model type:
 - * General model,
 - * Submodel:
 - Global results .frd,
 - * Slip wear model:
 - Results: All/Slip wear steps/Last increment of slip wear steps/Last increment of last slip wear step,
 - Number of cycles - number of slip wear cycles,
 - Cycles increment - increment of slip wear cycles,
 - Wear smoothing steps - number of slip wear smoothing steps,
 - BDM remeshing: On/Off - enabling boundary displacement method for remeshing due to the applied surface wear displacements after each wear cycle, boundary displacement step must be used to define fixed model regions to prevent all rigid body motions.
 - Absolute zero – value of the absolute zero temperature used for radiation,
 - Stefan-Boltzmann – value of the Stefan-Boltzmann constant used for radiation,
 - Gravitational constant – value of the Newton's gravitational constant.
- Edit CalculiX Keywords – a keyword editor that allows adding CalculiX keywords which are currently unsupported by PrePoMax.
- Tools
 - Create Boundary Layer – creates a prismatic boundary layer on the surface (used in analyses with a pre-tension load), the thickness of the layer can be specified, preview option is also available,
 - Find Model Edges By Angle (searches for the feature edges based on the angle in the imported meshes),
 - Remesh Elements – remeshes the selected shell region using specified max/min element size and max Hausdorff distance for boundaries approximation (0.01 is suitable for objects of size 1 in all directions).
 - Update Nodal Coordinates From File – updates nodal mesh coordinates from .inp files based on node IDs, can be used to introduce imperfections to nonlinear buckling analysis based on the scaled mode shape from linear buckling analysis (previously exported as deformed mesh to the .inp file)
- Node
 - Renumber All
 - * Start node id.

- Element
 - Renumber All
 - * Start node id.
- Part
 - Edit: Name, Part type, Mesh, Element type (can be changed; in the case of axisymmetric models, plane stress elements can also be selected since CalculiX supports axisymmetric + plane stress mixed models), Appearance,
 - Transform: Translate, Scale, Rotate,
 - Merge – assigns elements from multiple separate parts to one common part without changing the mesh,
 - Hide, Show and Show Only,
 - Set Transparency,
 - Delete.
- Node set
 - Create,
 - Edit: Name, Number of nodes, Center of gravity, Bounding box center,
 - Duplicate,
 - Delete.
- Element set
 - Create,
 - Edit: Name, Number of elements,
 - Duplicate,
 - Convert to Part - part created this way can be deleted to remove selected elements from the model,
 - Delete.
- Surface
 - Create,
 - Edit: Name, Surface type, Region type,
 - Duplicate,
 - Delete.
- Reference point
 - Create,
 - Edit,
 - Duplicate,
 - Delete.

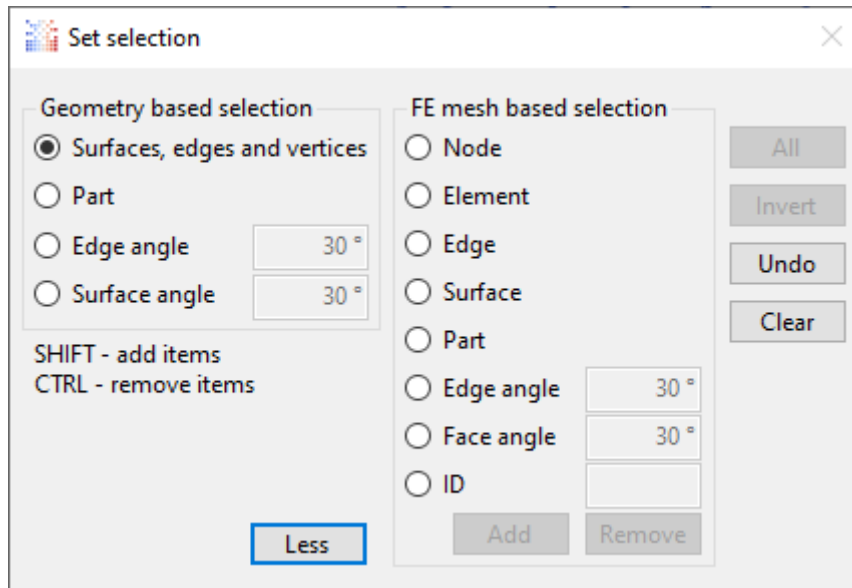


Figure 12: Set selection window

When creating sets and surfaces, a Set selection window opens (Figure 12). Sets and surfaces are named groups of nodes/elements and element faces used to define specific FEM features discussed below. Geometry based selection is more limited in terms of features, but its definition is not lost when the model is remeshed. FE mesh based selection offers more options, but if the model is remeshed the selection definition becomes invalid since the mesh has changed.

Negative face angle in FE mesh based selection mode of set/surface creation can be used to select a single element face.

Surfaces can be of element type or node type. They can be also created from existing node sets.

Reference points can be created using one of the following methods:

- Selection/Coordinates,
- Between two points,
- Circle center by 3 points,
- Center of gravity,
- Bounding box center.

The user can also change the color of the reference point.

5.2 Property Menu

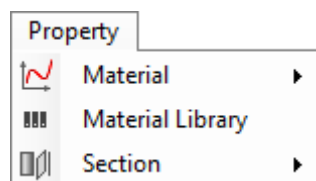


Figure 13: Property menu

The `Property` menu (Figure 13) contains the following options:

- Material

- Create,
 - Edit,
 - Duplicate,
 - Import from .inp,
 - Export to .inp,
 - Delete.
- Material Library – several materials are built into PrePoMax and the user can add new materials to the library as well.
- Section
 - Create,
 - Edit,
 - Duplicate,
 - Delete.

Material Library editor (Figure 14) allows access to materials library that is stored in the materials.lib file in the base PrePoMax directory. The library file is a text-based file that can be edited using any .txt editor, like Notepad++. It is assumed that all material data is stored using the mm, ton, s, °C unit system. But the library file is not meant to be edited this way since it is created directly from the material objects using JSON serialization and thus not easily readable. The library is meant to be edited using the PrePoMax built-in material editor.

5.2.1 Libraries

Multiple material libraries can be managed using the Libraries section at the top of the Material Library Editor.

5.2.2 Library materials section

The material library items consist of categories and materials. Categories are used to organize the materials into smaller manageable groups. A new category can be created inside the existing category using the button and removed from the library using the button. Each item in the library can be renamed using the bottom text field and the button. If changes were made to the library, they must be saved using button. All materials inside one category must have a unique name.

5.2.3 FE model materials section

The section for the model materials contains only button which is used to remove the materials from the FE model. All materials in the model must have a unique name.

5.2.4 Using the material library

To use an existing material in the FE model, the user must first select the material in the library material section. Using the right arrow button, the selected library material can be copied to the model materials.

To add a user material to the material library the material must be created first using the Material editor. Then the material library editor (Figure 14) can be used to add the user material to the library. In the material library editor, a desired category must be selected or created first and then the left arrow button can be used to copy the user material to the library. Once a material is added to the library the changes must be saved for future use.

