

PrePoMax v1.1.0

Manual

29rd of July 2021

Written by Jakub Michalski and Matej Borovinšek

Contents

- 1 What is PrePoMax? 1
- 2 Currently available analysis features..... 2
- 3 Workflow, general options..... 4
 - 3.1 File Menu 6
 - 3.2 Edit Menu 7
 - 3.3 View Menu 8
 - 3.4 Tools Menu 9
 - 3.5 Help Menu 11
- 4 Geometry and mesh options..... 11
 - 4.1 Geometry Menu 11
 - 4.2 Mesh Menu 12
- 5 FE Model options..... 14
 - 5.1 Model Menu 14
 - 5.2 Property Menu 16
 - 5.3 Interaction Menu..... 20
 - 5.4 Initial Condition Menu..... 22
 - 5.5 Step Menu 23
 - 5.5.1 Step..... 24
 - 5.5.2 History Output..... 26
 - 5.5.3 Field Output..... 27
 - 5.5.4 Boundary condition 27
 - 5.5.5 Load 29
 - 5.5.6 Defined Field..... 32
 - 5.6 Analysis Menu 33
- 6 Results options 34
 - 6.1 Results Menu..... 34

1 What is PrePoMax?

PrePoMax is a freeware pre- and postprocessor for open-source Finite Element Analysis solver CalculiX. PrePoMax is being developed by dr. Matej Borovinšek from University of Maribor, Slovenia. The current version is 1.1.0. 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/>

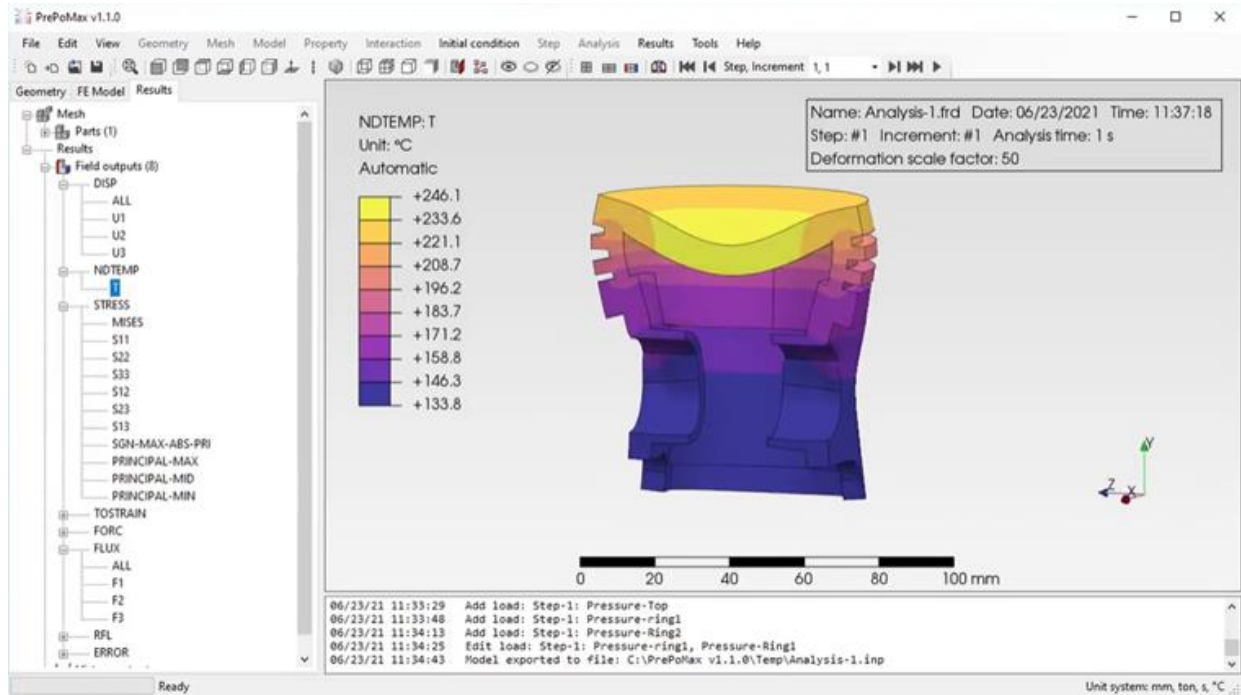


Figure 1: PrePoMax user interface

2 Currently available analysis features

- Analysis step types:
 - Static
 - Natural frequency extraction
 - Linear buckling
 - Heat transfer
 - Uncoupled temperature-displacement
 - Coupled temperature-displacement
- Finite element types:
 - Solid tetrahedrons (first and second order)
 - Shell triangles or quadrilaterals (first and second order)
 - Import of solid wedges (first and second order)
 - Import of solid hexahedrons (first and second order)
- Material models (isotropic, with optional temperature dependency):
 - Linear elasticity
 - Plasticity
 - Thermal
- Constraints:
 - Rigid body
 - Tie
- Interactions:
 - Contact
 - Friction
 - Gap conductance
- Initial conditions
 - Temperature
- Boundary conditions:
 - Fixed
 - Displacement/Rotation
 - Submodel
 - Temperature

- Loads:
 - Concentrated force
 - Moment
 - 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. Each tab also activates proper menus in the top toolbar:

- 1) Geometry – geometry editing and mesh generation (Figure 2).

