PrePoMax 3.0.0 Manual

Jakub Michalski, Ihor Rokach and Matej Borovinšek

November 4, 2026

Contents

1 What is PrePoMax? 3

2 Currently available analysis features 3

3 Workflow, general options 5
3.1 File Menu . . . 9
3.2 Edit Menu . . . 11
3.3 View Menu . . . 12
3.4 Tools Menu . . . 14
3.5 Help Menu . . . 18

4 Geometry and mesh options 18
4.1 Geometry Menu . . . 19
4.2 Mesh Menu . . . 20

5 FE Model options 25
5.1 Model Menu . . . 25
5.2 Property Menu . . . 30
5.2.1 Libraries . . . 31
5.2.2 Library materials section . . . 31
5.2.3 FE model materials section . . . 32
5.2.4 Using the material library . . . 32
5.2.5 Exchanging material data between users . . . 32
5.3 Interaction Menu . . . 35
5.4 Distribution Menu . . . 41
5.5 Amplitude Menu . . . 42
5.6 Initial Condition Menu . . . 42
5.7 Step Menu . . . 44
5.7.1 Step . . . 45
5.7.2 History Output . . . 51
5.7.3 Field Output . . . 52
5.7.4 Boundary condition . . . 53
5.7.5 Load . . . 55
5.7.6 Defined Field . . . 61
5.8 Analysis Menu . . . 61

6 Results options 62
6.1 Results Menu . . . 62

1 What is PrePoMax?

PrePoMax is an open-source pre- and postprocessor for also open-source Finite Element Analysis solver CalculiX CrunchiX. The syntax of CalculiX is mostly common with Abaqus and PrePoMax may prepare input decks for Abaqus (and even submit its jobs and open its .odb results or convert them to CalculiX .frd format) as well. PrePoMax partially supports the open-source FEMaster solver too. PrePoMax is being developed by Dr. Matej Borovinšek from University of Maribor, Slovenia. The current stable version is 3.0.0. Source code of the development version of the program is available on GitHub. The former GitLab repository is read-only and development moved to GitHub. 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/`. PrePoMax is designed for Windows, running it on other systems including Linux requires virtual machines, emulators or similar software such as Wine. Installation is not necessary - the zip file just needs to be unpacked in a directory not requiring admin rights (otherwise, some features may not work properly), revealing the executable. User forum is available here: `https://prepomax.discourse.group/`. There is also a Discord server.

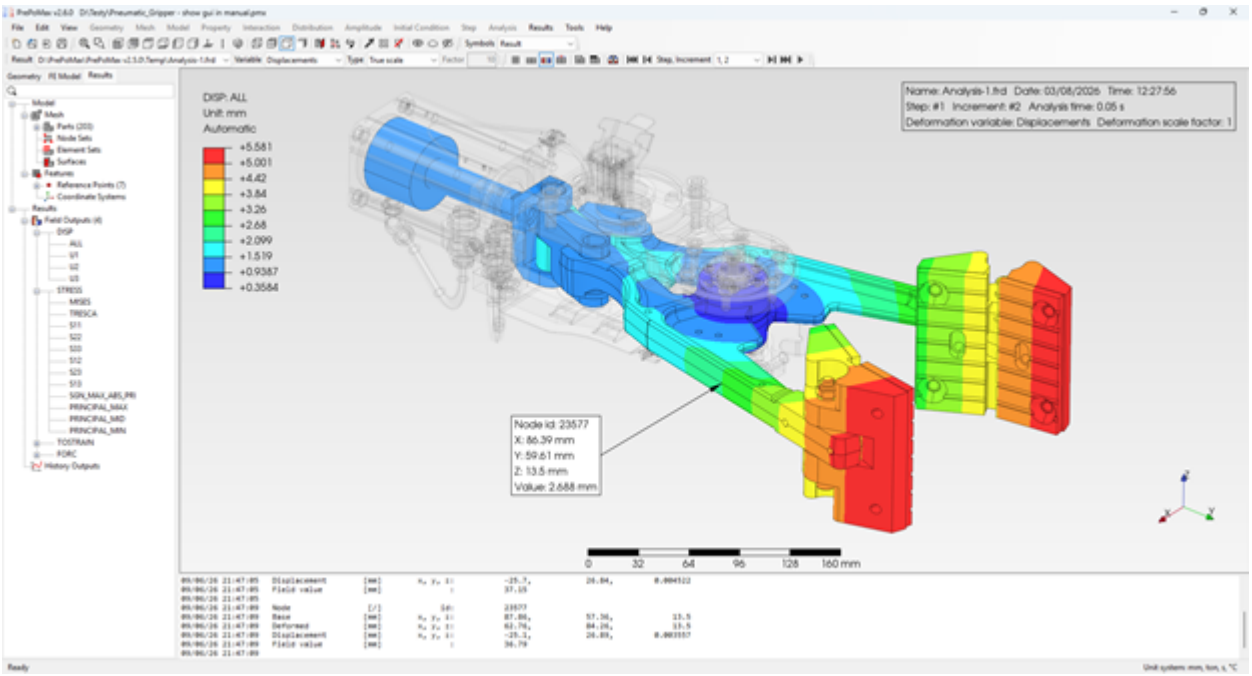


Figure 1: PrePoMax user interface

2 Currently available analysis features

- Analysis step types:
  - Static
  - Natural frequency extraction (real and complex)
  - 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, hexahedrons, pyramids (in the form of collapsed wedges/hexahedrons) and wedges/pentahedrons (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)
  - One-node mass elements (available as Point Mass and Distributed Mass sections)
  - One-node spring elements (available as Point Spring and Surface Spring constraints)
- Material models (with optional temperature dependency):
  - Linear elasticity (isotropic and orthotropic)
  - Plasticity
  - Thermal
  - Slip wear
- Constraints:
  - Point spring
  - Surface support (spring or compression-only)
  - Rigid body
  - Tie and cyclic symmetry
  - Joints
  - Kinematic and distributing coupling
- Interactions:
  - Contact
    - Friction
    - Gap conductance
- Initial conditions and defined fields
  - Temperature (initial condition and defined field)
  - Velocity (initial condition)
  - Angular velocity (initial condition)
- Boundary conditions:
  - Fixed
  - Displacement/Rotation

  - Submodel
  - Temperature

- Loads:
  - Concentrated force
  - Moment
  - 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

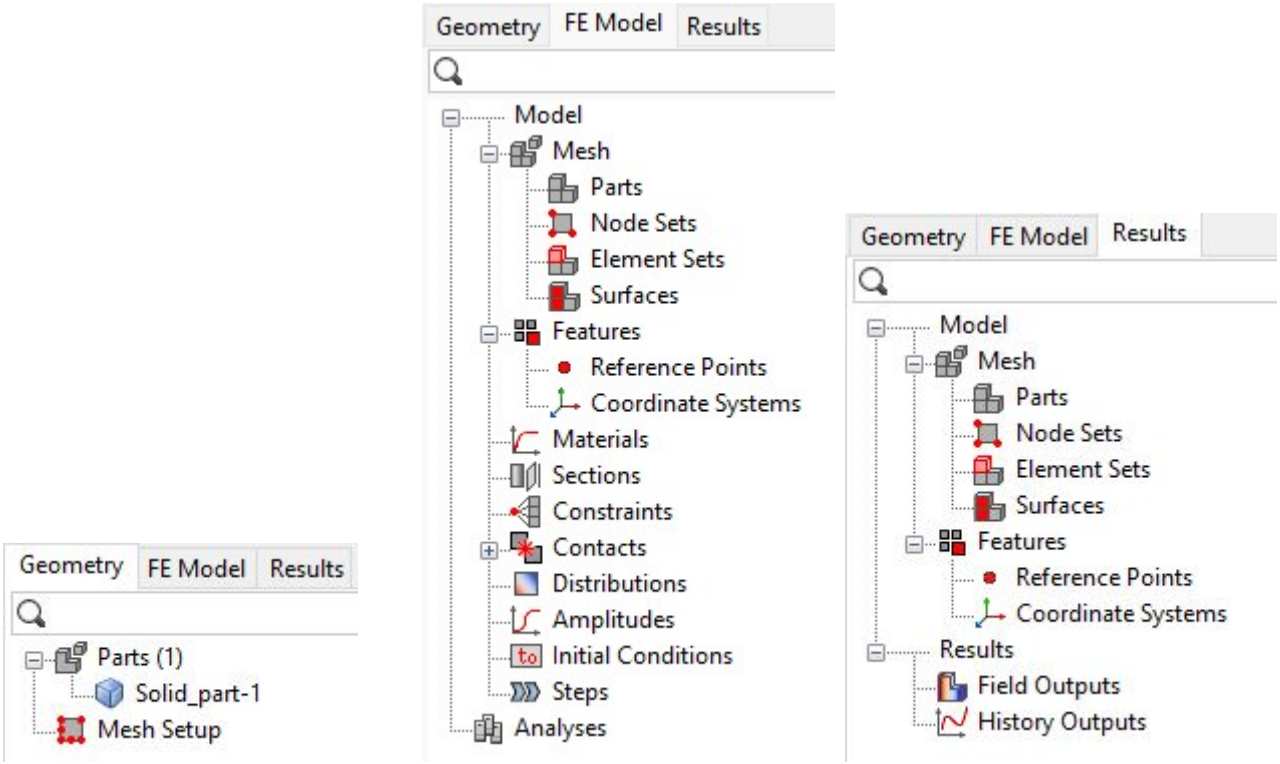
3 Workflow, general options

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. Their normals must point in the positive Z direction (they can be flipped using Geometry > CAD Part > Flip Face Normal). Moreover, for axisymmetric analyses, faces representing parts must lie to the right of the Y axis. Yellow triangle with white exclamation mark next to the part's name indicates potential issues with geometry that may require further investigation and repair (in CAD software before importing to PrePoMax). Broken edges are highlighted with a violet color. Another exclamation mark may appear if a part has an edge shorter than 1/1000 of the part?s bounding box diagonal, or a face narrower than that (a thin strip or sliver). When you select the part, its small edges and their vertices are highlighted. Remove them with the Repair tool (removes small faces and edges, fuses faces, joins edges), using Preview to see what will be removed. Other relevant tools are Fuse Faces and Defeature Solid.