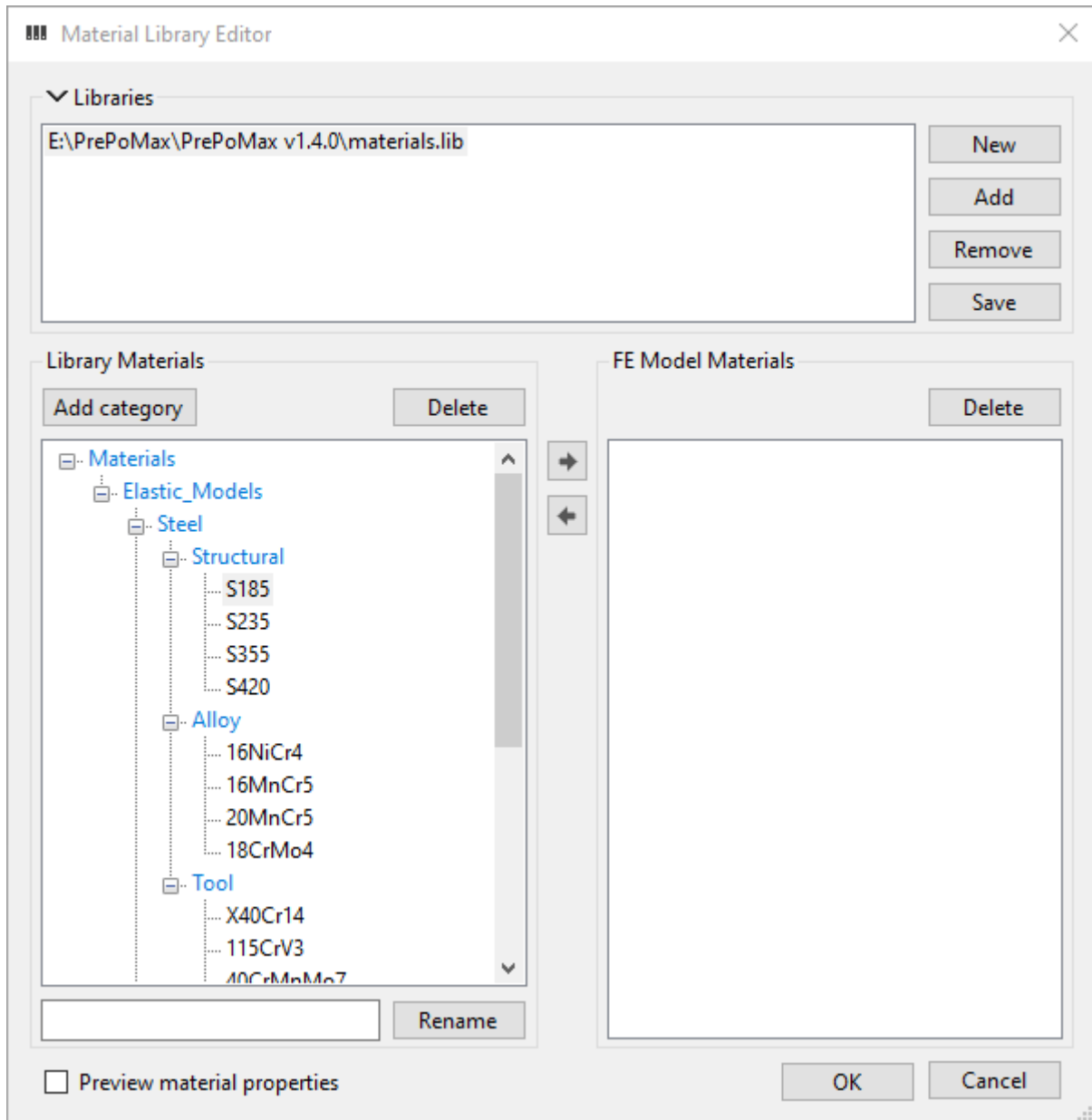


Figure 14: Material library

5.2.5 Exchanging material data between users

All material data in the material library can be replaced by replacing the `materials.lib` file. In this way the whole materials library can be sent to a second user. But replacing the library file also removes all the changes done by the second user.

If only a selected material is to be exchanged with the second user, a PrePoMax model `.pmx` file must be used. The first user creates a simple (but not empty) model containing the selected user material. This file must then be sent to the second user who opens the model file and adds the user material from the file to his materials library using the Material library editor (Figure 14). At the bottom of the Material Library Editor there is a checkbox to enable preview of properties of a selected material.

Usage of the Material library is presented in the following video: [PrePoMax & CalculiX - Material Library](#).

Since PrePoMax 1.3.0 there are two new ways of transferring the material model from one FE model to another FE model or from one computer to another computer. The first possibility is the transfer using the material library files and the second is the possibility of importing/exporting material models using the CalculiX/Abaqus .inp files.

Material libraries: PrePoMax supports editing of multiple material libraries using the built-in material library editor (Figure 14). The added Libraries group-box contains the list of opened library files and the following buttons for their manipulation:

- New - creates a new, empty material library file.
- Add - adds an existing material library file to the list of opened libraries.
- Remove - removes the selected opened library from the list (the material library file is not deleted).
- Save - saves the changes (denoted by an asterisk at the end of the file name) in the selected material library to the material library file.

To transfer the material model from source FE model to the target FE model the built-in default material library can be used, or a new material library can be created. The material model in the source FE model should then be copied from the FE model to the library by firstly selecting the library in the list of opened libraries, secondly selecting the material model in the FE Model materials group-box and lastly pressing the left arrow button. After that the updated library should be saved.

To copy the saved material model from the library to the target FE model, the target FE model must be opened first. Then the material library editor should be opened and the appropriate material library added or selected in the list of opened material libraries. Lastly, the material model in the library should be selected and copied to the FE model by pressing the right arrow button.

In order to transfer the material model from the source computer to the target computer the same process should be followed. But after saving the material model to a library file, the library file should be sent from the source computer to the target computer.

This method is unit aware which means that transferring the material model from one FE model to another FE model that is using a different unit system is possible and all material parameters are converted accordingly.

CalculiX/Abaqus .inp files: Additional menu entries were added to the *Property -> Material* menu. These new menus are:


- Import from .inp - imports all materials from a selected .inp file.
- Export to .inp - opens a selection window where materials from the FE model should first be selected and then saved to a new .inp file.

Transferring the material models from one FE model to another FE model or from one computer to another computer is done with the same procedure as described in **Material libraries** paragraph. This method is not unit aware which means that when transferring the material model from one FE model to another FE model that is using a different unit system the material properties are not converted.

When a new material is created, it is possible to define:

- Density,
- Slip wear

- Hardness.
- Wear coefficient.
- Elasticity
 - Young's modulus.
 - Poisson's ratio.
- Plasticity
 - Hardening:
 - * Isotropic.
 - * Kinematic.
 - * Combined.
 - Data points (yield stress vs plastic strain).
- Thermal properties
 - Thermal expansion:
 - * Thermal expansion coefficient.
 - * Zero temperature.
 - Thermal conductivity.
 - Specific heat.

The arrow button  is used to add a selected property to the new material definition.

It's also possible to specify the material's name, provide the description and include temperature dependency (Use temperature dependent data points).

There are three available section types:

- Solid section
 - Name – specify the section name.
 - Material – select the material for the section.
 - Thickness – specify the thickness of plane stress/strain part (only for 2D models).
 - Region – select the region for the section assignment.
- Shell section
 - Name – specify the section name.
 - Material – select the material for the section.
 - Thickness – define the shell thickness.
 - Offset – set the offset of the shell mid-surface (shell thickness is the unit).
 - Region – select the region for the section assignment.
- Membrane section
 - Name – specify the section name.
 - Material – select the material for the section.
 - Thickness – define the membrane thickness.
 - Offset – set the offset of the membrane mid-surface (membrane thickness is the unit).
 - Region – select the region for the section assignment.