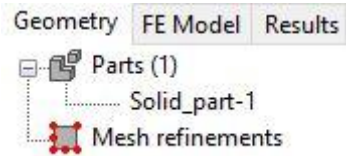


Figure 2: Geometry tab of the model tree

- 2) FE Model – analysis set up (pre-processing) (Figure 3).

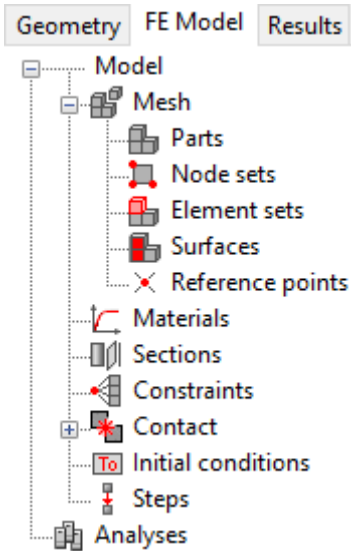


Figure 3: FE Model tab of the model tree

- 3) Results – post-processing (Figure 4).

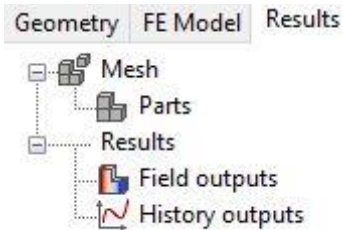


Figure 4: Results tab of the model tree

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

- 1) Create a new model (File → New) and select a unit system.
- 2) Import CAD geometry for the analysis (File → Import).
- 3) Set mesh parameters (Mesh → Meshing Parameters) and generate a mesh (Mesh → Create Mesh).
- 4) When the mesh is successfully created, proceed to the FE Model part of the model tree.
- 5) Define material for the analysis (Materials → Create), add Elasticity, Density (for performing a modal analysis or using gravity or centrifugal loads) and Plasticity (to account for permanent deformations).
- 6) Create a new section with the material defined before (Section → Create). Section type can be Solid (for using solid elements) or Shell (for using shell elements). In the latter case, the thickness must also be specified. 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) and/or initial condition (Initial condition → 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 advised to work using node and elements 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 also possible to add features without creating sets/surfaces first. In such cases, the selection tool within the feature window will let the user pick only parts of the mesh.

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

View controls in PrePoMax:

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

For selection, use the left mouse button (drag the mouse to use the selection box).

3.1 File Menu

The first menu in the menu bar is the File menu shown in Figure 5.

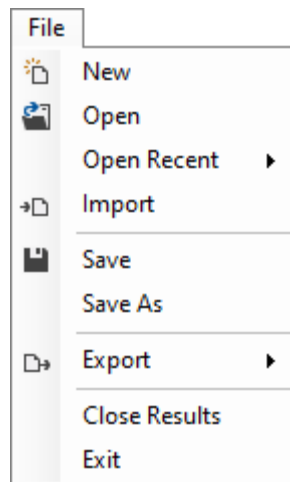


Figure 5: File menu

It contains the following options:

- New – create a new model. The following unit systems are available (derived units are listed when a unit system is selected):
 - Unitless,
 - m, kg, s, °C,
 - mm, ton, s, °C,
 - m, ton, s, °C,
 - in, lb, s, °C.
- Open – open a previously created model.
- Open Recent – open one of the recently used models or clear the list of the recent files.
- Import – import CAD geometry, mesh or model saved in one of the following formats:
 - Step,
 - Iges,
 - Brep,
 - STL,
 - Universal,
 - Netgen,
 - Abaqus/Calculix inp,
 - Mmg mesh.
- Save – 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:
 - Calculix input file,
 - Abaqus input file,
 - Step,
 - Brep,

- Mmg,
- STL.
- Close Results – close the results from the post-processing part of the model tree (saving the pmx model without the results reduces the file size).
- Exit – close PrePoMax (the user will be asked if the model should be saved first).

3.2 Edit Menu

The second menu in the menu bar is the Edit menu shown in Figure 6.

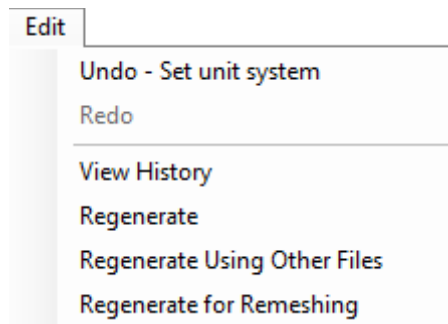


Figure 6: Edit menu

It contains the following options:

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

The following menu in the menu bar is the View menu shown in Figure 7.

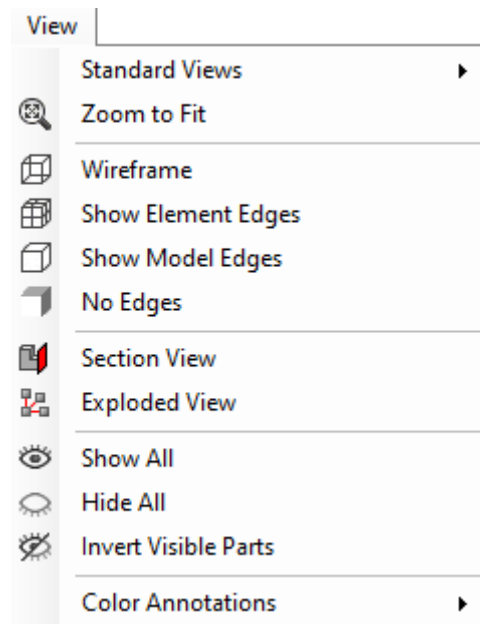


Figure 7: View menu

It contains the following options:

- Standard Views – apply one of the predefined views:
 - o Front/Back,
 - o Top/Bottom,
 - o Left/Right,
 - o Normal,
 - o Vertical,
 - o Isometric.
- Zoom to Fit – zoom the model in such a way that it fits in the window.
- Wireframe – apply a view showing only model edges.
- Show Element Edges – apply a view showing the model with the finite element edges.
- Show Model Edges – apply a view showing the model and its feature edges.
- No Edges – apply a view with showing the model without all edges.
- Section View – cut the model using a plane defined by a point and a normal or disable the section view (this option can be used in post-processing as well).
- Exploded View – show assembly parts in an exploded view (select exploded view style, magnification and scale factor).
- Show All – show (unhide) all parts of the model.
- Hide All – hide all parts of the model.
- Invert Visible Parts – invert the visibility of parts.
- Color Annotations – use colors to distinguish items of the model based on various criteria.

Selected options from the File and View menus can be accessed directly using a toolbar menu shown in Figure 8.



Figure 8: File and View toolbars

3.4 Tools Menu

PrePoMax also features the Tools and Help menus. The Tools menu is shown in Figure 9.

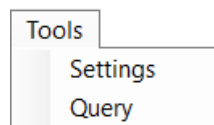


Figure 9: Tools menu

It contains the following options:

- Settings
 - General
 - Open last file, last file name.
 - Save results in .pmx files.
 - Default unit system.
 - Import mesh → Edge angle.
 - Graphics
 - Background: Type, Top color, Bottom color.
 - Geometry: CAD deflection.
 - Lighting: Ambient component, diffuse component.
 - Smoothing: Point smoothing, Line smoothing.
 - Widgets: Coordinate system visibility, Scale widget visibility.
 - Default Colors
 - Face orientation: Front face color, Back face color.
 - General.
 - Meshing – default settings for meshing
 - Mesh size: Grading, Elements per edge, Elements per curvature,
 - Mesh optimization: Optimize steps 2D, Optimize steps 3D,
 - Mesh type: Second order, Midside nodes on geometry, Quad-dominated mesh,
 - Mesh operations: Split compound mesh.
 - Pre-processing
 - Color bar: Background type, Draw a border rectangle.
 - Selection: Primary/secondary/mouse highlight color.

- Symbols: Constraints/BCs/Loads color, Symbol size, Node symbol size, Draw symbol edges.
 - Calculix
 - Calculix: Work directory, Use .pmx folder as work directory, Executable, Default solver.
 - Parallelization: Number of processors, Environment variables.
 - Post-processing
 - Deformation
 - Deformation scale factor: Automatic/True scale/Off/User defined,
 - Draw undeformed model: Yes/No,
 - Undeformed model color.
 - Limit values
 - Show max value location: Yes/No,
 - Show min value location: Yes/No.
 - History output
 - Max number of the history output entries.
 - Legend
 - Color spectrum settings
 - Color spectrum type: Cool-warm/Rainbow, Warm, Cool, Cividis, Viridis, Plasma, Black body, Inferno, Kindlmann, Grayscale,
 - Brightness,
 - Reverse colors: Yes/No,
 - Number of discrete colors – in the range of 2-24.
 - Color spectrum values
 - Number format: Scientific/General,
 - Number of significant digits – in the range of 2-8,
 - Min/Max limit type: Automatic/Manual.
 - Design
 - Background type: None/White,
 - Draw a border rectangle: Yes/No.
 - Status block
 - Design
 - Background type: None/White,
 - Draw a border rectangle: Yes/No.
- Query
 - Point/Node – check the node ID and its value.
 - Element – check the element ID.
 - Edge – check the edge ID and its length.
 - Surface – check the surface ID and its area.
 - Part – check part information (name, type, number of nodes and elements).
 - Assembly – check the number of parts, nodes and elements.
 - 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 circle passing through three selected points.

3.5 Help Menu

The Help menu is shown in Figure 10.

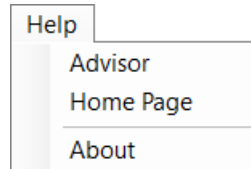


Figure 10: Help menu

It 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 a title card.

4 Geometry and mesh options

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

- Edit – allows editing of the name and the color of the part.
- Meshing Parameters – displays the parameters for meshing.
- 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 – enables to display geometry parts in the post-processing.
- Hide/Show/Show Only – options to hide and show the selected part.
- Set Transparency – setting translucency color component of the part.
- Expand All/Collapse All – expands or collapses all objects listed in the Part container
- Delete – deletes the selected part.

4.1 Geometry Menu

The Geometry menu is shown in Figure 11.