(a) *Geometry* tab of the model tree

(b) *FE Model* tab of the model tree

(c) *Results* tab of the model tree

Figure 2: Three PrePoMax tabs

3. For CAD and STL geometry prepare mesh setup (Mesh › Mesh Setup Item › Create) and generate a mesh (Mesh › Create Mesh). The mesh can also be generated with no setup items - it will use default settings.

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), distributions (Distribution › 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
- smooth zoom: Ctrl + mouse scroll or Ctrl + middle mouse button;
- rotate model: middle mouse button or keyboard arrows;
- pan (move model): Shift + middle mouse button.

PrePoMax also supports touchpad/trackpad navigation. By default, two finger drag is used for rotation, pinching fingers together or spreading them apart controls zoom while holding Shift replaces rotation with panning for two finger drag. Optionally, the behavior can be inverted (panning without holding Shift and rotating with Shift held).

Pressing middle mouse button anywhere on the part moves that location to the center of the screen and sets it as rotation center. Double-clicking on the background activates Zoom to Fit. Clicking on any axis of the coordinate system widget in the corner enters the normal or isometric view (depending on whether the arrow or arrowhead is clicked) in the direction of this axis. Clicking on the origin point applies isometric view too.

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.

Equations are supported in several places across PrePoMax (such as parameters, input fields, distributions and equation-based custom outputs). They have to be preceded by the equal sign. They use the open-source Ncalc library with the available functions described here. To display a list with the functions, use the Ctrl+Space. Autocompletion is also available.

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.

Usually, it is possible to open old files in new PrePoMax versions, but since it evolves very quickly, despite all the efforts, backward compatibility cannot be guaranteed. This is also why older versions are still provided for download.

The *PrePoMax.com* file makes it possible to easily use PrePoMax from the command window by just calling its name (and directory if the working directory is not the same as PrePoMax directory). The following command line options are currently supported:

- **-f, –file** - name of a file to be opened or imported
- **-g, –showGui** - *Yes* (default) or *No* - whether to show the graphical interface, the latter option can only be used for regeneration
- **-o, –overwrite** - *Yes* or *No* (default) - whether to overwrite the .pmx file after regeneration

- **-p, –parameters** - overwrite the .pmx parameters. To overwrite parameter *a* use *[a=1.2]*. To overwrite parameters *a* and *b* use quotations and semicolons *["a=1.2; b=31.4"]*. To overwrite parameters by importing a parameters file, specify the .pmp file name. Can only be used for regeneration.
- **–parametersExport** - the .pmp file name to be used when exporting parameters from the .pmx file defined by the **-r** switch.
- **-r, –regenerate** - name of a .pmx to be used for regeneration (a form of automation - rerunning all commands in the history of a .pmx file). A work directory **-w** can be defined for regeneration. If no work directory is defined the current directory is used. All files needed during regeneration (geometry, mesh) must be located in the work directory. They can be swapped with the new ones (given the same names) to change the geometry used for the simulation. Use **–r0**, **–r1**, **–r2** and **–r3** to regenerate all history commands, only pre-processing commands, only analysis commands and only post-processing commands, respectively.
- **-u, –unitSystem** - unit system to be used when importing (works also with .frd files) [M_KG_S_C | MM_TON_S_C | M_TON_S_C | IN_LB_S_F | UNIT_LESS].
- **-w, –workDirectory** - directory path to be used as work directory
- **-x, –exitAfterRegeneration** - *Yes* (default) or *No* - whether to exit PrePoMax after regeneration
- **–help** - display the help screen with a list of commands
- **–version** - display version information

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.

Parts and materials, as well as step-dependent features can be deactivated (right-click and choose Deactivate) - excluded from the analysis.

All tree items can be renamed by pressing F2 or clicking the left mouse button twice slowly.

Tabular data across PrePoMax (e.g. amplitude or plasticity definitions) can be copied from a file and pasted directly to a table in PrePoMax if the separator is space, comma or tab. It is also possible to plot the tabular data by selecting a range of cells, right-clicking in the table and selecting the *Plot* option or using the Ctrl+P shortcut.

All model tree items apart from steps can be grouped using the right-click context menu. Dragging in and out of groups as well as drag and drop to change the item order is also supported.

Widgets such as color legend and status block have right-click context menus to access their settings.

PrePoMax has 3 side panels:

- Python Script Editor
  - divided into script editor (top) and Python console (bottom)
  - there are two ways of using the Python panel:
    - Interactive - all PrePoMax Python commands are directly executed using the built-in history commands. The script is not recorded in history, but the commands that started from the script are. If the model is rebuilt using Regenerate, the commands are run from the command history.
    - Workflow execution (Record script into command history), the script is recorded as a history command. During regeneration, the script is run again.

- XY Plot
  - used only for visualization of history outputs, no new/changed history outputs can be created using it
  - up to 4 separate plots are supported
  - a history output can be added to the active (selected) plot by right-clicking it in the tree (Add to Plot, Add to New Plot)
  - the plot and curve settings are available by double-click
- Advisor - interactive workflow assistant

PrePoMax supports MCP (Model Context Protocol) and thus can be driven from an AI agent. The instructions for its use are included in the GitHub repository. The documentation for Python API can be found in the Python_API_Overview.html file provided with the PrePoMax builds and accessible using the Help button in the Python Script Editor.

3.1 File Menu

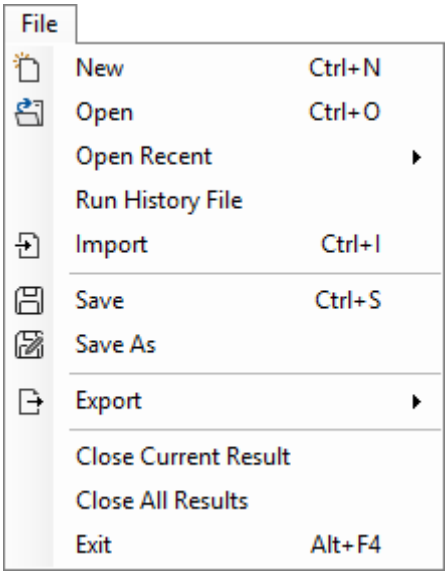


Figure 3: File menu

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, lbf · s^2/in, s, °F – imperial units.
- Open (Ctrl + O) – open a previously created model (.pmx), history (.pmh), CalculiX results (.frd), Abaqus results (.frd, Abaqus executable has to be selected in the Settings) or OpenFOAM files (.foam)

- Open Recent – open one of the recently used models or clear the list of the recent files.
- Run History File – run PrePoMax command history file.
- 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,
  - Stereolitography: .stl,
  - Universal: .unv,
  - Netgen: .vol,
  - Abaqus/CalculiX input files: .inp,
  - Mmg mesh: .mesh,
  - Parameters files: .pmp.
- 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:
  - Geometry: STEP parts/assembly (.stp), BREP parts/assembly (*brep), STL (.stl), Mmg (.mesh),
  - Mesh: CalculiX (.inp), Abaqus (.inp), Gmsh (.msh), Mmg (.mesh), STL (.stl),
  - Model: CalculiX (.inp), Abaqus (.inp),
  - Results: Deformed parts (.inp or .stl, respects the current deformation scale factor, .inp includes solid elements if output is set to 3D for analysis with shell elements), Field output (.csv), History output (.csv),
  - 3D Manufacturing Format (.3mf),
  - Python session journal (.py).
- 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 (Alt + F4) – close PrePoMax (the user will be asked if the model should be saved first).

3.2 Edit Menu

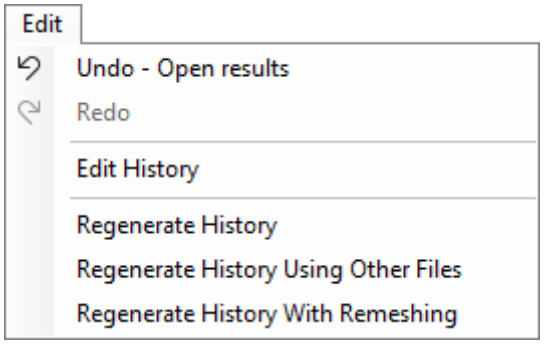


Figure 4: Edit menu

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

- Undo – undo the last action.
- Redo – redo the action that was undone previously.
- Edit History – open history editor, making it possible to view the history of performed operations and delete the selected command entries by choosing the first column of the row and pressing the Delete key. The entire history can be deleted using the Clear All button or reorganized. Drag and drop is also supported for commands. Changing the history will change the Regenerate functionality, so it should be used cautiously. Clearing a very long history might help with the responsiveness of PrePoMax since the history is saved to a file after each new command is added to it (i.e. after each feature is added to the model). History can also be opened from file and saved to file.
- Regenerate History – regenerate all or only the FE model (only model commands - commands for analysis and postprocessing are not regenerated).
- Regenerate History Using Other Files – regenerate all or only the FE model after swapping the model's geometry with a similar one from another file so that it is not necessary to redefine the model.
- Regenerate History With Remeshing – regenerate all or only the FE model after remeshing the model while keeping all the node sets, element sets and surface data defined using mesh-based selections - the model can be remeshed without the need to redefine the geometry-based selections.

3.3 View Menu

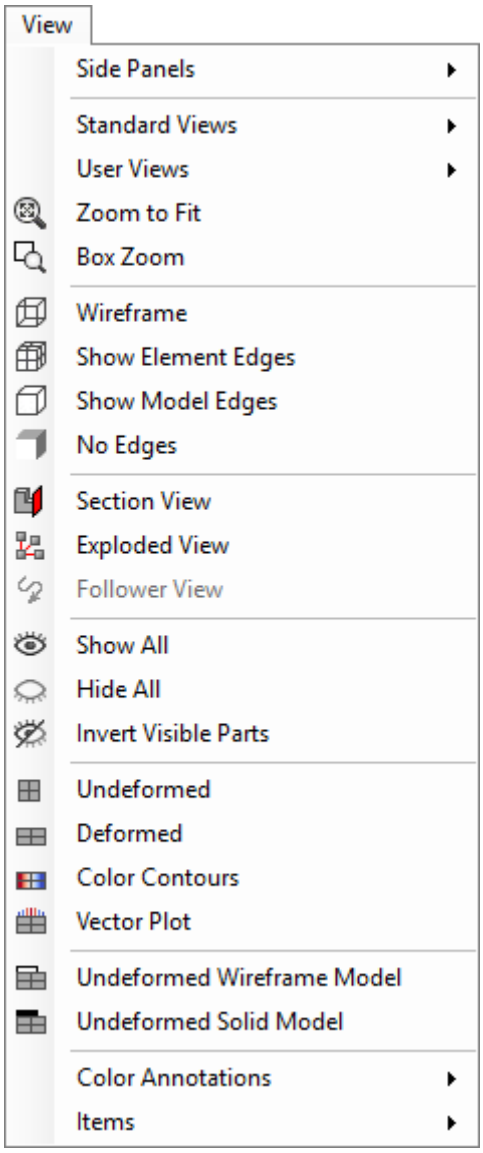


Figure 5: View menu

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

- Side Panels: Python Script Editor, XY Plot, Advisor.
- Standard Views – apply one of the predefined views:
  - Front or Back,
  - Top or Bottom,
  - Left or Right,
  - Normal,
  - Vertical,
  - Isometric.
- User Views – opens a window where views can be created (from the current orientation of the camera), applied, updated, moved up and down the list or removed from it.