5.3 Interaction Menu

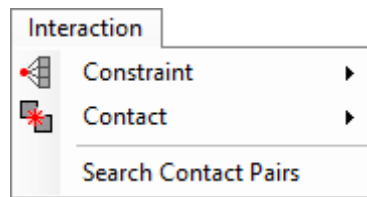


Figure 15: Interaction menu

The Interaction menu (Figure 15) contains the following options:

- Constraint
 - Create.
 - Edit.
 - Duplicate.
 - Swap Master/Slave.
 - Merge by Master/Slave.
 - Hide.
 - Show.
 - Delete.
- Contact:
 - Surface Interaction:
 - * Create.
 - * Edit.
 - * Duplicate.
 - * Delete.
 - Contact Pair:
 - * Create.
 - * Edit.
 - * Duplicate.
 - * Swap Master/Slave.
 - * Merge by Master/Slave.
 - * Hide.
 - * Show.
 - * Delete.
- Search Contact Pairs.

Available constraints include:

- Point spring:
 - Name.
 - Region.
 - Stiffness:

- * K1 – stiffness per node in the X direction.
 - * K2 – stiffness per node in the Y direction.
 - * K3 – stiffness per node in the Z direction.
 - Color.
- Surface spring:
 - Name.
 - Region.
 - Stiffness per area: No/Yes
 - Stiffness:
 - * K1 – stiffness in the X direction.
 - * K2 – stiffness in the Y direction.
 - * K3 – stiffness in the Z direction.
 - Color.
- Rigid body:
 - Name.
 - Reference point.
 - Region.
 - Color.
- Tie:
 - Name.
 - Position tolerance – the largest distance from the master surface for which the slave nodes will be included in the tie constraint (default: 2.5% of the typical element size).
 - Adjust – projection of the slave nodes on the master surface.
 - Master region.
 - Slave region.
 - Master surface color.
 - Slave surface color.

The rigid body constraint requires a reference point since it constrains the motion of the selected nodes to the motion of the reference point. It is used to define a selected region as infinitely rigid (non-deformable). For example, it's often used when modelling manufacturing tools. However, it may also serve as a way to apply moments and remote loads to models. Rigid body constraint is the only way to apply a torque to the surface of a solid part (since solid element nodes don't have the rotational degrees of freedom). Its usage in aforementioned applications is presented in the following videos: [PrePoMax \(CalculiX FEA\) - Tutorial 8 - Remote load](#) and [PrePoMax \(CalculiX FEA\) - Tutorial 2 - Torsion of an elliptical bar](#).

The tie constraint works like a bonded/welded contact. It connects two surfaces in such a way that they can't separate or move relative to each other during the analysis, they are permanently connected. Tie constraint is often used to connect parts with dissimilar meshes.

Contact is a more complex form of interaction between two surfaces than the tie constraint. It allows the transfer of forces between touching surfaces. However, contact is a highly nonlinear phenomenon and requires careful usage as it may cause convergence issues. When defining contact, a surface interaction must be created first. Three options are available:

- Surface behavior – behavior of the contact in the normal contact direction (pressure-overclosure relationship)
 - Hard.
 - Linear.
 - Exponential.
 - Tabular.
 - Tied.
- Friction – behavior in the tangential direction
 - Friction coefficient.
 - Stick slope.
- Gap conductance – thermal conductance across the contact interface
 - Constant
 - Tabular – pressure and temperature dependent

When the surface interaction is created, the contact pairs can be defined. The options here are as follows:

- Name
- Surface interaction
- Method
 - Node to surface – uses node-to-face penalty formulation.
 - Surface to surface – uses face-to-face penalty formulation.
 - Mortar – uses formulation with Lagrange multipliers, typically more accurate but requires more time to solve and may cause convergence issues.
 - Linear mortar – should be used only when Mortar fails.
 - PG linear mortar – should be used only when Mortar fails.
- Small sliding – available when Node to surface method is selected
- Adjust – projection of the slave nodes on the master surface.
- Adjustment size – distance inside which the slave nodes are projected to the master surface.
- Master region
- Slave region
- Master surface color
- Slave surface color

The usage of contact is presented in the following videos: [PrePoMax & CalculiX - GUI Contact](#) and [PrePoMax \(CalculiX FEA\) - Tutorial 10 - Hertz contact](#).

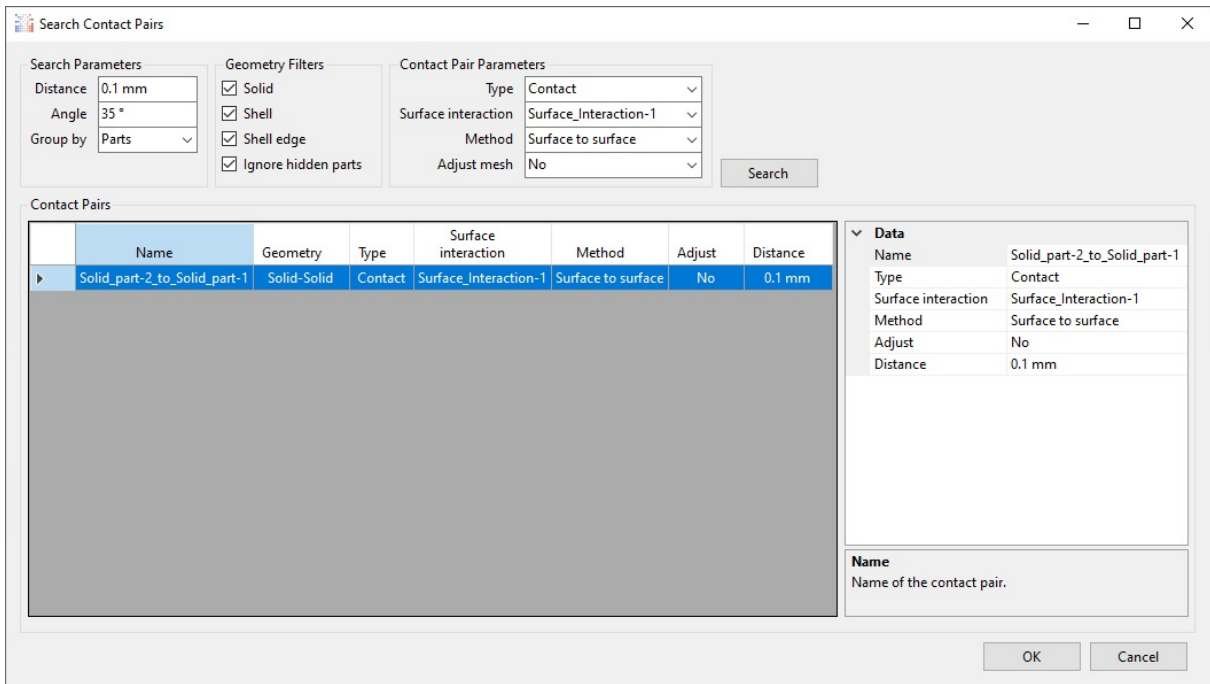


Figure 16: Search Contact Pairs tool

The *Search Contact Pairs* tool with exemplary input is shown in Figure 16. This tool allows users to automatically create contact pairs and tie constraints. Upon clicking the `Search` button the tool searches for surface pairs that meet the following criteria:

- distance between them is smaller or equal to the specified distance.
- angle between them is smaller or equal to the specified angle.