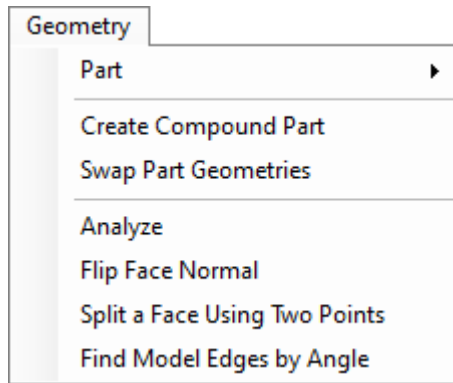


Figure 11: Geometry menu

It contains the following options:

- Part – selected options discussed above.
- Create Compound Part – merges two or more parts into one part (body).
- Swap Part Geometries – swaps part geometries between two parts.
- Analyze – checks the geometry for the following:
 - o Short edges,
 - o Close edges,
 - o Small faces.
- Flip Face Normal – flips the shell face orientation.
- Split a Face Using Two Points – divides a face using two selected points by projection.
- Find Model Edges by Angle – finds the model (feature) edges that fit into the specified angle criterion (only for stl files).

4.2 Mesh Menu

The Mesh menu is shown in Figure 12.

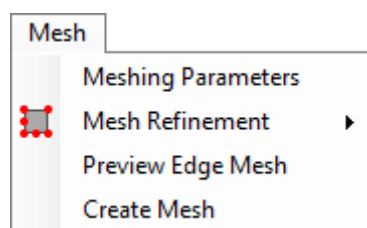


Figure 12: Mesh menu

It contains the following options:

- Meshing Parameters – settings for the mesher:
 - o Mesh size
 - Max element size,
 - Min element size,
 - Grading (0 – uniform mesh, 1 – aggressive local grading),
 - Elements per edge (number of elements per edge of the geometry),

- Elements per curvature (number of elements per curvature radius).
 - Mesh optimization
 - Optimize steps 2D (number of optimization steps to use for the 2D mesh),
 - Optimize steps 3D (number of optimization steps to use for the 3D mesh).
 - Mesh type
 - Second order: Yes/No,
 - Midside nodes on geometry (project the midside nodes on the geometry): Yes/No,
 - Quad-dominated mesh: Yes/No (for shell parts only).
 - Mesh operations
 - Split compound mesh: Yes/No (splits compound part mesh to unconnected part meshes).
- Mesh Refinement
 - Create,
 - Edit,
 - Delete.
- Preview Edge Mesh
- Create Mesh

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

5 FE Model options

Switching to FE Model tab activates the following menus: Model, Property, Interaction, Step, BCs, Load 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

The Model menu is shown in Figure 13.

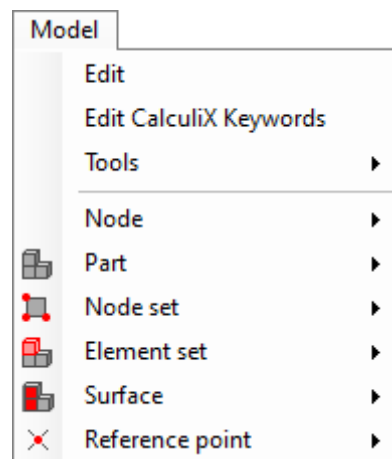


Figure 13: Model menu

It contains the following options:

- Edit
 - o Name,
 - o Type: Global or Submodel,
 - o Absolute zero – value of the absolute zero temperature used for radiation,
 - o Stefan-Boltzmann – value of the Stefan-Boltzmann constant used for radiation,
 - o Gravitational constant – value of the Newton’s gravitational constant.
- Edit CalculiX Keywords – a keyword editor that allows adding CalculiX keywords which are currently unsupported by PrePoMax.
- Tools
 - o Create Boundary Layer – creates a prismatic boundary layer on the surface, the thickness of the layer can be specified (used in analyses with a pre-tension load),
 - o Find Model Edges By Angle (searches for the feature edges based on the angle in the imported meshes),
 - o 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).
- Node

- Renumber All.
- Part
 - Edit,
 - Transform: Translate, Scale, Rotate,
 - Merge,
 - Hide/Show/Show Only,
 - Set Transparency,
 - Delete.
- Node set
 - Create,
 - Edit,
 - Duplicate,
 - Delete.
- Element set
 - Create,
 - Edit,
 - Duplicate,
 - Convert to Part,
 - Delete.
- Surface
 - Create,
 - Edit,
 - Delete.
- Reference point
 - Create,
 - Edit,
 - Delete.

When creating sets, surfaces and reference points, a Set selection window opens (Figure 14)

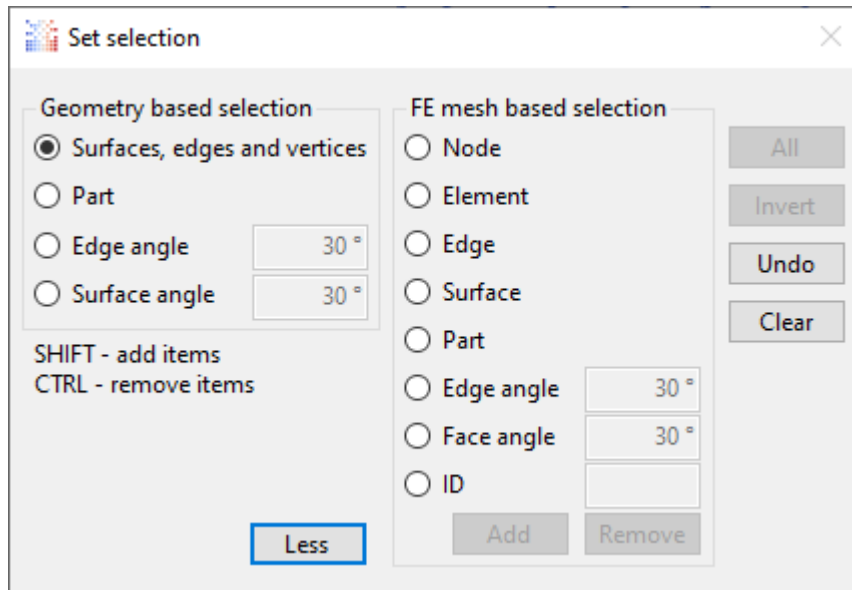


Figure 14: Set selection window

Sets, surfaces and reference points 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.