- Zoom to Fit – zoom the model in such a way that it fits in the window.
- Box Zoom – zoom to rectangular area defined bv dragging the cursor (keyboard shortcut: hold z and drag the cursor).
- 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). The additional settings are: Lighten section colors, Use single section color, Hatch pattern and Element-wise section (to choose if finite elements on the section plane should be cut).
- Exploded View – show assembly parts in an exploded view - 3 methods available:
  - Disassembly: Contact tolerance, Exploded view direction, Magnification, Scale factor, Step by step,
  - Assembly center: Exploded view direction, Magnification, Scale factor,
  - Center point: X/Y/Z coordinates, Exploded view direction, Magnification, Scale factor.
- Follower View – accessible only in the Results tab, opens a window where a follower view can be created. Two types are available: Point (follows the selected node) and Plane (follows a plane defined by a center node and two direction nodes). When enabled, the camera will always show the selected location even when it moves due to deformation in some frames. Follower view can be disabled after reopening its window.
- 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.
- Color Contours – show deformed model with contour plot in results.
- Vector Plot – creates a vector plot of vector fields and principal tensor values, also working with animations.
- Undeformed Wireframe Model – show deformed model with contour plot and wireframe representation of undeformed model in results.
- Undeformed Solid Model – 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, mesh types, parts, materials, sections, section thicknesses, all symbols, reference points, constraints, contact pairs, initial conditions, BCs, loads, defined fields.
- Items – display vertex/edge/surface/part IDs, part names, node/element IDs.



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

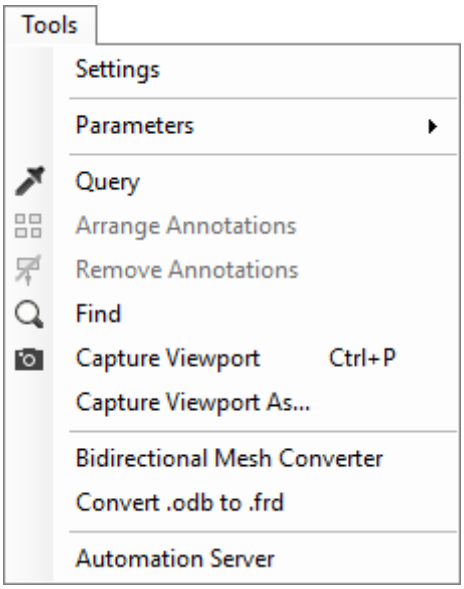


Figure 7: Tools menu

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.
    - Compress .pmx files: No compression, Fastest, Optimal.
    - Default unit system.
    - Import mesh → Edge angle.
    - Open results → Run history postprocessing (Yes/No/Ask).
    - Property editor → Show all properties.
    - Regenerate older files → Split periodic faces, Number of resulting faces.
  - Number formatting
    - Number format: General, Scientific,
    - Number of significant digits.
  - Graphics
    - Background: Type, Top color, Bottom color.
    - Geometry: Linear CAD deflection type (Relative/Absolute), Linear CAD deflection, Angular CAD deflection.

    - Lighting: Ambient component, Diffuse component.
    - Smoothing: Point smoothing, Line smoothing.
    - Widgets: Coordinate system visibility, Scale widget visibility, Widget highlighting.
  - Navigation
    - Common: Rotate sensitivity, Zoom sensitivity, Zoom direction (Normal/Reversed), Smooth zoom (On/Off).
    - Trackpad: Two finger gestures (Automatic/Off/On), Two finger drag (Rotate/Pan), Pan sensitivity, Left - right direction (Normal/Reversed), Up - down direction (Normal/Reversed).
  - Default Colors
    - Face orientation: Front face color, Back face color.
    - Mesh types: Transfinite/Sweep/Revolve/Tetrahedral/Free/Other color.
    - General: Color palette (Default, Expevo, Minou, Reading, Alger, Heatmap0, Pantone23, Lincoln, Seafarer, Paloma, Ploen, Gaby, Custom), Saturation, Colors 1-10.
  - Annotations
    - Design: Background type, Draw a border rectangle.
    - Edge/Surface annotation: Show edge/surface id, Show edge/surface type, Show edge/surface size, Show edge/surface topology, Show maximum value, Show minimum value, Show sum value, Show average value.
    - Node annotation: Show node id, Show coordinates.
    - Part annotation: Show part name, Show part id, Show part type, Show volume/area, Show mass, Show number of elements, Show number of nodes.
  - Meshing – default settings for meshing
    - Mesh size: Mesh size definition (Absolute/Relative), Max element factor, Min element factor, Grading, Elements per edge, Elements per curvature.
    - Mesh type: Second order, Midside nodes on geometry.
    - Mesh operations: Split compound mesh, Merge compound parts, Merge cells.
    - Parallelization: Numbers of threads.
    - Time limit: Mesher time limit [min].
  - Pre-processing
    - Color bar: Background type, Draw a border rectangle.
    - Selection: Default geometry selection mode (Selection by location/ID), Primary/secondary/mouse highlight color, Splitting tool color.
    - Symbols: Constraint/Initial condition/Boundary condition/Load/Defined field color, Symbol size, Node symbol size, Draw symbol edges.
  - Solver
    - General: Work directory, Use .pmx folder as work directory, Environment variables.
    - Parallelization: Number of processors.
    - CalculiX: Executable, Default matrix solver, Convert pyramid elements to (Collapsed wedges/Collapsed hexahedrons), Binary results file (Yes/No).
    - Abaqus: Executable.
    - FEMaster: Executable.
  - Post-processing
    - Undeformed model

            - Draw undeformed model: None/Wireframe body/Solid body,
            - Undeformed model color.
        * Vector plot
            - Vector plot scale factor.
            - Vector length scale factor.
            - Vector plot density.
        * Limit values
            - Show min value and location: Yes/No,
            - Show max value and location: Yes/No.
        * History output
            - Max history output entries.
            - Max curves per plot.
    - Legend
        * Color spectrum settings
            - Color spectrum type: Cool-warm, Rainbow, Rainbow desaturated, Turbo, Jet, Warm, Cool, Cividis, Viridis, Plasma, Black body, Inferno, KindImann, Grayscale,
            - Color mode: Discrete/Smooth,
            - Brightness,
            - Reverse colors: Yes/No,
            - Number of discrete colors – in the range of 2-24.
        * Color spectrum values
            - Min/Max limit type: Automatic/Manual – for Manual: Min/Max value, Min/Max color, Show min/max color in legend
        * Design
            - Background type: None/White,
            - Draw a border rectangle: Yes/No.
    - Status Block
        * Design
            - Status block visibility: On/Off,
            - Background type: None/White,
            - Draw a border rectangle: On/Off.
    - Automation
        * Server
            - Start with application: Yes/No,
            - Read only: Yes/No,
            - Echo to the Python panel: Yes/No,
            - Request timeout [s].
        * Connection
            - Port,
            - Address,
            - Access token.

- Parameters
    - Edit

        * Name
        * Value/Equation
        * Evaluates to
        * Description
    - Export – exports parameters to .pmp file,
    - Import – imports parameters from .pmp file.

- 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, type and 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, type, number of edges/vertices, number of elements/nodes and 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, area/volume, mass, number of facets/vertices, number of elements/nodes).
    - Assembly – check the number of parts, mass, number of facets/vertices and elements/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.

- Arrange Annotations – moves all annotations to their original positions.

- Remove Annotations – deletes all annotations.

- Find
    - Item type: Vertex/Node, Facet/Element, Edge, Surface, Part.
    - Data: Add annotation, Item id.

- Capture Viewport – shortcut Ctrl+P, takes a screenshot of the viewport and places it in the clipboard.

- Capture Viewport As... – takes a screenshot of the viewport and optionally places it in the clipboard and saves to a file (additional settings: Transparent background and Magnification).

- Bidirectional Mesh Converter – allows conversion between Nastran and CalculiX mesh format (supplied by Capgemini).

- Convert .odb to .frd – converts the selected Abaqus .odb results file to CalculiX .frd file.

- Automation Server – can generate an access token for an AI assistant and start an automation server.

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.

Legend limits can be changed also directly on the legend by clicking on the min/max labels.

3.5 Help Menu

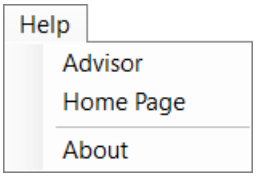


Figure 8: Help menu

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 two main containers – *Parts* and *Mesh Setup*. 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, provides information about its type, mesh, geometry (volume/area) and center of mass, 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 Color/Reset Color/Set Transparency – setting color and transparency of the part.
- Delete – deletes the selected part.