In addition to these criteria, user may choose whether the pairs are grouped by parts or not. Before performing the search, one can change the type of interaction (Contact or Tie), specify previously created contact property definition and choose the type of contact (surface to surface or node to surface). For both contact and tie creation, it's possible to enable or disable adjustment. After the search, users can still change the aforementioned settings (as well as position tolerance and name of the pair) using the *Data* window on the right side of the tool. In addition, right clicking on the pair in *Contact Pairs* window brings an option to swap master and slave surfaces. When more than one pair is selected, an option to merge by master/slave also becomes available. Pairs can be removed from the list (and thus prevented from being created) by selecting them and pressing `Delete`. The usage of this tool is presented in the following videos: [PrePoMax \(CalculiX FEA\) - Tutorial 14 - Chain links](#) and [PrePoMax \(CalculiX FEA\) - Tutorial 15 - Shell-to-solid connection](#).

5.4 Amplitude menu

The `Amplitude` menu (Figure 17) contains the following options:

- Create.
- Edit.
- Duplicate.
- Delete.

The following amplitude types and settings are available:

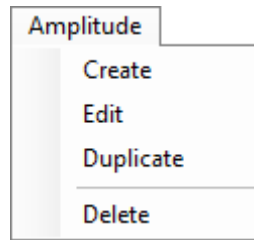


Figure 17: Amplitude menu

- Tabular – can be used to specify the time/frequency - amplitude value data points
 - Properties
 - * Name
 - * Time span: Step time/Total time
 - * Shift time
 - * Shift amplitude
 - Data points - tabular data in form of time/frequency-amplitude pairs

5.5 Initial Condition Menu

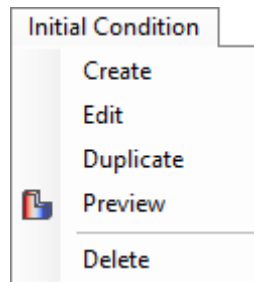


Figure 18: Initial condition menu

The **Initial condition** menu (Figure 18) contains the following options:

- Create.
- Edit.
- Duplicate.
- Preview.
- Delete.

The following initial condition types and settings are available:

- Temperature – can be used to specify the temperature at the beginning of a thermal or thermo-mechanical analysis
 - Name.
 - Region.
 - Temperature.

- Velocity – can be used to specify the velocity at the beginning of a dynamic analysis
 - Name.
 - Region.
 - Velocity components:
 - * V1 - velocity in the X axis.
 - * V2 - velocity in the Y axis.
 - * V3 - velocity in the Z axis.
 - Magnitude - resultant velocity (it's calculated automatically when the components of the velocity are specified).

5.6 Step Menu

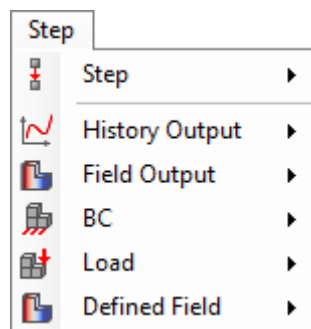


Figure 19: Step menu

The **Step** menu (Figure 19) contains the following options:

- Step
 - Create.
 - Edit.
 - Duplicate.
 - Delete.
- History Output
 - Create.
 - Edit.
 - Duplicate.
 - Propagate – propagates the definition to subsequent steps.
 - Delete.
- Field Output
 - Create.
 - Edit.
 - Duplicate.
 - Propagate – propagates the definition to subsequent steps.
 - Delete.

- BC – Boundary condition
 - Create.
 - Edit.
 - Duplicate.
 - Propagate – propagates the definition to subsequent steps.
 - Hide.
 - Show.
 - Delete.

- Load
 - Create.
 - Edit.
 - Duplicate.
 - Preview – can be used to preview pressure loads as contour plots in the Results tab, especially useful for hydrostatic and imported pressure loads.
 - Propagate – propagates the definition to subsequent steps.
 - Hide.
 - Show.
 - Delete.

- Defined Field
 - Create.
 - Edit.
 - Duplicate.
 - Propagate – propagates the definition to subsequent steps.
 - Preview – can be used to preview defined temperature fields as contour plots in the Results tab, especially useful for fields defined from results (.frd) files.
 - Delete.

5.6.1 Step

The step types and their options available in PrePoMax are:

- Static step
 - Name.
 - Solver: Default, PaStiX, Pardiso, Spooles, Iterative scaling, Iterative Cholesky.
 - Nlgeom: On/Off (enable or disable the geometric nonlinearity – large deformations and large displacements).
 - Incrementation: Default/Automatic/Direct (the incrementation options listed below are for Automatic incrementation).
 - Max increments – maximum number of increments in the step.
 - Time period – time period of the step.
 - Initial time increment – initial value of time increment in the step.

- Min time increment – minimum time increment allowed.
- Max time increment – maximum time increment allowed.
- Output frequency – integer N indicating that only results of every N-th increment will be stored.
- Slip wear step
 - Name.
 - Solver: Default, PaStiX, Pardiso, Spooles, Iterative scaling, Iterative Cholesky.
 - Nlgeom: On/Off (enable or disable the geometric nonlinearity – large deformations and large displacements).
 - Incrementation: Default/Automatic/Direct (the options listed below are for Automatic incrementation).
 - Max increments – maximum number of increments in the step.
 - Time period – time period of the step.
 - Initial time increment – initial value of time increment in the step.
 - Min time increment – minimum time increment allowed.
 - Max time increment – maximum time increment allowed.
 - Output frequency – integer N indicating that only results of every N-th increment will be stored.
- Boundary displacement step
 - Name.
 - Output frequency – integer N indicating that only results of every N-th increment will be stored.
- Frequency step
 - Name.
 - Solver: Default, PaStiX, Pardiso, Spooles, Iterative scaling, Iterative Cholesky.
 - Perturbation: On/Off (when it's set to On, preload from the previous static step can be included).
 - Storage: Yes/No – stores eigenvalues, eigenmodes, mass and stiffness matrix in a binary form in a file jobname.eig for further use in steady state dynamics or modal dynamics step.
 - Number of frequencies – number of eigenfrequencies to compute.
 - Lower frequency bound – lower bound of the frequency range.
 - Upper frequency bound – upper bound of the frequency range.
 - Output frequency – integer N indicating that only results of every N-th increment will be stored.
- Buckle step
 - Name.
 - Solver: Default, PaStiX, Pardiso, Spooles, Iterative scaling, Iterative Cholesky.
 - Perturbation: On/Off (when it's set to On, preload from previous static step can be included).