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

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

A reference point position can also be entered by coordinates. The user can also change the color of the reference point.

5.2 Property Menu

The Property menu is shown in Figure 15.

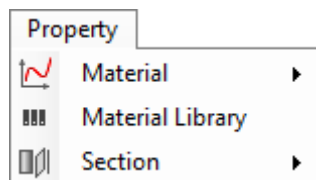


Figure 15: Property menu

It contains the following options:

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

Material Library editor is shown in Figure 16.

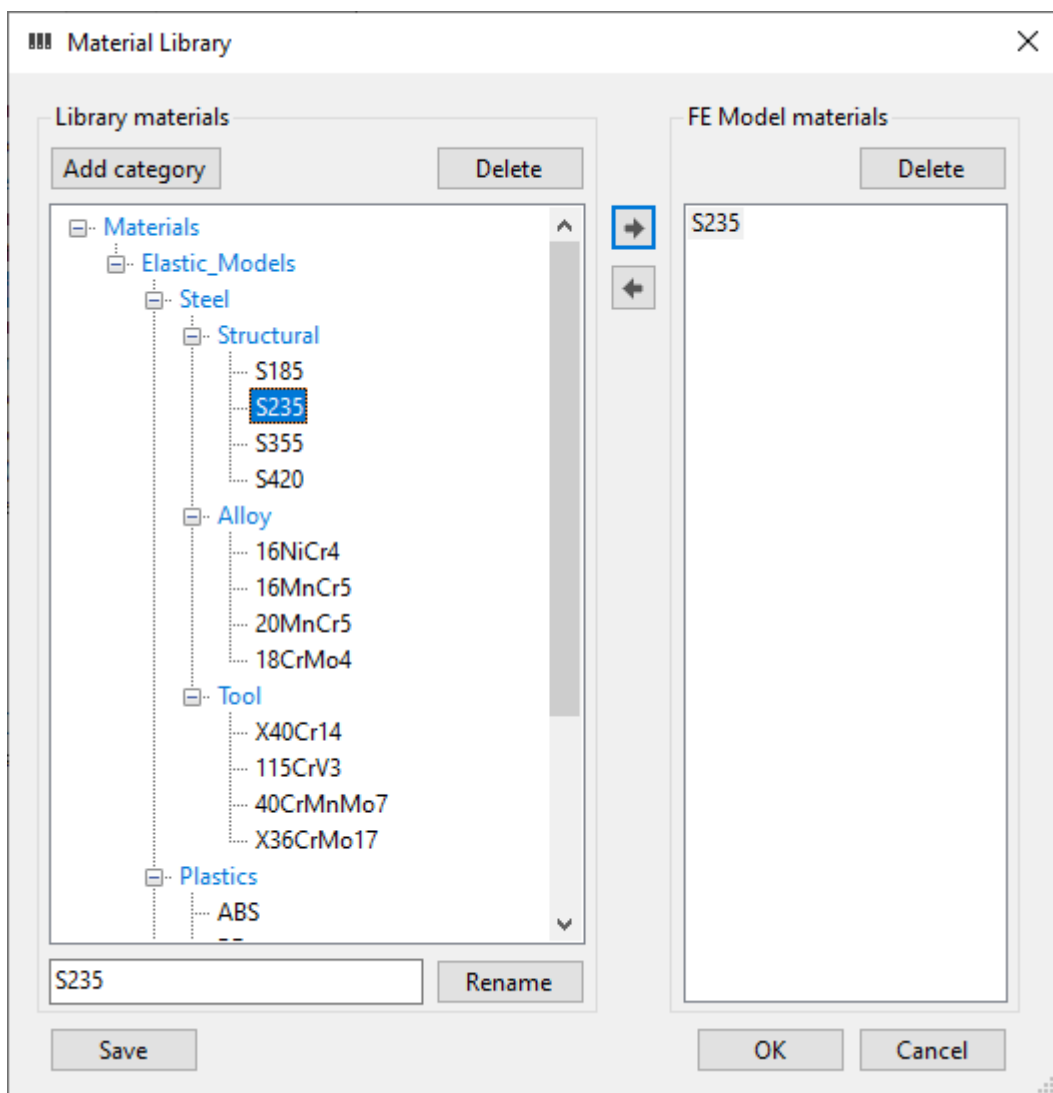


Figure 16: Material library editor

Materials library 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 is not easily readable.

The library is meant to be edited using the PrePoMax built-in material library editor shown in Figure 16. The editor is divided in two sections, a section for library materials and a section for the model materials.

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.

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.

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 first the material must be created using the Material editor shown in Figure 17. Then the material library editor (Figure 16) can be used to add the user material to the library. In the material library editor first, a desired category must be selected or created and then the left arrow button can be used to copy the user material to the library. Once a material is added to the library the changes must be saved for future use.