4.1 Geometry Menu

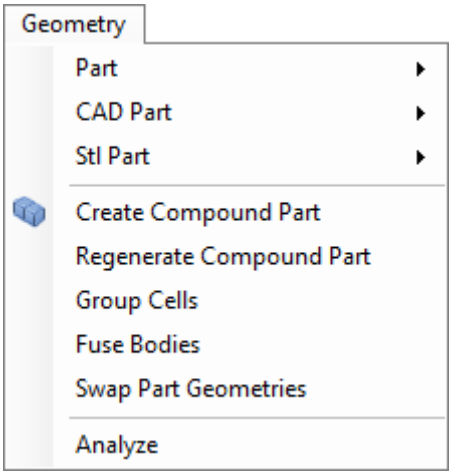


Figure 9: Geometry menu

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

- Part – general part geometry options:
  - Edit: Name, Part type, Geometry, Mesh, Volume, Center of mass, Appearance,
  - Transform,
    - Scale – scales or creates a scaled copy of the part based on the specified center point coordinates and scale factors in X, Y and Z axes.
  - Copy Geometry to Results,
  - Hide/Show/Show Only,
  - Set Color/Reset Color/Set Transparency,
  - Delete.
- CAD Part - CAD part geometry options:
  - Convert Shell to Solid – converts closed shell part to solid part,
  - Convert Solid to Shell – converts solid part to closed shell part,
  - Flip Face Normal – flips the shell face orientation,
  - Split Face – divides a shell surface or a solid's face (also includes optional preview and mesh type coloring as well as undo button) using one of the following methods: Line through Two Points, Plane by Three Points, Plane by Point and Normal, Extend Face, Extrude Edge,
  - Split Periodic Faces – splits periodic faces of the selected part into a specified number of faces,
  - Fuse Faces – inverse of Split Face, fuses CAD part faces that lie on one surface,
  - Offset Face – applies a given offset to a face, creating a new shell part,
  - Delete Faces – removes selected faces,

  - Split Solid – divides a solid into cells (subvolumes) to allow hex meshing (also includes optional preview and mesh type coloring as well as undo button), offers the following methods: Plane by Three Points, Plane by Point and Normal, Extend Face, Extrude Edge,
  - Defeature Solid – removes fillets, chamfers, holes, engravings and other features by replacing the selected faces. With each use of the tool, the old geometry is replaced by the new one,
  - Repair – offers a tolerance setting and the following options: Sew faces, Fix shape, Remove small faces, Remove small edges, Fuse faces, Join edges.

- 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 Face Normals – flips the orientation of the surfaces,
  - Smooth Part – applies smoothing to STL part.
  - Delete Part Faces – deletes the selected faces.

- Create Compound Part – merges two or more parts into one part (body). For compounding to work, the geometry must fit perfectly (for example, planar faces of parts in contact with a cylindrical part may require merging the geometry in CAD software, elongating the parts to penetrate the cylinder wall or just using tie constraints instead). Moreover, to mesh a compound properly, there should be no edges splitting the surface only partially (instead of dividing it into two faces).

- Regenerate Compound Part – regenerates a compound part.

- Group Cells – does not change the geometry, only collects the selected cells (subvolumes obtained by splitting solids) under one subpart, which becomes one mesh part.

- Fuse Bodies – changes the geometry, the selected cells (subvolumes) become one solid, and the faces between them are removed. Can be used to undo an unwanted cut or to remove an interior face that affects the mesh. Fusing all cells of a part gives back a plain 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

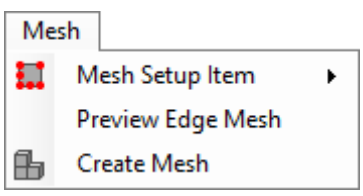


Figure 10: Mesh menu

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

- Mesh Setup Item: Create, Edit, Duplicate, Delete – settings for the mesher
- Preview Edge Mesh
- Create Mesh

The following mesh setup items can be created:

- Meshing parameters – global meshing settings:
  - Data
    - Name,
    - Mesh settings: Basic, Advanced,
    - Region.
  - 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 type
    - Second order: Yes/No,
    - Midside nodes on geometry – project the midside nodes on the geometry, options: Yes/No.
  - Mesh operations
    - Split compound mesh: Yes/No (splits compound part mesh to unconnected part meshes),
    - Merge compound parts: Yes/No (merges compound part mesh to a single mesh part),
    - Merge cells: Yes/No (merges the cell/subvolume meshes of each part cut by a splitting tool into a single mesh part).
- Local Mesh Size – settings for local mesh refinement:
  - Name,
  - Defined by: Element size/Number of elements,
  - Region,
  - Element size/Number of elements.
- Shell Mmg:
  - Name,
  - Region,
  - Geometry approximation: Hausdorff definition (Relative/Absolute), Relative Hausdorff,
  - Mesh operations: Keep model edges (Yes/No).
- Shell Gmsh:
  - Name,

  - Region,
  - 2D meshing algorithm: MeshAdapt, Automatic, Initial mesh only, Delaunay, Frontal-Delaunay, Frontal-Delaunay for quads, Packing of parallelograms, Quasi-structured quad,
  - Recombine algorithm: None, Simple, Blossom, Simple full-quad, Blossom full-quad,
  - Transfinite 3-sided faces: Polar/Mapped/No – Polar collapses one corner with triangles there and arcs around it while Mapped meshes three blocks of quads around one inner node,
  - Transfinite 4-sided faces: Yes/No,
  - Transfinite threshold angle – faces with a corner angle larger than this angle or smaller than 180 minus this angle are meshed without the transfinite constraint,
  - Mesh optimization:
    * First order elements: None, Laplace 2D, Relocate 2D, Quasi-structured quad.

- Shell QuadWild:
  - Name,
  - Region,
  - Triangulation: Mesher (Netgen/Mmg/Gmsh), Preview (Yes/No),
  - Transfinite 3-sided faces: Polar/Mapped/No – Polar collapses one corner with triangles there and arcs around it while Mapped meshes three blocks of quads around one inner node,
  - Transfinite 4-sided faces: Yes/No,
  - Transfinite threshold angle – faces with a corner angle larger than this angle or smaller than 180 minus this angle are meshed without the transfinite constraint,
  - Flow profile: Aligned/No alignment,
  - Alpha – controls the final quadrangulation trade-off between regularity and isometry,
  - Smooth: Yes/No.

- Thicken Shell Mesh – can be used to obtain solid meshes from surface geometries/shell meshes by extrusion along local shell normals:
  - Name,
  - Region,
  - Thickness – thickness of the resulting solid mesh,
  - Number of layers – number of solid element layers,
  - Offset – controls the shift (in length units) of the extruded thickness relative to the original surface (0 means extrusion symmetrically in both positive and negative normal direction while positive and negative values mean shift in the positive and negative normal direction, respectively),
  - Keep model edges: Yes/No.

- Netgen Mesh:
  - Name,
  - Region,
  - Optimize steps 2D,

  - Optimize steps 3D,
  - Quad-dominated mesh: Yes/No,
  - Transfinite 3-sided faces: Polar/Mapped/No – Polar collapses one corner with triangles there and arcs around it while Mapped meshes three blocks of quads around one inner node,
  - Transfinite 4-sided faces: Yes/No.

- Tetrahedral Gmsh:
  - Name,
  - Region,
  - 2D meshing algorithm: MeshAdapt, Automatic, Initial mesh only, Delaunay, Frontal-Delaunay, Frontal-Delaunay for quads, Packing of parallelograms, Quasi-structured quad,
  - 3D meshing algorithm: Delaunay, Initial mesh only, Frontal, MMG3D, R-tree, HXT,
  - Transfinite 3-sided faces: Polar/Mapped/No – Polar collapses one corner with triangles there and arcs around it while Mapped meshes three blocks of quads around one inner node,
  - Transfinite 4-sided faces: Yes/No,
  - Transfinite threshold angle – faces with a corner angle larger than this angle or smaller than 180 minus this angle are meshed without the transfinite constraint,
  - Experimental For Pyramids:
    * Recombine algorithm: None, Simple, Blossom, Simple full-quad, Blossom full-quad.
  - Mesh optimization:
    * First order elements: None, Gmsh, Netgen, Relocate 3D,
    * Second order elements: None, High order, High order elastic, High order fast curving.

- Transfinite Mesh:
  - Name,
  - Region,
  - Transfinite 3-sided faces: Polar/Mapped/No – Polar collapses one corner with triangles there and arcs around it while Mapped meshes three blocks of quads around one inner node,
  - Transfinite threshold angle – faces with a corner angle larger than this angle or smaller than 180 minus this angle are meshed without the transfinite constraint,
  - Recombine: Yes/No,
  - Experimental For Pyramids:
    * Allow pyramid elements: Yes/No.

- Sweep Mesh – sweeps the surface meshes (can be previewed) created on automatically or manually selected base faces in automatically detected directions (shown by pink arrows) along the side surfaces up to the target surfaces:
  - Name,
  - Source surfaces: Selected surfaces/Automatic,
  - Symmetry: Off/Automatic,

  - Region,
  - 2D meshing algorithm: MeshAdapt, Automatic, Initial mesh only, Delaunay, Frontal-Delaunay, Frontal-Delaunay for quads, Packing of parallelograms, Quasi-structured quad, Quad-Wild,
  - Recombine algorithm: None, Simple, Blossom, Simple full-quad, Blossom full-quad,
  - Recombine min quality,
  - Transfinite 3-sided faces: Polar/Mapped/No – Polar collapses one corner with triangles there and arcs around it while Mapped meshes three blocks of quads around one inner node,
  - Transfinite 4-sided faces: Yes/No,
  - Transfinite threshold angle – faces with a corner angle larger than this angle or smaller than 180 minus this angle are meshed without the transfinite constraint,
  - Mesh optimization:
    * First order elements: None, Laplace 2D, Relocate 2D, Quasi-structured quad.
  - Mesh smoothing:
    * Global smooth steps.

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 Local Mesh Size option can be used for local refinement/mesh controls. A surface, edge or vertex can be selected, and local mesh size can be specified or number of elements can be entered. The latter mode works correctly only for Gmsh and only if structured-quad algorithms are not used. There is no direct way to enforce a certain number of elements for the Netgen mesher. The mesh refinement can be previewed before generating the mesh.

Selecting a part highlights its mesh setup items and vice versa.

Hexahedral meshes can be created using two Gmsh-based mesh types: Transfinite and Sweep. The following rules apply:

- All these methods cannot be used on STL parts.
- Hexahedral meshes are generated by default if recombination (of triangles into quads) is enabled.The quasi-structured quad algorithm doesn't need it, though.
- The transfinite algorithm for 3-sided and 4-sided faces can be used to obtain more structured meshes.
- **Transfinite** meshing requires volumes with 5 or 6 faces. Those faces should have 3 or 4 edges each.
- For **Sweep** meshing the numbers of elements on connected edges are matched automatically, so the mesh stays conformal between the neighboring cells.

