

# SimLab 2017.2.2 Release Notes

## Highlights

---

### ▼ System

- Undo supported for following tools
  - Geometry > Face > Create > From Edges
  - Geometry > Body > Create > Extrude
  - Geometry > Body > Replace instances
  - Mesh > Verify > Qcheck > Clean up method: Modify element / Move node
  - Assembly > Connect > Separate
  - Trials > Simplify clamps
  - Advanced > Pipes, tubes and wires > Pipes > Connect clips

### ▼ Import Enhancements

File > Import > CAD

CATIA Direct (New)

- CATIA R27 reading supported for Direct (new) import.
- Direct option (Approach to import CATIA part with CAPRI feature) is removed.
- Direct (new) option is renamed as Direct. This approach will allow to import the CATIA part directly without having CATIA installation.

**Note:** To import the design parameters, colors and publication, we need to configure the CATIA with SimLab.

- Option added to set part name as body name for the CATIA part contains one body. This option is added for CATIA Direct(New) method.

Step

- Support added to import step files through spatial libraries.
- By using this approach, step files from Space Claim will be imported without any errors.
- Support added to read part name instead of body name when environmental variable is set as `SL_STEP_READ_PART_NAME=TRUE`

## File > Import > Solver Input File

### Abaqus

- Added support to import Hex 27 elements.

### OptiStruct

- Added support to import the Dynamic Pressure load from Non-Linear Analysis type.

### ADVENTURECluster

- Supported to import linear Pyramid elements.

### AcuSolve

- Supported to import multiplier function and reference frame global commands.

### Permas

- Added support to import Hex 27 elements.

### Radioss

- Added support to import the nodes, elements and sets.

## File > Import > Results

### H3D

- Reading support is added for RBE, Bar and Spring elements.

## ▼ Graphics/User Interface

- Loads and Constraints Browser
  - Added a support to swap the master and slave surfaces in contact manager.
  - Option "Start Animation" is added. Wall boundary particles can be moved during a simulation with a prescribed motion. This option will animate the visualization of motion.
  - Support added to review nodes and solid elements for N2S contacts in contact manager. Distinct colors are used to represent master and slave elements.
  - Added support to include or exclude additional elements on the contact face by using modify contact tool in contact manager.
- Property browser
  - Support added to display the following the property browser
    - Material ID
    - Property ID
    - Entity type (Shell / Solid / Bar / Rigid bar)

- Field table support added for Element Id vs Value format.
- Added a support to list the standard materials like Steel, Aluminum, Cast Iron, Air and Water by default. These materials will be loaded every time when we open a new application/database.
- In addition to this, user defined materials stored in material database will also be listed by default. To activate this tool, turn on "Material Database" toggle in File > Preferences > Analysis.
- Results browser
  - Added a support to create response by using cylinder, sphere and points (XYZ).
- Inspect > Area
  - Improvements done in calculating area and volume. In case of Tri 6 mesh with mid nodes projected, area and volume calculation will be done based on the projected mid nodes.
- Inspect > Mass properties

Added a new tool to calculate the following parameters from the selected bodies like Solid, Shell, Bar and RBE with mass. The following parameters can also be found using this tool

  - Volume
  - Mass
  - Center of Gravity
  - Mass Moments of Inertia with respect to centroid
  - Principal Moments of Inertia with reference to centroid
- Added a new selection filter "Solid element". By turning on this filter and solid elements can be selected directly. Also, picking type (Rectangle, Polygon and Intersection) is supported to select bunch of solid elements.
- Enhanced user defined Keyboard shortcuts support for all workbench.
  - Import / Export option is supported to import and export the keyboard shortcuts in (\*.xml) format.
  - "S" key will open the keyboard shortcuts dialog
  - "Current keys" option will display the tools/options which are assigned with shortcuts
  - "Print keys" will enable to print the shortcut keys. It will writes currentkeys.csv and currentkeys.txt file in .altair folder.  
Eg: C:\Users\\.altair\SimLab\_V2017.2.2\simlab
- Added search in top right corner of the graphics screen. This will be very helpful to locate the tool easily. Use "ctrl + F" shortcut key to access the search option.