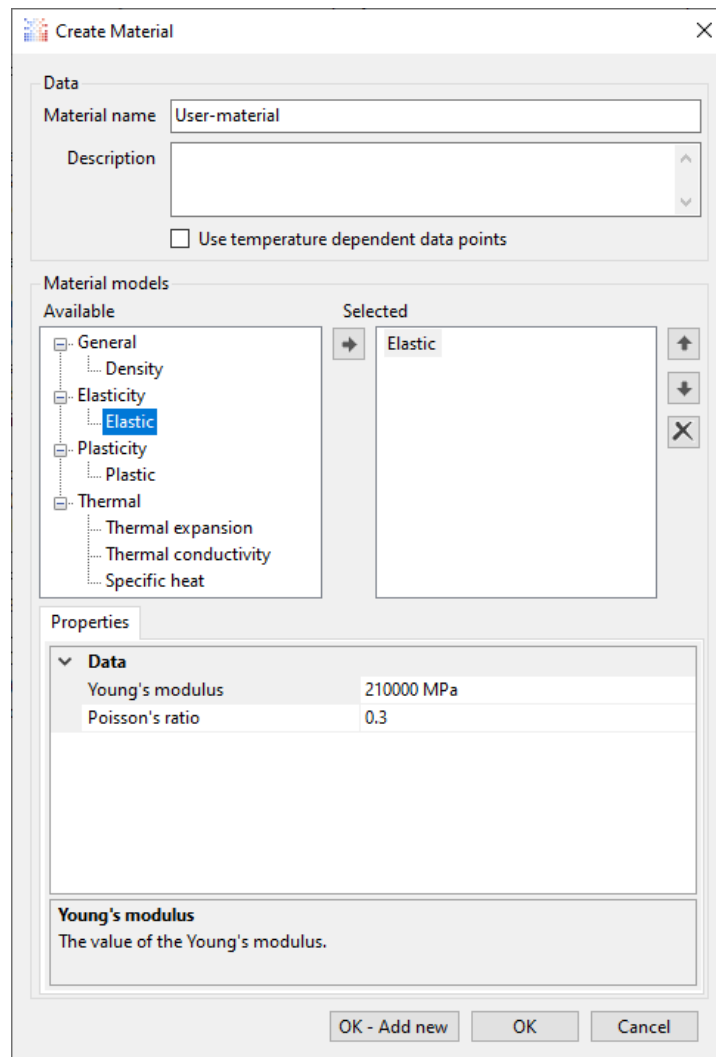


Figure 17: Material editor

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 (not empty) model containing the selected user material. This file must then be sent to the second user which opens the model file and adds the user material from the file to his materials library using the Material library editor (Figure 16).

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

- Density
- Elasticity
 - o Young's modulus,
 - o Poisson's ratio.

- Plasticity
 - Hardening:
 - Isotropic,
 - Kinematic,
 - Combined.
 - Data points (yield stress vs plastic strain).
- Thermal properties
 - Thermal expansion:
 - Thermal expansion coefficient,
 - Zero temperature.
 - Thermal conductivity.
 - Specific heat.

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

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

There are two available section types:

- Solid section
 - Name – specify the section name,
 - Material – select the material for the section,
 - 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.

5.3 Interaction Menu

The Interaction menu is shown in Figure 18.

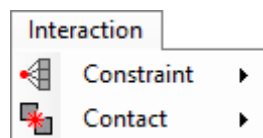


Figure 18: Interaction menu

It contains the following options:

- Constraint
 - Create,
 - Edit,
 - Hide,

- Show,
- Delete.
- Contact
 - Surface Interaction
 - Create,
 - Edit,
 - Duplicate,
 - Delete.
 - Contact Pair
 - Create,
 - Edit,
 - Hide,
 - Show,
 - Delete.

Available constraints include:

- Rigid body
 - Name,
 - Reference point,
 - Region,
 - Color.
- Tie
 - Name,
 - Position tolerance – the largest distance from the master surface for which the slave nodes will be included in the tie constraint (default: 2.5 % of the typical element size),
 - Adjust – projection of the slave nodes on the master surface,
 - Master region,
 - Slave region,
 - Master surface color,
 - Slave surface color.

The rigid body constraint requires a reference point since it constrains the motion of the selected nodes to the motion of the reference point. It is used to define a selected region as infinitely rigid (non-deformable). For example, it's often used when modelling manufacturing tools. However, it may also serve as a way to apply moments and remote loads to models. Rigid body constraint is the only way to apply a torque to the surface of a solid part (since solid element nodes don't have rotational degrees of freedom).

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

Contact is a more complex form of interaction between two surfaces than the tie constraint. It allows the transfer of forces between touching surfaces. However, contact is a highly

nonlinear phenomenon and requires careful usage as it may cause convergence issues. When defining contact, a surface interaction must be created first. Two options are available:

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

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

- Name
- Surface interaction
- Method
 - o Surface to surface,
 - o Node to surface.
- 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

5.4 Initial Condition Menu

The submenus of the Initial condition menu are shown in Figure 19.

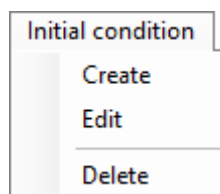


Figure 19: Initial condition menu

It contains the following options:

- Create,
- Edit,
- Delete.

The following initial condition types and settings are available:

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

5.5 Step Menu

The submenus of the Step menu are shown in Figure 20.