If no meshing algorithm is applied, PrePoMax checks the geometry of every part and meshes it with the most structured mesh it supports:

- a solid with 5 or 6 three- or four-sided faces (a box or a prism) gets a transfinite hexahedral mesh,
- a solid that can be swept from one set of faces to the opposite set gets a swept hexahedral mesh,

- a solid that can be turned around an axis gets a revolved mesh,
- all other solids are meshed with tetrahedra by Netgen.
- CAD shell parts are meshed quad-dominant by Gmsh, with structured grids on 3- and 4-sided faces.
- open STL shells are meshed by Netgen.

The usual hex meshing workflow is: enable the Mesh Types coloring, look at the parts colored as tetrahedral, check the reason for that in the Output window, split using appropriate modes of the Split Solid tool, and repeat. The Split Solid dialog has a button that toggles the Mesh Types colors while you split. Note that splitting a plain part replaces it with a new compound part, so analysis features that referred to the old part must be redefined. Split the geometry before defining the analysis.

The following YouTube tutorials cover original Gmsh meshing in PrePoMax:

- PrePoMax & CalculiX - Introduction to advanced meshing
- PrePoMax & CalculiX - Transfinite shell meshes
- PrePoMax & CalculiX - Transfinite volume meshes
- PrePoMax & CalculiX - Extruded meshes
- PrePoMax & CalculiX - Revolved meshes

5 FE Model options

Switching to FE Model tab activates the following menus: Model, Property, Interaction, Distribution, 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

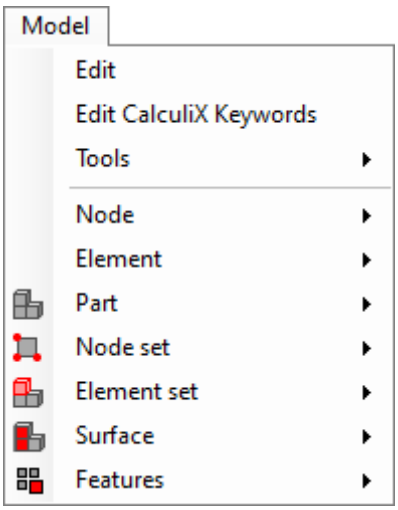


Figure 11: Model menu

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 const – value of the Stefan-Boltzmann constant used for radiation,
  - Gravitational const – value of the Newton's gravitational constant.

- Edit CalculiX Keywords – a keyword editor that allows adding CalculiX keywords which are currently unsupported by PrePoMaxand modifying the ones generated by the preprocessor. Keywords can be deactivated. By default, folding is used to hide long data lines, but it can be disabled by unchecking the Use hiding for faster operation button. There are buttons to add keywords, move them up and down, undo edits and remove added keywords. Modifications to existing keywords are output-level overrides - later GUI model changes may reset the affected edits.

- Tools

  - Find Model Edges By Angle – searches for the feature edges based on the angle in the imported meshes (regenerates the mesh surfaces to merge them based on the selected angle),
  - Create Boundary Layer – creates a prismatic boundary layer consisting of wedge elements on the selected surface for analyses with a bolt pre-tension load (needed for tetrahedral meshes because, as per the CalculiX manual, the internal surface where the pre-tension load is applied shouldn't include any edges or vertices of elements that don't have a face on this internal surface), the thickness of the layer can be specified, preview option is also available, boundary layer can be removed by undoing its creation or remeshing the part
  - 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),
  - Thicken Shell Mesh – access to the Thicken Shell Mesh feature, available also as a mesh setup item (this one can be applied to imported meshes too),

  - Split Part Mesh Using Surface – can split the selected base part region (volume mesh) into a coincident two-region mesh using a splitter surface (shell mesh) - useful in various scenarios where the CAD geometry is absent or when a fast mesh division is needed,
  - 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.
  - Merge Coincident Nodes
    - Tolerance,
    - Nodes to keep: Smaller ID/Larger ID.

- Element
  - Renumber All
    - Start element id.
  - Element Quality – uses Gmsh element quality metrics, can create a set based on the specified criteria
    - Quality metric: minDetJac (minimal Jacobian determinant), maxDetJac (maximal Jacobian determinant), minSJ (minimal scaled Jacobian), minSICN (minimal signed inverted condition number), minSIGE (signed inverted gradient error), gamma (ratio of the inscribed to the circumscribed sphere radius), innerRadius (inner radius), outerRadius (outer radius), minIsotropy (minimum isotropy measure), angleShape (angle shape measure), minEdge (minimum straight edge length), maxEdge (maximum straight edge length), volume (element volume),
    - Highlight criteria: Smaller/Larger,
    - Highlight limit,
    - Minimum/Maximum/Average/Standard deviation.

- 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), Geometry (Area/Volume), Center of mass, Appearance,
  - Transform: Translate, Rotate, Mirror, Scale – performs the selected transformation on the part, Translate and Rotate may also create patterns, Mirror and Scale can modify the original part or create a transformed copy; the user has to specify:
    - Translate – start/end point coordinates, number of copies (for linear pattern),
    - Rotate – start/end axis point coordinates, rotation angle, number of copies (for polar pattern),
    - Mirror – mirror point coordinates, start/end direction point coordinates,
    - Scale – center point coordinates, scale factors in X, Y and Z axis.
  - Merge – assigns elements from multiple separate parts to one common part without changing the mesh,
  - Hide, Show and Show Only,

  - Set Color/Reset Color/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.

- Features
  - Reference point
    - Create,
    - Edit,
    - Duplicate,
    - Hide/Show/Show Only,
    - Delete.
  - Coordinate System
    - Create,
    - Edit,
    - Duplicate,
    - Hide/Show/Show Only,
    - Delete.

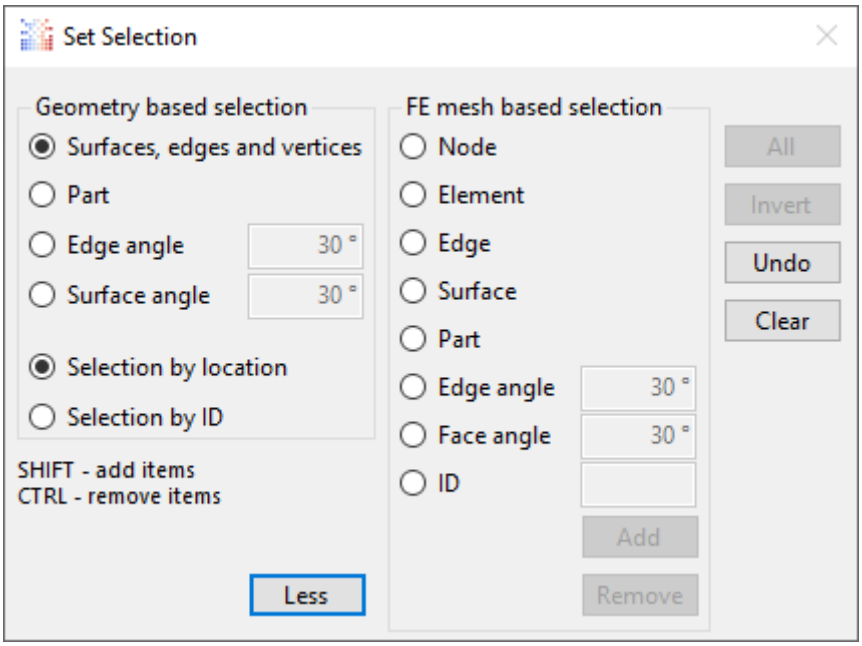


Figure 12: Set selection window

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.

Two selection methods are available - by location and by ID. The first one is based on the location of the user selection (point or area). Location data is saved for when there is a need to recreate the selection (remeshing, regenerating). If the selection is recreated the location data is used to create the selection again. The second one uses the location of the user selection to get the geometry IDs of the underlying geometry and saves only the IDs. When the selection needs to be recreated the geometry IDs are directly used again. In standard usage, like remeshing and regenerating, those two methods are equivalent. However, if the geometry changes before remeshing or regenerating (Regenerate Using Other Files or Swap Part Geometries), the results of the selection may change. If the topology (numbering of vertices, edges, and faces) does not change and only the geometry (dimensions) changes, the ID-based selection will always select the same vertex, edge, face of part while the location-based method might fail to do so. The ID-based method can also be used to create reference points and coordinate systems. When two or three points are supposed to be selected (selection methods Between two points and Circle center), SHIFT must be used.

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:

- Coordinates,
- On point - uses the Set selection window described above,
- 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 and hide its name.

Coordinate systems (rectangular and cylindrical) can be created for use in boundary condition and load definitions (so that they act in local coordinate system directions instead of the global ones), in distributions and in postprocessing (for results transformation). To create such a local (user) coordinate system, one needs to specify 3 points - center, point in the 1st axis direction and point on the 1-2 plane. The following selection methods can be used to define each of those points:

- Coordinates,
- On point - uses the Set selection window described above,
- Between two points,
- Circle center by 3 points.

5.2 Property Menu

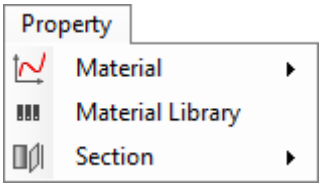


Figure 13: Property menu

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.