- Num of buckling factors – number of buckling factors desired (default: 1).
- Accuracy – accuracy desired (default is 0.01).
- Output frequency – integer N indicating that only results of every N-th increment will be stored.
- Modal dynamics step
 - Name.
 - Solver: Default, PaStiX, Pardiso, Spooles, Iterative scaling, Iterative Cholesky.
 - Perturbation: On/Off (when it's set to On, preload from previous static step can be included).
 - Steady state: On/Off (can be used to continue a modal dynamics calculation until steady state has been reached).
 - Damping type:
 - * Off
 - * Constant: Damping ratio.
 - * Direct: Damping ratios (table: Lowest mode, Highest mode, Damping ratio)
 - * Rayleigh: Alpha, Beta
 - Max increments – maximum number of increments in the step.
 - Initial time increment – initial value of time increment in the step.
 - Time period – time period of the step (when steady state is off) or Relative error – relative error for the solution to be considered as steady state (when steady state is on).
 - Output frequency – integer N indicating that only results of every N-th increment will be stored.
- Steady state dynamics step
 - Name.
 - Solver: Default, PaStiX, Pardiso, Spooles, Iterative scaling, Iterative Cholesky.
 - Perturbation: On/Off (when it's set to On, preload from previous static step can be included).
 - Harmonic: Yes/No (switches between harmonic periodic loading and nonharmonic periodic loading).
 - Lower frequency bound – lower bound of the frequency range.
 - Upper frequency bound – upper bound of the frequency range.
 - Number of data points – number of data points within the frequency range.
 - Bias – distribution bias of the data points within the frequency range (1 for equal spacing).
 - Damping type:
 - * Off
 - * Constant: Damping ratio.
 - * Direct: Damping ratios (table: Lowest mode, Highest mode, Damping ratio)
 - * Rayleigh: Alpha, Beta
 - Output frequency – integer N indicating that only results of every N-th increment will be stored.
- Dynamic step

- Name.
 - Solver: Default, PaStiX, Pardiso, Spooles, Iterative scaling, Iterative Cholesky.
 - Nlgeom: On/Off (enable or disable the geometric nonlinearity – large deformations and large displacements).
 - Alpha – amount of numerical damping (between -1/3 and 0).
 - Solution procedure :
 - * Implicit/Implicit– implicit structural and implicit fluid (currently unsupported) computations.
 - * Implicit/Explicit– implicit structural and explicit fluid (currently unsupported) computations.
 - * Explicit/Implicit– explicit structural and implicit fluid (currently unsupported) computations.
 - * Explicit/Explicit– explicit structural and explicit fluid (currently unsupported) computations.
 - Relative to absolute : On/Off (needs to be enabled when the coordinate system in the previous step was attached to a rotating system and the coordinate system in the present dynamic step should be absolute).
 - Damping type:
 - * Off
 - * Rayleigh: Alpha, Beta
 - Incrementation: Default/Automatic/Direct (the incrementation options listed below are for Automatic incrementation).
 - Max increments – maximum number of increments in the step.
 - Time period – time period of the step.
 - Initial time increment – initial value of time increment in the step.
 - Min time increment – minimum time increment allowed, should be set to the desired minimum time increment if mass scaling is to be used in explicit dynamic calculations.
 - Max time increment – maximum time increment allowed.
 - Output frequency – integer N indicating that only results of every N-th increment will be stored.
- Heat transfer step
 - Name.
 - Solver: Default, PaStiX, Pardiso, Spooles, Iterative scaling, Iterative Cholesky.
 - Steady state: Yes/No – switching between the steady state and the transient analysis
 - Incrementation: Default/Automatic/Direct (the options listed below are for Automatic incrementation).
 - Max increments – maximum number of increments in the step.
 - Time period – time period of the step.
 - Initial time increment – initial value of time increment in the step.
 - Min time increment – minimum time increment allowed.
 - Max time increment – maximum time increment allowed.
 - Output frequency – integer N indicating that only results of every N-th increment will be stored.

- Uncoupled temperature-displacement step
 - Name.
 - Solver: Default, PaStiX, Pardiso, Spooles, Iterative scaling, Iterative Cholesky.
 - Nlgeom: On/Off (enable or disable the geometric nonlinearity – large deformations and large displacements).
 - Steady state: Yes/No – switching between the steady state and the transient analysis
 - Incrementation: Default/Automatic/Direct (the options listed below are for Automatic incrementation).
 - Max increments – maximum number of increments in the step.
 - Time period – time period of the step.
 - Initial time increment – initial value of time increment in the step.
 - Min time increment – minimum time increment allowed.
 - Max time increment – maximum time increment allowed.
 - Output frequency – integer N indicating that only results of every N-th increment will be stored.

- Coupled temperature-displacement step
 - Name.
 - Solver: Default, PaStiX, Pardiso, Spooles, Iterative scaling, Iterative Cholesky.
 - Nlgeom: On/Off (enable or disable the geometric nonlinearity – large deformations and large displacements).
 - Steady state: Yes/No – switching between the steady state and the transient analysis
 - Incrementation: Default/Automatic/Direct (the options listed below are for Automatic incrementation).
 - Max increments – maximum number of increments in the step.
 - Time period – time period of the step.
 - Initial time increment – initial value of time increment in the step.
 - Min time increment – minimum time increment allowed.
 - Max time increment – maximum time increment allowed.
 - Output frequency – integer N indicating that only results of every N-th increment will be stored.

The slip wear step and the boundary displacement step (available only after adding the slip wear step) are not standard CalculiX analysis procedures and thus require additional explanation. They are based on the Archard's model [1] and can be used to estimate the progressing sliding wear between contact surfaces. This linear model predicts the volume of the removed material due to the wear process. Since the model neglects the changes in the geometry of the contacting surfaces during multiple wear cycles, a multistep numerical procedure can be used to determine the wear parameters better. Archard's wear model was developed based on the results of the wear experiments between metals in dry conditions. Usually, the model is described as:

$$V = k \cdot \frac{F}{H} \cdot s \quad (1)$$

where: V - wear volume, k - dimensionless wear coefficient, F - normal contact load, H - hardness of the worn surface, s - sliding distance.

This equation has to be rewritten to the following form to use it in a multistep numerical procedure where the change of the wear depth must be computed for each wear cycle [2]:

$$h_i = k \int \frac{k}{H} \cdot p_i \cdot ds \quad (2)$$

where: h_i - wear depth accumulated in the i -th wear cycle, p_i - normal contact pressure.

The wear depth after multiple wear cycles is then computed by summation of all single cycle wear depths.

The implementation of the sliding wear analysis in the PrePoMax is general in terms of model geometry, boundary conditions and loads and involves the following assumptions:

- the continuous wear process between the parts can be discretized into several repeating wear cycles,
- the wear occurs only on a softer surface in the contact pair,
- the geometry of the contacting surfaces does not change during a single wear cycle,
- the dimensionless wear coefficient remains constant during the entire wear process.

Based on these assumptions, a finite element model of a single wear step is first prepared. The model must accurately capture the evolution of the contact conditions through time; thus, a non-linear time-dependent analysis is required. The wear computation starts after the completion of the analysis. All computations of the wear parameters are done in the finite element nodes of the slave surface, which are in contact during the wear cycle. Nodal pressure and slip values are computed from the contact pressure slip fields. The nodal wear depths are then calculated using equation (2) and material properties at contact nodes. The processing of a single wear cycle completes with the computation of the contact surface geometry change due to wear, defined by nodal wear displacements. Single-cycle wear displacements are calculated by multiplying the scalar nodal wear depth value by the nodal surface normal. Wear displacements of a single cycle are then added to the wear displacements of the previous wear cycles.

The combined wear displacements are then considered during model preparation for the next wear cycle. The initial finite element mesh of the model is updated by the combined nodal wear displacements considering the model boundary conditions. If a zero boundary condition is prescribed on a nodal degree of freedom where the wear displacement component is computed, the wear displacement component is set to zero. The analysis of the symmetric wear models is possible in this way. The procedure repeats until the final number of wear cycles is reached.

References:

1. J. F. Archard and W. Hirst, "The wear of metals under unlubricated conditions", Proc. R. Soc. London. Ser. A. Math. Phys. Sci., vol. 236, no. 1206, 1956.
2. X. Shen, L. Cao, and R. Li, "Numerical simulation of sliding wear based on Archard model", in 2010 International Conference on Mechanic Automation and Control Engineering, MACE2010, 2010.

5.6.2 History Output

History output is used to generate X-Y plots in post-processing. Available options include:

- Node output
 - Name.