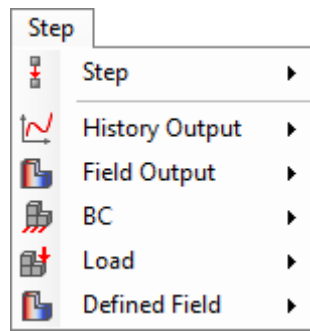


Figure 20: Step menu

It contains the following options:

- Step
 - o Create,
 - o Edit,
 - o Duplicate,
 - o Delete.
- History Output
 - o Create,
 - o Edit,
 - o Propagate,
 - o Delete.
- Field Output
 - o Create,
 - o Edit,
 - o Propagate,
 - o Delete.
- BC – Boundary condition
 - o Create,

- Edit,
- Propagate,
- Hide,
- Show,
- Delete.
- Loads
 - Create,
 - Edit,
 - Propagate,
 - Hide,
 - Show,
 - Delete.
- Defined Field
 - Create,
 - Edit,
 - Propagate,
 - Delete.

5.5.1 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 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.
- Frequency 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),
 - Storage: Yes/No – stores eigenvalues, eigenmodes, mass and stiffness matrix in a binary form in a file jobname.eig for further use,
 - Number of frequencies – number of eigenfrequencies to compute.
- 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).
- 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.
- 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.
- 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.

5.5.2 History Output

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

- Node output
 - Name,
 - Frequency – results of every N-th increment will be saved,
 - Variables to output
 - RF – reaction forces,
 - U – displacements,
 - NT – temperatures,
 - RFL – external concentrated heat sources,
 - Totals – this parameter applies to the external forces,
 - Region – region for creation of the history output.
- Element output
 - Name,
 - Frequency – results of every N-th increment will be saved,
 - 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,
 - EVOL – volume,
 - EBHE – heating power,
 - Totals – this parameter applies to the whole element variables (ELSE, EVOL),
 - Region – region for the creation of the history output.
- Contact output
 - Name,
 - Frequency – results of every N-th increment will be saved,
 - 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 – this parameter applies to the energy (CELS),
 - Contact pair – contact pair for creation of the history output.

No history outputs are created by default.

5.5.3 Field Output

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

- Node output
 - Name,
 - Frequency – results of every N-th increment will be saved,
 - Variables to output
 - RF – reaction forces,
 - U – displacements,
 - NT – temperatures,
 - RFL – external concentrated heat sources,
- Element output
 - Name,
 - Frequency – results of every N-th increment will be saved,
 - Variables to output
 - S – stresses,
 - E – total strains,
 - ME – mechanical strains,
 - PEEQ – equivalent plastic strain,
 - ENER – energy density,
 - HFL – heat flux,
 - ERR – extrapolation error estimator for stress calculations (ERR and ZZS are mutually exclusive),
 - HER – extrapolation error estimator for heat calculations (HER and ZZS are mutually exclusive),
 - ZZS – Zienkiewicz-Zhu improved stress (ZZS and ERR are mutually exclusive),
- Contact output
 - Name,
 - Frequency – results of every N-th increment will be saved,
 - 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.

5.5.4 Boundary condition

The Boundary Conditions menu is shown in Figure 21.

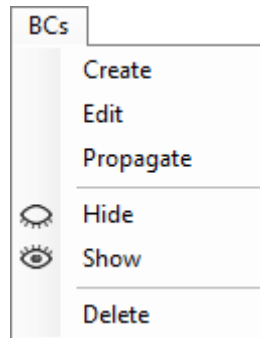


Figure 21: Boundary Conditions menu

It contains the following options:

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

The following boundary condition types and settings are available:

- Fixed
 - o Name,
 - o Region,
 - o Color.
- Displacement/Rotation
 - o Name,
 - o Region,
 - o 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,
 - o Color.
- Submodel
 - o Name,
 - o Step number – global model step number from which to read the displacements,
 - o Region,
 - o 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,
 - 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 – the displacement/rotation from the previous step is fixed in this step,
- 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 for various DOFs in the same boundary condition can not be used simultaneously.

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

- Unconstrained,
- From global model.

5.5.5 Load

The Load menu is shown in Figure 22.

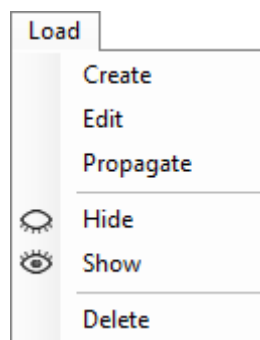


Figure 22: Load menu

It contains the following options:

- Create,
- Edit,
- Propagate,
- Hide/Show,

- Delete.