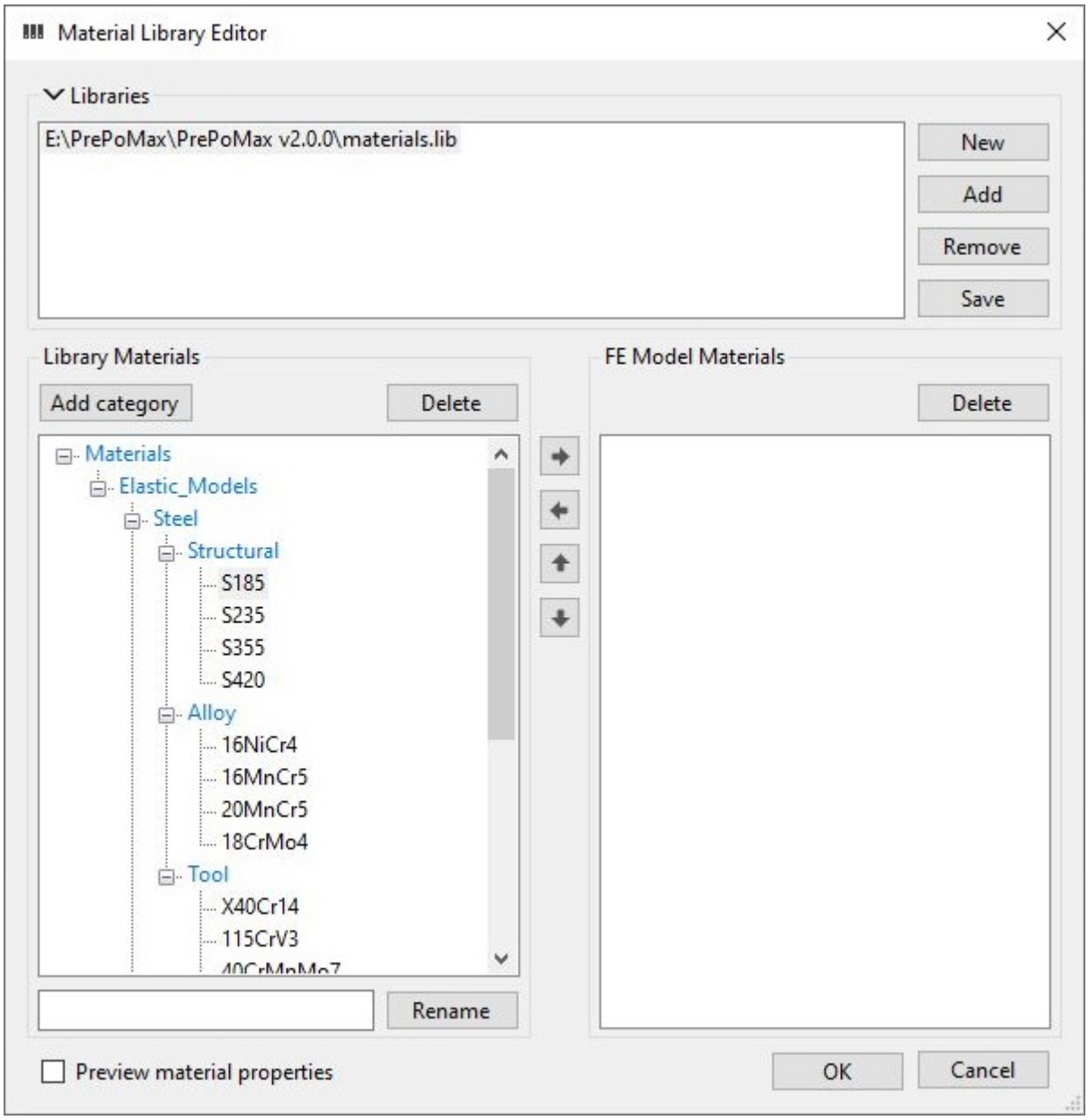


Figure 14: Material library

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 Add category button and removed from the library using the Delete button. Each item in the library can be renamed using the bottom text field and the Rename button. If changes were made to the library, they must be saved using Save 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 Delete 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.

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.
- Elastic
  - Young's modulus.
  - Poisson's ratio.
- Engineering Constants
  - Normal moduli: E1, E2, E3.
  - Poisson's ratios: v12, v13, v23.
  - Shear moduli: G12, G13, G23.
- Lamina
  - Normal moduli: E1, E2.
  - Poisson's ratios: v12.
  - Shear moduli: G12, G13, G23.
- 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).

The following section types are available:

- 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.
- Point Mass
  - Name – specify the section name.
  - Mass – enter the mass to be applied to each selected node.
  - Region – select the region for the section assignment.
- Distributed Mass
  - Name – specify the section name.
  - Mass – enter the total mass to be distributed to the selected region.
  - Region – select the region for the section assignment.

5.3 Interaction Menu

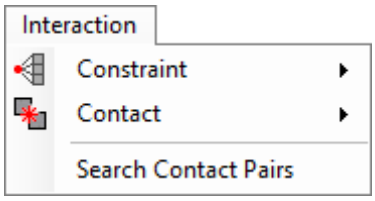


Figure 15: Interaction menu

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 Support:
    - Name.
    - Support type: Spring/Compression only.
    - Region.
    - For Spring:
        * Stiffness per area: No/Yes
        * Stiffness:
            · K1 – stiffness in the X direction.
            · K2 – stiffness in the Y direction.
            · K3 – stiffness in the Z direction.
    - For Compression only:
        * Gap clearance.
        * Spring stiffness.
        * Tensile force.
        * Offset.
        * Nonlinear: No/Yes
    - Color.

- Rigid Body:
    - Name.
    - Reference point.
    - Region.
    - Color.

- Coupling:
    - Name.
    - Coupling type: Kinematic/Distributing.
    - Reference point.
    - Region.
    - DOF: U1-U3 (Coupled or Unconstrained) for Kinematic, U1-U3 and UR1-UR3 (Distributed or Unconstrained) for Distributing.
    - 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.

- Joint:
  - Name.
  - Joint type: Hinge/Cylindrical/Slider/Ball/Weld.
  - Formulation: Rigid body/Springs.
  - Master Region.
  - Slave Region.
  - Joint centre: Defined by: Master surface/Coordinates/On point/Between two points/Circle center by 3 points.
  - Joint axis: Defined by: Master surface/User defined, not available for Ball and Weld types.
  - Spring stiffness: Defined by: From model/User defined, available only for Springs formulation.
  - Master surface color.
  - Slave surface color.

- Cyclic Symmetry:
  - 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.
  - Number of sectors.
  - Displayed sectors.
  - Check sector angle.
  - Axis point coordinates.
  - Axis direction components.
  - Master surface color.
  - Slave surface color.

The compression-only constraint internally uses GAP elements connected to other finite elements via *EQUATION. The Offset parameter enables the offset of the ground node of GAP element away from the model for better visualization of the results. In the results, GAP elements are merged into one part. IMPORTANT: geometric nonlinearity (NLGEOM) should be enabled when using the compression-only constraint. Otherwise, the compression-only support has 1/2 of the assigned stiffness and acts as a surface spring support. Setting the Nonlinear option to Yes also makes it possible to avoid this linearization by adding a dummy nonlinear material to the input file.

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.

Coupling constraints couple the motion of nodes on the selected surface with the motion of a reference node. Kinematic couplings add rigidity to the nodes, similarly to rigid body constraint. To work with shells, they require node-based surfaces. Distributing couplings (implemented as *COUPLING with *DISTRIBUTING in PrePoMax) don't add rigidity - they apply point loads equivalent to forces/moments defined at the reference node (boundary conditions cannot be prescribed this way) and allow for the deformation among the nodes on the selected surface. They don't seem to work with shells.

Joints connect two surfaces in one of two ways:

- with rigid bodies - each region becomes a rigid body whose reference node is located at the joint centre (the joint is exact, but a rigid body on shell elements does not converge in a geometrically nonlinear analysis).
- with springs - each region node is connected by axial springs to nodes on the joint axis, and the axis nodes of the two regions are tied by linear equations. The joint is as stiff as its springs. It needs no rigid body constraints and works with shells in nonlinear analyses.

The available joint types are:

- Hinge - the two regions share the joint centre and rotate relative to each other only about the joint axis (other DOFs are constrained via linear equation constraints).
- Cylindrical - the two regions rotate about and translate along the joint axis relative to each other (other DOFs are constrained via linear equation constraints).
- Slider - the two regions translate along the joint axis relative to each other and do not rotate (other DOFs are constrained via linear equation constraints).
- Ball - the two regions share the joint centre and rotate freely relative to each other.
- Weld - the two regions move as one rigid body.

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.

Cyclic symmetry allows the models exhibiting rotational periodic symmetry (repetitive circular patterns around a symmetry axis) to be analyzed using just a single representative sector. The assumption is that boundary conditions and loads also exhibit this form of symmetry. Tangential loads can be applied and thus torsion can be simulated this way. However, centrifugal load is commonly used with such models. Cyclic symmetry can be defined by selecting the two faces of the cut similarly as with tie constraint and setting the number of sectors, the number of displayed sectors and the symmetry axis. Note that boundary conditions can't be applied to the nodes of the slave surface of cyclic symmetry (slave surface in the underlying tie constraint definition) since that would cause an overconstraint.

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.
  - Massless – used to model perfectly hard contact within explicit dynamic calculations.
- 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.

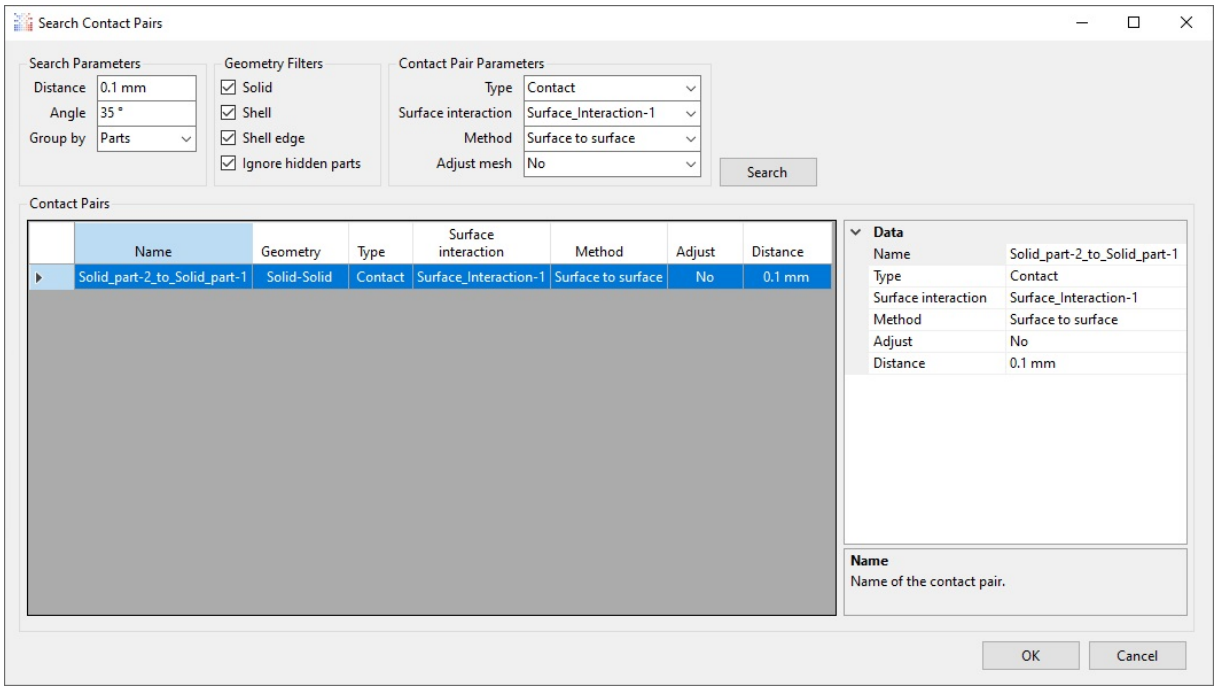


Figure 16: Search Contact Pairs tool

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 graph 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 Distribution Menu

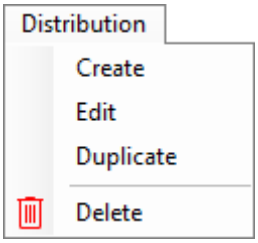


Figure 17: Distribution menu

The Distribution menu (Figure 17) contains the following options:

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

Distributions are used to define the spatial variation of selected magnitudes of loads, initial/boundary conditions and defined fields.