- Measure bounding box tool is added. It is used to measure the dimensions of bounding box of all selected bodies.
- Body group input support added for the tools which contain body input. For example, Project to CAD, Boolean, Transfer groups etc. This will help during automation.
- Value remembrance support added for following tools
  - File > Import > CAD > Parasolid
  - Model browser > Right click > Search by ID
  - Mesh > Modify > Layers
  - Assembly > Modify > Align
  - FEM > Node > Move
- Body right click > Lock and unlock
  - Display and Display only is renamed as lock and unlock respectively.
  - New option Invert is added. This option will display the bodies which are not present in the lock list.
  - Enhancement done in lock option to add more bodies in addition to the locked bodies.
- Support added to display the free and non-manifold edges in wire frame mode. This option will be displayed in right click menu only when free and non-manifold edges toggle in Render mode > Edge display is turned on.
- Point mass marker is changed. Rotation performance was improved with the new marker.
- Supported sets input for non-line edit dialog options.
- Marker drawing support added for excitation load.
- Import flux data
  - Support added to Import Flux Bulk Data in Loads and Constraint Browser.
  - Solver setting will be created by linking the imported bulk data file as include file.
  - For every DLOAD, Model Frequency subcases will be created with solution parameter, where corresponding DLOAD ID and FREQ ID will be referred.
  - Along with this a Normal mode subcase will be created and that too linked with all MFR subcase, which is used to run the Model Frequency Response Analysis.
  - Node which have load information in Bulk file will be collected and created as a node group.
- Supported to reposition or edit the result title and legend bar.

## ▼ Geometry

Edge > Create > From Edges

- Enhancement done such that output face will be created based on the element type present in the input edges.

Face > Create > Cylinder

- Support added to create cylinder with tri 6 elements.

Face > Replace Face

- Improvements done such that, a group "Replace\_New\_Faces" is created after replace face operation. Replaced face is added in that group. This helps in automation.

Face > Modify > Break

- Enhancement done to select bodies with quad elements as input.
- The intersecting elements are converted to tri elements after face break.
- Converted Tri elements are added to the group.

Body > Create

- Added support to create Tri 6 elements for cylinder, cone and sphere.

Body > Create > Extrude > Up to faces

- Added an option "Connect to limit faces". Turn on this toggle will automatically connect the extruded face with the limit face.

Body > Simplify Sensor/Clip

- Enhanced this tool helps in simplifying geometry with too many features. Like clips, sensors etc.
- Simplified representation of the original part is created based on the selected input faces.

Body > Replace instance

- New tool is added to identify all the instances in the assembly and replace with simplified body.

## ▼ Mesh

2D create > Mesh control

- Improvements done in Remove logo mesh control such that, during surface mesh it will remove the logos and as well it will re-mesh the base face. Previously, remeshing was not applied for the base face.

#### 2D create > Surface mesh

- Support added to transfer the face and edge attributes from CAD to FEM body when imprint meshing is turned on in advanced options. To get this add the following environment variable.

`SL_IMPRINT_MESHING_ATTRIBUTE_TRANSFER=TRUE`

- Improvements were done in imprint meshing such that the local mesh size is respected as intended.

#### 2D create > Re-mesh

- Added a support to re-mesh the faces from solid mesh body directly.

**Note:** Option “Number of Tet-Element layers considered during local mesh modification” is added in File > Preferences > Mesh, to control the number of Tet layers when local mesh modification is done. This option is currently supported only for local re-mesh and layered elements.

- Improved the performance of the re-mesh when the input face is selected from the surface mesh body. Earlier, if large number of sub-assemblies present in the model, remeshing was slow.

#### Verify > Quality

- Added clean up support for Quad warpage quality by using move node clean up method.
- Support added to specify the distance where the movement of node will be controlled within the given distance. This will avoid the node deviation from the geometry. Use the following environment variable to specify the distance.

`SL_MOVE_DISTANCE = 2`

#### Verify > 2D quality check

- Support added to create mesh quality specification by listing multiple mesh qualities. Mesh quality spec creation is added under mesh control browser right click.
- Created mesh quality spec is listed under 2D quality check tool. User can select the mesh quality spec and display the failed elements.
- Support added to display the failed element for multiple mesh quality with different color representation. After displaying the failed elements for multiple mesh quality, a legend will be displayed in the screen.

**Note:** This feature is a Beta release. Further improvement will be done in the coming versions.

#### Modify > By Element > Swap

- Enhanced swap element tool such that in case of flipped element created during swap, it will automatically resolve without creating free edges.

#### Modify > Fill hole

- Enhancement made in edge option. The selected input edges from different bodies, then the filled face will be added to the corresponding body.

#### Modify > Grid mesh

- Enhanced grid mesh tool to select faces from different bodies.