The types of loads and their options are listed below:

- Concentrated force
 - o Name,
 - o Region,
 - o 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,
 - o Magnitude – resultant force (it's calculated automatically when the components of the force are specified),
 - o Color.
- Moment
 - o Name,
 - o Region,
 - o Moment components – components of the moment per node
 - M1 – moment in the X axis,
 - M2 – moment in the Y axis,
 - M3 – moment in the Z axis,
 - o Magnitude – resultant moment (it's calculated automatically when the components of the moment are specified),
 - o Color.
- Pressure
 - o Name,
 - o Region,
 - o Magnitude – value of the pressure load,
 - o Color.
- Surface traction
 - o Name,
 - o Region,
 - o Force components
 - F1 – force in the X axis,
 - F2 – force in the Y axis,
 - F3 – force in the Z axis
 - o Magnitude – resultant force (it's calculated automatically when the components of the force are specified),
 - o Color.
- Normal shell edge load
 - o Name,
 - o Region,
 - o Magnitude – value of the shell edge load,
 - o 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),
- Color.
- Centrifugal load
 - Name,
 - Region,
 - Axis point – coordinates of the axis point
 - X,
 - Y,
 - Z,
 - Axis direction
 - 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,
 - Rotational speed – rotational speed around the axis defined by a point and a direction,
 - Color.
- Pre-tension
 - Name,
 - Type,
 - Region,
 - Auto compute: Yes/No – automatic computation of the pre-tension direction,
 - Magnitude – force magnitude for the pretension load,
 - Color.
- Concentrated flux
 - Name,
 - Add flux: Yes/No – add flux to the previously defined fluxes,
 - Region,
 - Flux – value of the flux per node,
 - Color.
- Surface flux
 - Name,
 - Region,
 - Flux magnitude per surface area,
 - Color.
- Body flux
 - Name,
 - Region,

- Flux magnitude per volume,
- Color.
- Convective film
 - Name,
 - Region,
 - Sink temperature,
 - Convective 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,
 - Color.

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.

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).

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, ...).

5.5.6 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 or buckle step
 - Name,
 - Define temperature: By value,
 - Region,
 - Temperature,
 - Define temperature: From file
 - Results file,
 - Step number.

5.6 Analysis Menu

The Analysis menu is shown in Figure 23.

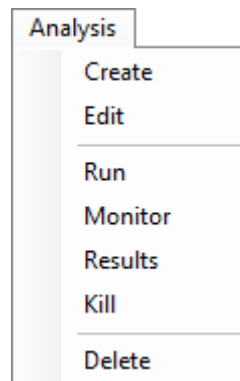


Figure 23: Analysis menu

It contains the following options:

- Create,
- Edit,
- Run – submit the analysis,
- Monitor – open the window that displays solver messages,
- Results – open results of the completed analysis,
- Kill – stop the analysis before it finishes,
- Delete.

6 Results options

The results tree consists of the following containers:

- Mesh
 - Parts,
- Results
 - Field outputs,
 - History outputs.

6.1 Results Menu

The results menu has two options (Figure 24).

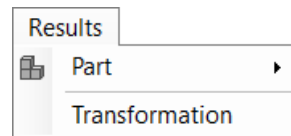


Figure 24: Results menu

It contains the following options:

- Part
 - Edit,
 - Hide/Show/Show only,
 - Set Transparency,
 - Color Contours off,
 - Color Contours on,
 - Delete.
- Transformation – a mirror or a pattern can be applied to the results for visualization purposes (use the arrow 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.

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

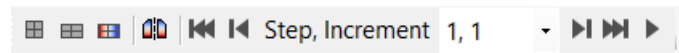


Figure 25: Results toolbar

These options include, from the left:

- Undeformed,
- Deformed,
- Deformed with color contours,
- Transformation,
- First increment,
- Previous increment,
- Drop-down list of increments,
- Next increment,
- Last increment,
- Animate.

There are several settings for animating the results. They are shown in Figure 26.

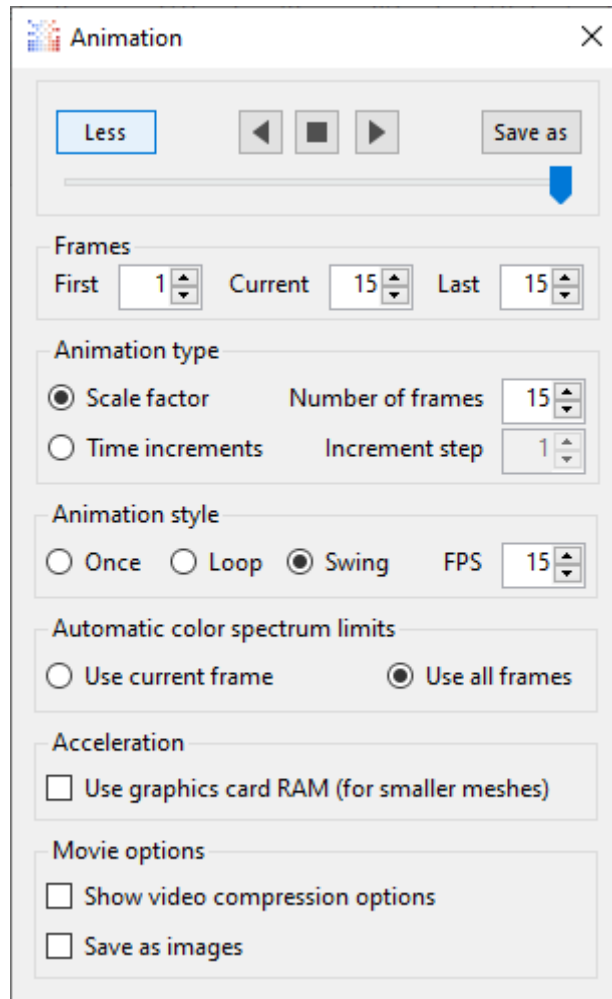


Figure 26: Animation window

Point/Node query tool has additional functionality in the Results module – it allows the user to check the value of the currently shown output variable at the selected node.