The following distribution types and settings are available:

- From Equation – defines a distribution based on the specified equation
  - Name
  - Type: Scalar/Vector
  - Magnitude – magnitude equation
  - Coordinate system – Global or selected local coordinate system
- From File – defines a distribution from values defined in ASCII (text) file
  - Name
  - File name: selects .txt file with distribution definition
  - Interpolator: Closest point/Gauss/Shepard
  - Interpolator radius – for Gauss and Shepard interpolator
  - Scale factors – scale factors for the X, Y and Z direction
  - Translations – translations in the X, Y and Z direction
  - Coordinate system – Global or selected local coordinate system

The distribution from file needs an ASCII (text) file that contains the data in the following format for scalar distributions:

X Y Z V1

where X, Y and Z are the spatial coordinates of the point. V1 is the value of the distribution. The # symbol can be used to denote a comment line.

For a vector distribution, three values V1, V2 and V3 are expected. Vector distribution can be used to specify the distribution of the Surface traction load in all three directions. The unit for the distributed surface traction is force/area.

5.5 Amplitude Menu

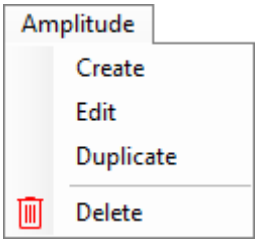


Figure 18: Amplitude menu

The Amplitude menu (Figure 18) contains the following options:

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

The following amplitude types and settings are available:

- 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

5.6 Initial Condition Menu

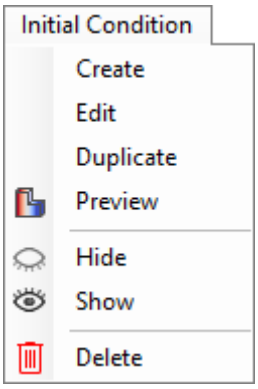


Figure 19: Initial condition menu

The Initial condition menu (Figure 19) contains the following options:

- Create.
- Edit.
- Duplicate.

- Preview.
- Hide.
- Show.
- 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.
  - Distribution – Constant or selected distribution item
  - Color.
- 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).
  - Color.
- Angular velocity – can be used to specify the angular velocity at the beginning of a dynamic analysis
  - Name.
  - Region.
  - Rotation center coordinates:
    - X - X coordinate of the axis point.
    - Y - Y coordinate of the axis point.
    - Z - Z coordinate of the axis point.
  - Rotation axis components:
    - N1 - axis component in the direction of the first axis.
    - N2 - axis component in the direction of the second axis.
    - N3 - axis component in the direction of the third axis.
  - Magnitude - value of the angular velocity around the axis defined by the point and direction.
  - Color.

5.7 Step Menu

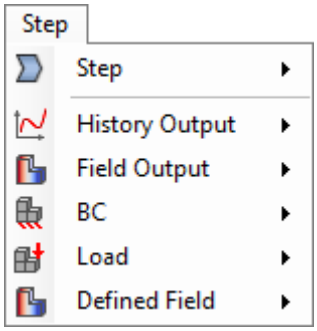


Figure 20: Step menu

The Step menu (Figure 20) contains the following options:

- Step
  - Create.
  - Edit.
  - Edit Controls – opens step controls editor where advanced solver controls (*CONTROLS keyword) can be changed. The following controls are available: Reset, Time Incrementation, Field, Contact. Those settings should be changed only as a last resort and when fully understanding their meaning.
  - 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.
  - Preview – can be used to preview temperature boundary conditions.

  - Hide.
  - Show.
  - Delete.

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

5.7.1 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.

- Complex frequency step
  - Name.
  - Perturbation: On/Off (when it's set to On, preload from the previous static step can be included).
  - Number of frequencies – number of eigenfrequencies to compute.
  - 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 od 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.

One has to be careful when using the Default incrementation and amplitudes - incrementation is not automatically adjusted to account for the spread and range of amplitude data points. One should change the time period, initial and maximum time increment accordingly.

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:

$$V = k \cdot \frac{F}{H} \cdot s \tag{1}$$

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]:

$$h_i = \int \frac{k}{H} \cdot p_i \cdot ds \tag{2}$$

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 nonlinear 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.

5.7.2 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 the values summed for all nodes in the set.
  - Global: Yes/No – whether the results are saved in global or local coordinate system
  - 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.
    - SDV – internal state variables.
  - 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.
  - Global: Yes/No – whether the results are saved in global or local coordinate system
  - 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.

- Section output
  - Name.
  - Variables to output
    - SOF – section forces.
    - SOM – section moments.
    - SOAREA – section area.
  - Region – region for the creation of the history output.

No history outputs are created by default.

5.7.3 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.
  - Global: Yes/No – whether the results are saved in global or local coordinate system

- 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).
    - SDV – internal state variables.
  - Global: Yes/No – whether the results are saved in global or local coordinate system

- 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.

5.7.4 Boundary condition

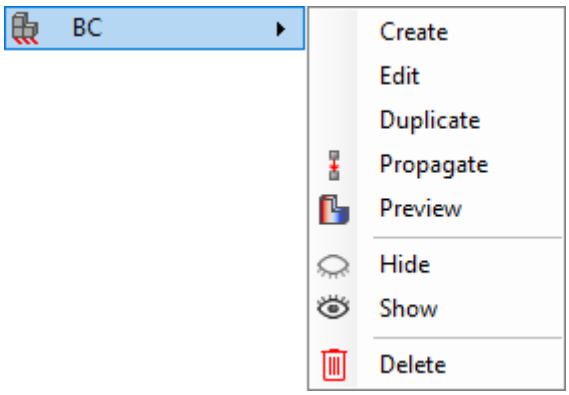


Figure 21: Boundary Conditions submenu

The Boundary Conditions (BC) submenu (Figure 21) 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 – Default or selected amplitude item.
  - Coordinate system – Global or user-defined one.
  - 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.
  - Distribution – Constant or selected distribution item.
  - Amplitude – Default or selected amplitude item.
  - 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 and From global model. Usage of submodeling technique is presented in the following videos: PrePoMax & CalculiX - Submodel and PrePoMax (CalculiX FEA) - Tutorial 13 - Stress concentration - submodeling.

5.7.5 Load

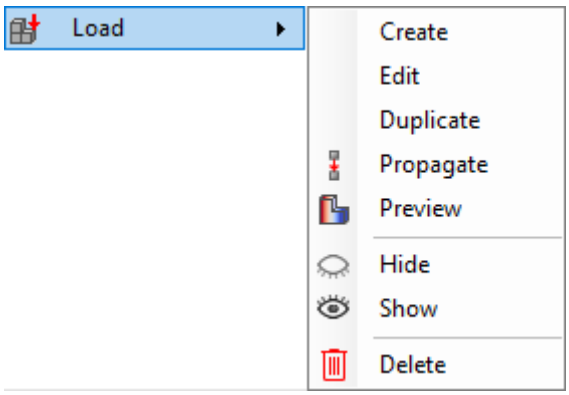


Figure 22: Load submenu

The Load submenu (Figure 22) 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 – Default or selected amplitude item.
  - Coordinate system – Global or user-defined one.
  - 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 – Default or selected amplitude item.
  - Color.
- Pressure
  - Name.
  - Region.
  - Magnitude – value of the pressure load.
  - Phase – pressure phase (for steady state dynamics only).
  - Distribution – Constant or selected distribution item.
  - Amplitude – Default or selected amplitude item.
  - 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 – Default or selected amplitude item.
  - 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 – Default or selected amplitude item.
  - 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).
  - Distribution – Constant or selected distribution item.
  - Amplitude – Default or selected amplitude item.
  - Coordinate system – Global or user-defined one.
  - Color.

- Normal shell edge load
  - Name.
  - Region.
  - Magnitude – value of the shell edge load.
  - Phase – edge load phase (for steady state dynamics only).
  - Amplitude – Default or selected amplitude item.
  - 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 – Default or selected amplitude item.
  - 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 – Default or selected amplitude item.
  - 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 – Default or selected amplitude item.
  - Color.
- Concentrated flux
  - Name.
  - Add flux: Yes/No – add flux to the previously defined fluxes.
  - Region.
  - Flux – value of the flux per node.
  - Amplitude – Default or selected amplitude item.
  - Color.
- Surface flux
  - Name.
  - Region.
  - Flux per area.
  - Amplitude – Default or selected amplitude item.
  - Color.
- Body flux
  - Name.
  - Region.
  - Flux per volume.
  - Amplitude – Default or selected amplitude item.
  - Color.
- Convective film
  - Name.
  - Region.
  - Sink temperature.
  - Film coefficient.
  - Sink amplitude – Default or selected amplitude item for sink temperature.
  - Coefficient amplitude – Default or selected amplitude item for film coefficient.
  - 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.
- Sink amplitude – Default or selected amplitude item for sink temperature.
- Emissivity amplitude – Default or selected amplitude item for emissivity.
- 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. Surface traction load may also be applied to free shell edges.