**Note:** Selected faces from different bodies should have node connectivity

#### Modify > Layers

- Edge input support added to change the number of layers.
- Added a support to select input faces from different bodies.

#### 3D Create > Element

- Added a support to create layered elements on the shared faces.
- Solid body input supported for layered element creation.

#### 3D Create > CFD BL

- Enhanced boundary layer tool such that, boundary layer output will be created in the same model.
- Added a support to retain the input face group in the boundary layer output.

**Known Issue:** To create volume mesh layers in a local region, layer mesh control can be assigned to select faces. This functionality is broken. We recommend users to export a gda file of the surface mesh, import in earlier version of SimLab, create the volume layer, and import back to the current database as workaround.

### Assembly

#### Connect > Join

- Added a toggle to re-mesh the shared faces automatically during join operation.
- Enhancement done to select bodies with quad elements as input.

**Note:** In Join tool, only planar join is supported for quad input

- The intersecting elements are converted to tri elements after face break.
- Converted Tri elements are added to the group.

#### Connect > Boolean

- Enhancement done to select bodies with quad elements as input.

- The intersecting elements are converted to tri elements after face break.
- Converted Tri elements are added to the group.

#### Connect > Fuse

- Improved the auto clean up on the interface to create better mesh quality while fusing cast and machined parts.

#### ▼ FEM

##### Node > Remove duplicate nodes

- Added a tool to identify the duplicate nodes in hex or point bodies and to remove duplicate nodes in point bodies.

##### Element > Modify

- Support added to convert Hex elements to Hex 27 elements.
- Support added to convert 2 Quad4 elements into 1 Quad element.

#### ▼ Analysis

##### Tools > Replace Bodies

- The tool is used to transfer the contacts, connections, loads, material and property associated with an existing body to a new body having changes in design / mesh.
- Also, this tool can be used to review the connected components of a body and the type of connection such as contact, 1D bolt.

##### Tools > Mapping

- Improved the mapping performance for "Results data + model" method.

##### Loads and constraints > Contact

- Enhanced to define set based contacts for ADVC and Permas solvers.

##### Solve > Include File Manager

- Added a tool to organize the model data into various include files. This file structuring is used while exporting solver Abaqus decks.

##### Loads and constraints > Nonstructural Mass

- Added a tool to define the mass which doesn't contribute to stiffness of the model. This is supported for Abaqus and OptiStruct solvers.



## Loads and constraints > Mass

- Added support to calculate the Mass, Center of Gravity and Mass Moment of Inertia for 1D bodies with uniform circular cross section.

## ▼ Export

### Abaqus

- Added support to export Hex 27 elements.
- Added support to define the isotropic materials for gasket behavior in Property dialog.
- Option added to export the spring element under the analysis type None.
- Supported to export the node and element sets under Part Assembly write option in export and solve dialog.

### OptiStruct

- Supported to define temperature load (Analysis > Loads and Constraints > Loads > Temperature) for Normal mode analysis.
- Supported below parameters for Gap element definition.
  - AUTO, HARD and SOFT options for KA value
  - AUTO option for KT value.
- Supported to export TEMP(LOAD,HTIME=ALL) card from Linear static load case solver setting dialog.
  - This card will be exported only when the transient heat transfer load step is referred in linear static load step and it will be exported when using couple thermal analysis type.
- Supported to write SEPARATE or COMBINED cards under PARAM, EXCEXB to control writing of Recovery Matrix in .exb file.
- Supported an option in USE\_CONTACT\_PARAMS contact type, to export the Separation type for Master and Slave surfaces.
- Pressure can be defined as a function of time in Non-Linear Analysis type.
- Bodies can be exported without property definition by setting the analysis type as None. It will export the bodies with property id as 0.
- Added an option "Termination Time" in load case solver setting dialog to write TTERM parameter in NLPARAM card for Non-Linear analysis type.
- Added support to define FASTCONT parameter card in solver setting dialog.
- Supported to export below cards.
  - CACINF3 and CACINF4 element types.
  - PFLUID and PACINF property definitions.
  - MAT10 material definition.

#### ADVENTURECluster

- Supported to export linear Pyramid elements.

#### AcuSolve

- Multiplier function and Reference frame global commands are supported.
- Added support to solve the Acusolve file directly from export and solve dialog.
- Supported to define fluid material and property definition.
- Supported to define temperature values for Inlet and Wall boundary conditions.