- Variables to output
 - * RF – reaction forces.
 - * U – displacements.
 - * V – velocities.
 - * NT – temperatures.
 - * RFL – external concentrated heat sources.
- Totals: Yes/Only/No – this parameter applies to the external forces, it enables output of values summed for all nodes in the set.
- Region – region for creation of the history output.

- Element output
 - Name.
 - Variables to output
 - * S – true (Cauchy) stress.
 - * E – total Lagrangian strain or total Eulerian strain.
 - * ME – mechanical Lagrangian strain or mechanical Eulerian strain.
 - * PEEQ – equivalent plastic strain.
 - * HFL – heat flux.
 - * ENER – energy density.
 - * ELSE – internal energy.
 - * ELKE – kinetic energy.
 - * EVOL – volume.
 - * EBHE – heating power.
 - Totals: Yes/Only/No – this parameter applies to the whole element variables (ELSE, EVOL), it enables output of values summed for all elements in the set.
 - Region – region for the creation of the history output.

- Contact output
 - Name.
 - Variables to output
 - * CDIS – relative contact displacements.
 - * CSTR – contact stresses.
 - * CELS – contact energy.
 - * CNUM – total number of contact elements.
 - * CF – total contact forces on slave surface.
 - Totals: Yes/Only/No – this parameter applies to the energy (CELS).
 - Contact pair – contact pair for creation of the history output.

No history outputs are created by default.

5.6.3 Field Output

Field output is used to create contour plots. Available options include:

- Node output
 - Name.
 - Last iterations: On/Off – disabled by default, turning it on enables storage of results from all iterations of the last increment in a separate file named ResultsForLastIterations.frd and can be useful for debugging of diverging analyses.
 - Contact elements: True/False – disabled by default, turning it on enables storage of contact elements in a file with .cel extension and their display in results.
 - Variables to output:
 - * RF – reaction forces.
 - * U – displacements.
 - * PU – displacements magnitude and phase.
 - * V – velocities.
 - * NT – temperatures.
 - * PNT – temperatures magnitude and phase.
 - * RFL – external concentrated heat sources.
- Element output
 - Name.
 - Last iterations: On/Off – disabled by default, turning it on enables storage of results from all iterations of the last increment in a separate file named ResultsForLastIterations.frd and can be useful for debugging of diverging analyses.
 - Contact elements: True/False – disabled by default, turning it on enables storage of contact elements in a file with .cel extension and their display in results.
 - Output (2D/3D):
 - * Default – results for shell elements are displayed in the expanded 3D form while for plane stress/strain and axisymmetric elements they are displayed in the original 2D form.
 - * 2D – results for all types of 2D elements are displayed in the original 2D form.
 - * 3D - results for all types of 2D elements are displayed in the expanded 3D form.
 - Variables to output:
 - * S – stresses.
 - * PHS – stresses magnitude and phase.
 - * E – total strains.
 - * ME – mechanical strains.
 - * PEEQ – equivalent plastic strain.
 - * ENER – energy density.
 - * HFL – heat flux.
 - * ERR – extrapolation error estimator for stress calculations (ERR and ZZS are mutually exclusive).
 - * HER – extrapolation error estimator for heat calculations (HER and ZZS are mutually exclusive).

- * ZZS – Zienkiewicz-Zhu improved stress (ZZS and ERR are mutually exclusive).
- Contact output
 - Name.
 - Last iterations: On/Off – disabled by default, turning it on enables storage of results from all iterations of the last increment in a separate file named ResultsForLastIterations.frd and can be useful for debugging of diverging analyses.
 - Variables to output
 - * CDIS – relative contact displacements.
 - * CSTR – contact stresses.
 - * PCON – contact states for frequency calculations.

Default field outputs are created in each new step added to the model.

5.6.4 Boundary condition

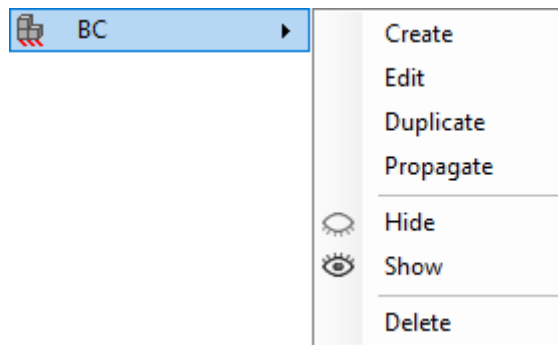


Figure 20: Boundary Conditions submenu

The Boundary Conditions (BC) submenu (Figure 20) contains the following options:

- Create.
- Edit.
- Duplicate.
- Propagate.
- Hide/Show.
- Delete.

The following boundary condition types and settings are available:

- Fixed
 - Name.
 - Region.
 - Color.
- Displacement/Rotation
 - Name.

- Region.
- DOF:
 - * U1 – translation in X axis.
 - * U2 – translation in Y axis.
 - * U3 – translation in Z axis.
 - * UR1 – rotation about X axis.
 - * UR2 – rotation about Y axis.
 - * UR3 – rotation about Z axis.
- Amplitude.
- Color.
- Submodel
 - Name.
 - Step number – global model step number from which to read the displacements.
 - Region.
 - DOF:
 - * U1 – translation in X axis.
 - * U2 – translation in Y axis.
 - * U3 – translation in Z axis.
 - * UR1 – rotation about X axis.
 - * UR2 – rotation about Y axis.
 - * UR3 – rotation about Z axis.
 - Color.
- Temperature
 - Name.
 - Region.
 - Magnitude.
 - Amplitude.
 - Color.

Fixed boundary condition constrains all the available degrees of freedom (DOF). Displacement/rotation BC allows the user to select individual DOFs. The following options are available for each DOF:

- Unconstrained – no boundary condition imposed on this DOF.
- Fixed – freezes the deformation from the previous step or, if there is no previous step, sets it to zero.
- Value (length unit/angle unit) – a prescribed value of displacement (current length units for translational DOFs and radians for rotational ones).

Keep in mind that a Fixed option and a Value option cannot be used simultaneously for various DOFs in the same boundary condition.

In the case of a submodel boundary condition, each DOF has the following two options:

- Unconstrained.
- From global model.

Usage of submodeling technique is presented in the following videos: [PrePoMax & CalculiX - Sub-model](#) and [PrePoMax \(CalculiX FEA\) - Tutorial 13 - Stress concentration - submodeling](#).

5.6.5 Load

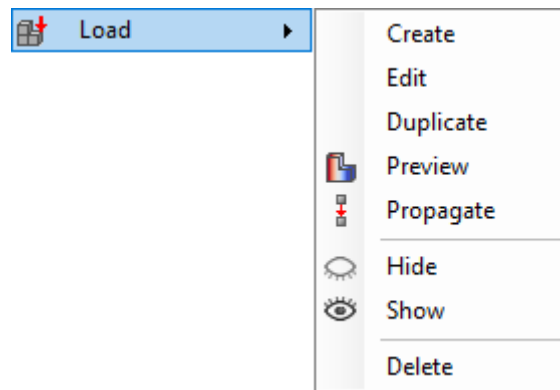


Figure 21: Load submenu

The **Load** submenu (Figure 21) contains the following options:

- Create.
- Edit.
- Duplicate.
- Preview.
- Propagate.
- Hide/Show.
- Delete.