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 Symbols dropdown list, that allows the selection of how the symbols are displayed (`None`, `Model`, `Step-1`, ...).

5.7.6 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.
    - Distribution – Constant or selected distribution item.
    - Amplitude – Default or selected amplitude item.
  - Define temperature: From file
    - Results file .frd.
    - Step number.
  - Color.

5.8 Analysis Menu

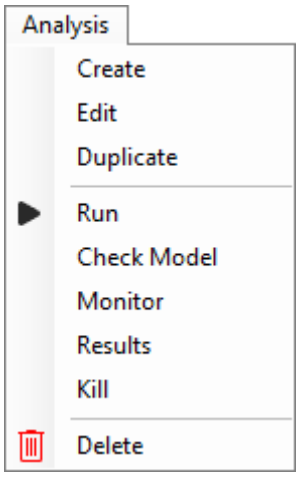


Figure 23: Analysis menu

The Analysis menu (Figure 23) contains the following options:

- Create.
- Edit – gives access to name, FEM solver selection (CalculiX, Abaqus or FEMaster), additional executable 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 output, job progress (incrementation and convergence plots) and status, convergence information and dat file contents.
- 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.
- Features
  - Reference Points – can be created in the same ways as in preprocessing.
  - Coordinate Systems – can be created in the same ways as in preprocessing and used for results transformation.
- Results
  - Field outputs.
  - History outputs.

6.1 Results Menu

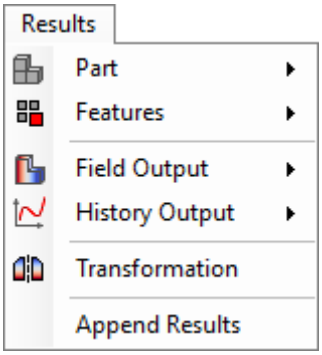


Figure 24: Results menu

The Results menu (Figure 24) contains the following options:

- Part

  - Edit – allows changing part name and color, provides information about its type, mesh, geometry (volume/area) and center of mass.
  - Merge.
  - Hide/Show/Show Only.
  - Set Color/Reset Color/Set Transparency.
  - Color Contours off – disables color contours only for selected parts.
  - Color Contours on.
  - Delete.

- Features
  - Reference Points.
    - Create.
    - Edit.
    - Duplicate.
    - Hide/Show/Show Only.
    - Delete.
  - Coordinate Systems.
    - Create.
    - Edit.
    - Duplicate.
    - Hide/Show/Show Only.
    - 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), Filter 1 / Filter 2 (None, Median, Average, Smooth): Source type (All nodes, Surface nodes), Number of layers, Number of iterations.
      - 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, Filter 1 / Filter 2 (None, Median, Average, Smooth): Source type (All nodes, Surface nodes), Number of layers, Number of iterations.
    - Equation - creates new field output by evaluating a specified equation on the existing outputs, supports functions from the Ncalc library, existing outputs are specified as Field_Name.Component_Name
      - Properties: Name, Equation, User defined unit, Filter 1 / Filter 2 (None, Median, Average, Smooth): Source type (All nodes, Surface nodes), Number of layers, Number of iterations.
    - Coordinate System Transform - transforms the selected field to the selected user-defined coordinate system

            - Properties: Name, Field name, Coordinate system, Filter 1 / Filter 2 (None, Median, Average, Smooth): Source type (All nodes, Surface nodes), Number of layers, Number of iterations.
    - Edit.
    - Duplicate.
    - Export.
    - Delete – selected field outputs can be deleted to decrease the file size.
- History Output
    - Create
        * From Field Output:
            - Name.
            - Region.
            - Field name.
            - Components.
            - Step id.
            - Node coordinates - whether to include node coordinates in the history output.
            - Filter 1/Filter 2: Type: None/Minimum/Maximum/Sum/Average.
            - Filter 1/Filter 2: Return: Column(s)/Row(s) - filters can be used in pairs since a single filter takes in the history output, which is, in general, a matrix (time versus the selected node values) and returns a row or a column. For example, if the maximum filter is selected and the return type is Column, the filter will look for the largest value in the history output matrix and return the column where this maximum value is located. This way, the entire time history for the node where the maximum value is determined is reported.
        * From History Output by Equation:
            - Name.
            - Equation - creates new history output by evaluating a specified equation on the existing outputs, supports functions from the Ncalc library, existing outputs are specified as History_Output_Name.Field_Name.Component_Name.
            - User defined unit.
            - Filter 1/Filter 2: Type: None/Minimum/Maximum/Sum/Average.
            - Filter 1/Filter 2: Return: Column(s)/Row(s)
        * From Integration – calculates line, surface or volume integral of the selected output variable:
            - Name.
            - Integral type: Line/Surface/Volume.
            - Integral result: Geometry/Field/Field by Geometry.
            - User defined unit.
            - Cross section: None/Thickness/Axisymmetric.
            - Region.
            - Field name.
            - Components.
            - Variable: Displacements/Forces/Surface Normals/Wear Depths/Mesh Deformation/Disp&Def/Disp&Def&Depth.

        - Filter 1/Filter 2: Return: Column(s)/Row(s)
        - Filter 1/Filter 2: Return: Column(s)/Row(s)
      - From Merged Histories – merges compatible history components:
        - Name.
        - History component 1-4.
        - Filter 1/Filter 2: Type: None/Minimum/Maximum/Sum/Average.
        - Filter 1/Filter 2: Return: Column(s)/Row(s)
      - From Extracted Entry – extracts a single entry from existing history component into a new history output:
        - Name.
        - History component.
        - Entry.
        - Filter 1/Filter 2: Type: None/Minimum/Maximum/Sum/Average.
        - Filter 1/Filter 2: Return: Column(s)/Row(s)
      - XY – creates XY plot from two history components:
        - Name.
        - X history component.
        - Y history component.
        - Filter 1/Filter 2: Type: None/Minimum/Maximum/Sum/Average.
        - Filter 1/Filter 2: Return: Column(s)/Row(s)
      - From Table – creates history output from data entered manually by user:
        - Name.
        - Number of Y columns.
        - User defined X-axis unit.
        - User defined Y-axis unit.
        - Filter 1/Filter 2: Type: None/Minimum/Maximum/Sum/Average.
        - Filter 1/Filter 2: Return: Column(s)/Row(s)
    - Edit.
    - Duplicate.
    - Export - exports selected history outputs to CSV file.
      - File name.
      - History outputs.
      - Delimiter - comma, semicolon or colon.
    - 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 25: Results toolbar - part 1



Figure 26: Results toolbar - part 2

Selected post-processing options are also available as shortcuts in the Results toolbar (Figure 25 - 26). 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 (for contact wear analyses), Mesh Deformation (mesh deformation from the previous step, for contact wear analyses), Disp&Def (sum of displacement and mesh deformation vectors, for contact wear analyses), Disp&Def&Depth (sum of displacement, mesh deformation and wear depth vectors, for contact wear analyses).

c. Type (deformation scale type): Undeformed, True scale, Automatic, Automatic x 0.25, Automatic x 0.5, Automatix 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. Color contours.

j. Vector plot.

k. Show undeformed wireframe model.

l. Show undeformed solid model.

m. Transformation.

n. First increment.

o. Previous increment.

p. Drop-down list of increments.

q. Next increment.

r. Last increment.

s. Animate.

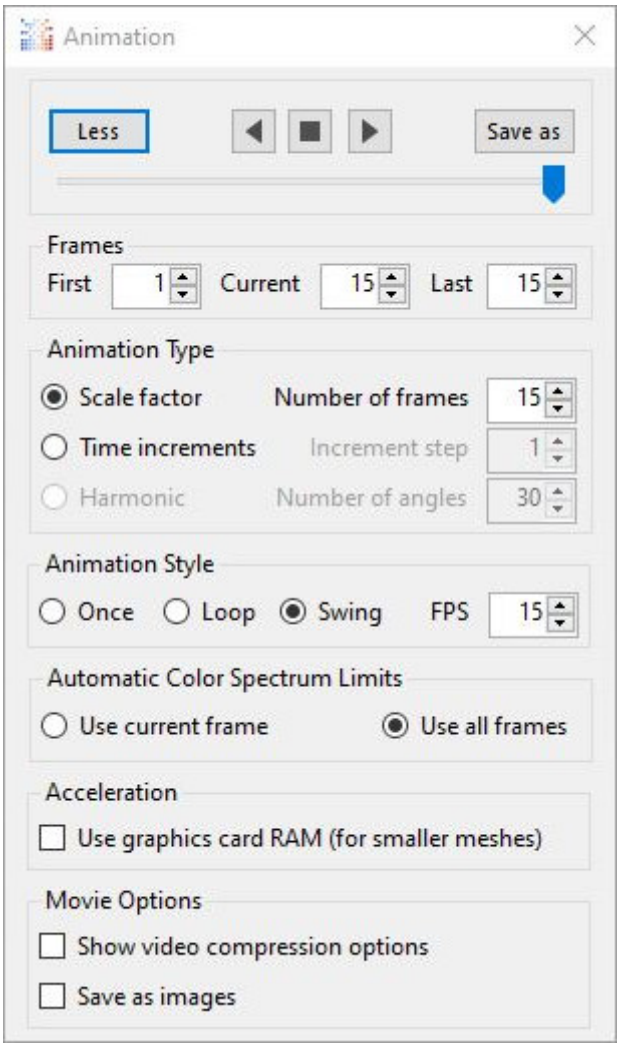


Figure 27: Animation window

There are several settings for animating the results available after clicking the More button. They are shown in Figure 27. The Scale factor animation mode goes from the undeformed shape to the currently defined deformation scale factor in the current results frame - other results frames are not used). The Time increments animation mode goes through all the results frames. The Harmonic animation mode is meant for Steady State Dynamics step results. It uses a single results frame and goes through the complex output angles.

The animation style can also be selected. It can run once, in a loop or swing back and forth. Other settings include the specification of the number of animation frames, result increments and complex output angles for each of the three animation modes. It is also possible to specify the number of frames per second (FPS), adjust the color legend limits (to be based on the current frame or on all frames), enable GPU acceleration, show video compression options when saving to a file and saving the animation as images. Normally, the Save as button allows the animation to be saved to .avi file.