#### Permas

- Added support to export Hex 27 elements.
- Support added to write following cards as a separate file during Permas export and this will be included in the main file. This is supported for linear, nonlinear and coupled thermal analysis types.
  - Thread pretension
  - Contact and spring stiffness
  - NLOAD (Supported only for linear and nonlinear analysis types)
- Added support to export the Local Coordinate System with corresponding node id's, by using "Write LCS as Nodes" toggle in Export and Solve dialog.

#### Radioss

- Added support to export the nodes, elements and sets.

#### ▼ Results

- Added a support to plot the vector for layer cut section. This is supported for Acusolve results.
- Option added in results right click to display the results based on following averaging method.
  - Not average
  - Average at corner nodes
  - Average at corner and mid nodes

#### Results > Contour

- Added an option called "All Frames" in animation settings dialog. This will animate all mode shapes for the current load case. This is supported only for H3D results.

#### Distortion > Bore

- New plotting tool is integrated to visualize Bore Distortion plots. To activate this, turn ON "Enable the new graphing tool" from File > Preference > Results.
- Added a support to plot the below distortion graph types in bore distortion plot option.

- Diametral Distortion
- Vertical Cut Plot
- Added a support to export the below distortion parameters from bore distortion export option.
  - Summation of Fourier Orders (Second, Third and Fourth Order)
  - Sine and Cosine components of different Harmonic (Fourier) orders
- Added a support to define the design limit for below distortion graph types.
  - Diametral Distortion
  - Fourier Coefficients

**Note:** Currently, new plotting tool is not supported for Abaqus results.

#### ▼ Advanced

Bolt modeling > 1D bolt > Bolt thread

- Option added to specify the pitch as ratio of depth (C). By using this bolt thread can be created easily for varying depth.

Bolt modeling > 3D bolt > Create bolt

- Support added to specify the diameter (D1) of the bolt head. Earlier, it will consider the diameter from the input washer surface. This is supported for "Pattern1 using face/group" & "Pattern2 using face/group"

Mid surface > Sketch > Midline

- Added a support to create mid lines on quad elements.
- Edge group "MidLine\_1" is created with the mid line edges created.

Pipes, tubes and wires > Pipes > Connect clips

- Tool added to automatically connect the clips and pipes.

Power Train > Bearing

- In radial and roller bearing type creation, if we use solid hex and wedge as roller type then following LBC will be created automatically
  - Node sets will be created for contact nodes.
  - Local cylindrical coordinate system will be created at the Bearing Axis center.
  - All the nodes in the bearing model will be assigned to the LCS.
  - Constraints will be created at each ball center.
- Radial ball bearing
  - Option added to define element count as 6 or 88 on bearing contact surface for below listed Bearing types.

- Radial Ball Bearings
- Roller Bearings
  - Option added to create a symmetry bearing for Radial Ball Bearing type.
- Roller bearing
  - Enhanced roller bearing to create races even if the parameter  $d1 = d2$  and  $d1 > d2$ . Earlier, roller bearing will be created only if  $d1 < d2$ .
- Taper roller bearing
  - New bearing profile "Taper roller bearing" is added.

#### Weld

- New weld tool is added to create weld beads with Triangular or ellipsoidal shape. This tool provides option to create Round or flat ends and handles gaps between bodies as well.
- Weld ribbon is consolidated and some of the options are combined with the new weld tool added.

#### Weld > Weld Bead

- Enhanced weld bead creation to define  $\theta_1$  is less than 90 and  $\theta_2$  is greater than 90 which will create an inward spline cross sectional weld bead.
- Added a support to create flat end beads. Turn off the cap length toggle will create flat ends on the weld whereas turn on this toggle will create weld with end caps.

#### ▼ Trials

##### Simplify clamps

- Multiple body input support added for simplify clips/clamps tool.

#### ▼ Scripting

- Added "getAllRootModelNames" utility function to get the root model name.
- Script support added for material export.
- Improvements done in Linux such that, SimLab splash screen will not pop up during script execution in batch mode. This was now avoided by using `-nographics`. Earlier, we have to add an additional argument `-ns` to avoid splash screen. With this both Windows and Linux platform will avoid splash screen by using `-nographics`.

### ▼ Results convergence

- Added a tool to study the results convergence. The process which is captured in script can be run for n iterations to do the results convergence study. The mesh size will be reduced automatically in the regions where the strain energy error density is maximum for each iteration. The convergence process will be stopped if the change in response value is within allowable value.
- This tool is available under response right click > Mesh convergence.  
**Note:** This feature is a Beta release. Further improvement will be done in the coming versions.