The types of loads and their options are listed below:

- Concentrated force
 - Name.
 - Region.
 - Force components – components of the concentrated force per node:
 - * F1 – force in the X axis.
 - * F2 – force in the Y axis.
 - * F3 – force in the Z axis.
 - Magnitude – resultant force (it's calculated automatically when the components of the force are specified).
 - Phase – force phase (for steady state dynamics only).
 - Amplitude.
 - Color.
- Moment
 - Name.
 - Region.
 - Moment components – components of the moment per node:

- * M1 – moment about the X axis.
- * M2 – moment about the Y axis.
- * M3 – moment about the Z axis.
- Magnitude – resultant moment (it's calculated automatically when the components of the moment are specified).
- Phase – moment phase (for steady state dynamics only).
- Amplitude.
- Color.
- Uniform pressure
 - Name.
 - Region.
 - Magnitude – value of the pressure load.
 - Phase – pressure phase (for steady state dynamics only).
 - Amplitude.
 - Color.
- Hydrostatic pressure
 - Name.
 - Region.
 - First point coordinates – by selection or by direct input of coordinates:
 - * X.
 - * Y.
 - * Z.
 - First point pressure magnitude – pressure magnitude at the first point.
 - Second point coordinates – by selection or by direct input of coordinates:
 - * X.
 - * Y.
 - * Z.
 - Second point pressure magnitude – pressure magnitude at the second point.
 - Pressure change direction – by selection (two points) or by direction components:
 - * N1 – direction component in the direction of the first axis.
 - * N2 – direction component in the direction of the second axis.
 - * N3 – direction component in the direction of the third axis.
 - Phase – pressure phase (for steady state dynamics only).
 - Cutoff: None/ Positive cutoff/ Negative cutoff - positive pressure cutoff sets all positive pressure values to 0 while negative pressure cutoff sets all negative pressure values to 0.
 - Amplitude.
 - Color.
- Imported pressure
 - Name.

- Region.
- Import from file – OpenFOAM (.foam) file from which the pressure results will be imported.
- Interpolator: Closest node or Closest point – the first method takes the value from the closest node on the source mesh while the second method interpolates the closest three nodal values.
- Magnitude factor – pressure magnitude scale factor.
- Phase – pressure phase (for steady state dynamics only).
- Scale factor – imported geometry scale factor.
- Amplitude.
- Color.
- Surface traction
 - Name.
 - Region.
 - Force components:
 - * F1 – force in the X axis.
 - * F2 – force in the Y axis.
 - * F3 – force in the Z axis
 - Magnitude – resultant force (it's calculated automatically when the components of the force are specified).
 - Phase – surface traction phase (for steady state dynamics only).
 - Amplitude.
 - Color.
- Normal shell edge load
 - Name.
 - Region.
 - Magnitude – value of the shell edge load.
 - Phase – edge load phase (for steady state dynamics only).
 - Amplitude.
 - Color.
- Gravity
 - Name.
 - Region.
 - Gravity components:
 - * F1 – gravitational acceleration in the X axis.
 - * F2 – gravitational acceleration in the Y axis.
 - * F3 – gravitational acceleration in the Z axis.
 - Magnitude – resultant acceleration (it's calculated automatically when components of the force are specified).
 - Phase – gravity load phase (for steady state dynamics only).

- Amplitude.
 - Color.
- Centrifugal load
 - Name.
 - Region.
 - Rotation center coordinates – by selection or by direct input of coordinates:
 - * X.
 - * Y.
 - * Z.
 - Rotation axis components:
 - * N1 – axis component in the direction of the X axis.
 - * N2 – axis component in the direction of the Y axis.
 - * N3 – axis component in the direction of the Z axis.
 - Magnitude – rotational speed around the axis defined by a point and a direction.
 - Phase – rotational speed phase (for steady state dynamics only).
 - Amplitude.
 - Color.
- Pre-tension
 - Name.
 - Type: Force/Displacement.
 - Region.
 - Auto compute: Yes/No – automatic computation of the pre-tension direction.
 - Magnitude – force or displacement magnitude for the pretension load.
 - Amplitude.
 - Color.
- Concentrated flux
 - Name.
 - Add flux: Yes/No – add flux to the previously defined fluxes.
 - Region.
 - Flux – value of the flux per node.
 - Amplitude.
 - Color.
- Surface flux
 - Name.
 - Region.
 - Flux per area.
 - Amplitude.
 - Color.

- Body flux
 - Name.
 - Region.
 - Flux per volume.
 - Amplitude.
 - Color.
- Convective film
 - Name.
 - Region.
 - Sink temperature.
 - Film coefficient.
 - Amplitude – sink temperature amplitude.
 - Color.
- Radiation
 - Name.
 - Cavity radiation: Yes/No – an option to make the selected surfaces belong to a radiation in a cavity (cavity name has to be specified if this option is switched to Yes)
 - Region.
 - Sink temperature.
 - Emissivity – surface emissivity, blackbody radiation is characterized by 1.
 - Amplitude – sink temperature amplitude.
 - Color.

Concentrated force load applies the specified force to each node belonging to the selected region. Thus, if this load is applied to a surface, each node on that surface will be subjected to the specified force which is usually not desired. Thus, surface traction load should be used to apply force to surfaces.

Hydrostatic pressure is defined using pressure magnitudes at two points with given coordinates and the direction of the pressure increase (specified by selecting two points or by providing the vector components directly). This tool can be used to define not only the hydrostatic pressure but also general linearly varying distributed load (triangular or trapezoidal), not associated with the presence of a fluid. The usage of this type of load is presented in the following videos: [PrePoMax & CalculiX - Hydrostatic pressure](#) and [PrePoMax \(CalculiX FEA\) - Tutorial 28 - Hydrostatic pressure - rectangular pressure vessel](#).

Imported pressure uses pressure results from OpenFOAM CFD simulation as loads in a structural analysis. Both ASCII and binary OpenFOAM results are supported. A file with .foam extension is needed for this type of load. It's an empty (dummy) file used to access folders with OpenFOAM results. If parallel processing was used for OpenFOAM analysis, the *reconstructPar* command needs to be utilized after the CFD simulation to merge the results from different processors. It's not necessary to import the same file with geometry as the one used for CFD analysis but the geometry has to be located in the same place in a 3D space and use the same units of length as the ones used in OpenFOAM. The user needs to be careful with units since the pressure in OpenFOAM results might be given as kinematic pressure (pressure divided by density). The scale factor can be used to compensate for this. Preview option allows users to ensure that the imported pressure load is defined

correctly - the applied pressure multiplied by the scale factor is visualized there along with the distances (closest distance between the PrePoMax mesh and the OpenFOAM mesh). Those distances should be small since they are the result of noncoincident meshes. The usage of this type of load is presented in the following video: [PrePoMax \(CalculiX FEA\) - Tutorial 32 - Pressure imported from CFD \(with OpenFOAM and FreeCAD\)](#).

Surface traction load internally applies concentrated forces to each node on the selected surface. Their values sum up to the total magnitude specified in the load definition window. If multiple surfaces are selected, the load acting on each of them sums up to the value specified in the load definition window.

Despite the name, gravity load can be used to prescribe any type of translational acceleration to the structure (such as deceleration due to braking or acceleration of the vehicle). Its usage is presented in the following videos: [PrePoMax & CalculiX - Gravity](#) and [PrePoMax \(CalculiX FEA\) - Tutorial 7 - Gravity load](#).

Pre-tension is a specific type of load that requires some special considerations. Prior to defining it, a boundary layer needs to be created. Pre-tension load should be disabled in the second step of the analysis, when actual loading is applied to the model. This can be done by changing the displacement-type pre-tension setting to Fixed. Usage of this type of load is presented in the following videos: [PrePoMax & CalculiX - Pre-tension - Bolt](#) and [PrePoMax \(CalculiX FEA\) - Tutorial 34 - Bolted joint with preload](#).

Items such as constraints, contact pairs, steps, boundary conditions and loads can be activated or deactivated (excluded from the analysis without the need to delete their definitions completely).

Below the menu bar there is a dropdown list, that allows the selection of how the symbols are displayed (None, Model, Step-1, ...).

5.6.6 Defined Field

Defined fields are used to prescribe predefined values to the nodes. The types of predefined fields and their options are listed below:

- Temperature – can be used to specify temperature field for static step
 - Name.
 - Define temperature: By value.
 - * Region.
 - * Temperature.
 - Define temperature: From file
 - * Results file .frd.
 - * Step number.

5.7 Analysis Menu

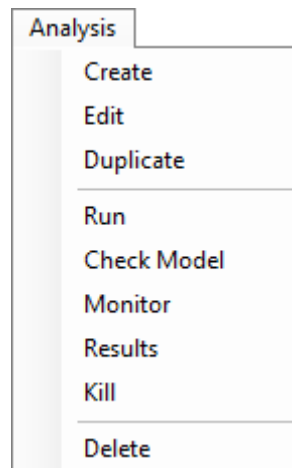


Figure 22: Analysis menu

The **Analysis** menu (Figure 22) contains the following options:

- Create.
- Edit – gives access to name, additional CalculiX arguments and compatibility mode (meant for use with Prool's CalculiX executable which eliminates some issues with steady state dynamics analyses).
- Duplicate.
- Run – submit the analysis.
- Check Model – performs input check, no calculations are performed.
- Monitor – open the window that displays solver messages.
- Results – open results of the completed analysis.
- Kill – stop the analysis before it finishes.
- Delete.

6 Results options

The **Results** tree consists of the following containers:

- Mesh
 - Parts.
 - Node sets.
 - Element sets.
 - Surfaces.
- Results
 - Field outputs.
 - History outputs.

6.1 Results Menu

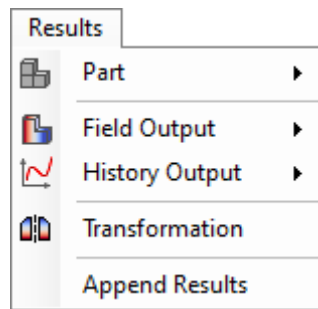



Figure 23: Results menu

The **Results** menu (Figure 23) contains the following options:

- Part
 - Edit.
 - Hide, Show and Show only.
 - Set Transparency.
 - Color Contours off – disables color contours only for selected parts.
 - Color Contours on.
 - Delete.
- Field Output
 - Create.
 - * Limit - creates the following contour plots: RATIO (output divided by specified limit) and SAFETY FACTOR (specified limit divided by output)
 - Properties: Name, Field name, Component name, Limit based on (Parts/Element sets).
 - Limit Values - part-limit table.
 - * Envelope - creates MAX, MIN and AVERAGE contour plots for selected outputs (values are taken from all frames)
 - Properties: Name, Field name, Component name.
 - Edit.
 - Delete – selected field outputs can be deleted to decrease the file size.
- History Output
 - Create
 - * History output from field output: Name, Field name, Component name, Complex, Step id, Increment id, Harmonic (Yes/No), Node coordinates (Off/Undeformed/Deformed), Region
 - Delete.
- Transformation – a mirror or a pattern can be applied to the results for the visualization purposes (use the  button to add the transformation to the Active part of the window to apply it)
 - Symmetry – a symmetry point has to be selected

- * Name.
- * X.
- * Y.
- * Z.
- Pattern
 - * Linear
 - Name.
 - Number of items.
 - Start point.
 - End point.
 - * Circular
 - Name.
 - Number of items.
 - Angle.
 - First axis point.
 - Second axis point.

- Append Results - adds results from another .frd file.

In addition to these options, selecting more than one part in the **Results** tree and right-clicking on this selection enables users to activate the **Merge Parts** option that creates a single results part.



Figure 24: Results toolbar - part 1



Figure 25: Results toolbar - part 2

Selected post-processing options are also available as shortcuts in the Results toolbar (Figure 24 - 25). These options include:

- a. Result – drop-down list with result (.frd) file selection, multiple results can be opened at the same time
- b. Variable (variable used to visualize the deformation): Displacements, Forces, Surface normals, Wear depths, Mesh deformation, Disp&Def&Depth.
- c. Type (deformation scale type): Undeformed, True scale, Automatic, Automatic x 0.25, Automatic x 0.5, Automatic x 2, Automatic x 5, User defined.
- d. Factor – deformation factor for User defined type.
- e. Complex (form of complex results from steady state dynamics): Real, Imaginary, Magnitude, Phase, Angle, Max, Angle at max, Min, Angle at min.
- f. Angle - angle for Angle type

- g. Undeformed.
- h. Deformed.
- i. Deformed with color contours.
- j. Show undeformed wireframe model.
- k. Show undeformed solid model.
- l. Transformation.
- m. First increment.
- n. Previous increment.
- o. Drop-down list of increments.
- p. Next increment.
- q. Last increment.
- r. Animate.

There are several settings for animating the results. They are shown in [Figure 26](#).

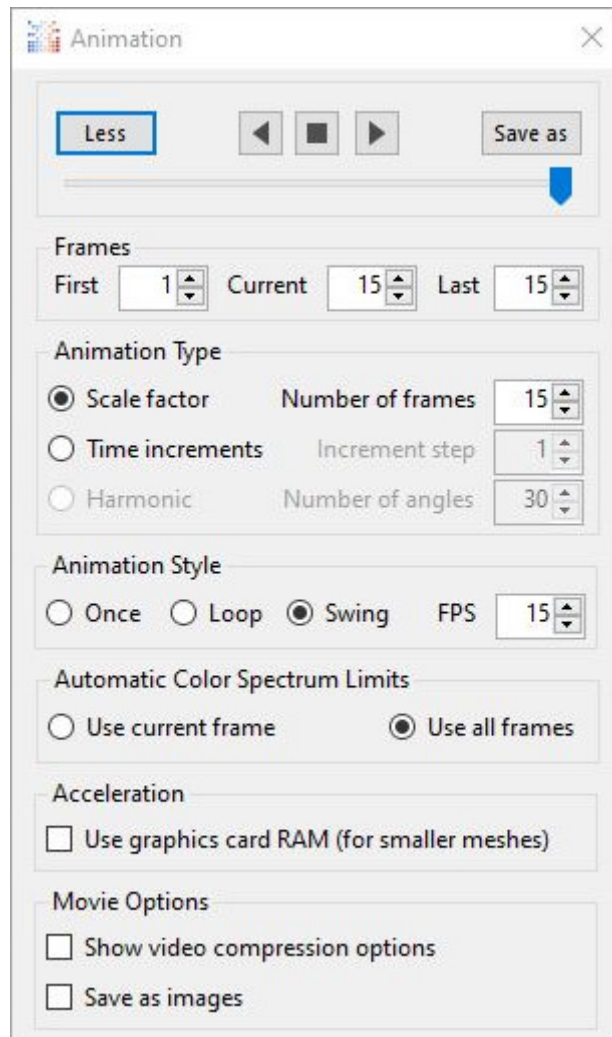


Figure 26: Animation window