

Altair SimLab 2019.1 Release Notes

Highlights

▼ Supported CAD

- Parasolid (Version 30.0)
- Step
- CATIA (Upto CATIA V5 R27)
- ProE (Wildfire 3.0, 4.0, 5.0, Creo 2.0, 3.0, 4.0)
- JT
- UG NX (Upto UG NX 12.0)
- SolidWorks
- Inventor

▼ System

- New Solutions approach supported to perform model setup for OptiStruct, AcuSolve, RADIOSS, Molding and Flux interfaces.
 - Loads will be displayed in the Analysis ribbon based on the analysis type selected during create solution.
 - Once the solution is created, the boundary conditions will not be listed in Loads and Constraints browser.
 - The AcuSolve database from version prior to SimLab V2019.1 can be exported using Analysis > Export option. From V2019.1 Solution should be defined to export in AcuSolve format.

▼ CAD Import / Export

File > Import > CAD

Inventor

- Support added to import inventor CAD models.

Parasolid

- Support added to import General Bodies.

- General bodies are non-manifold bodies or mixed body type (Combination of Solid / sheet / wire).

CATIA

- Added value remembrance and drag and drop support.

Chordal Deviation

- Supported chordal deviation-based tessellation for Parasolid, CATIA, SolidWorks, Step through spatial and Inventor models. By reducing the chordal deviation factor, geometry capturing will be finer.

File > Import > CAD through translation

- Support added to import Creo4.0 and CATIA R28 models.

File > Export > CAD

- Support added to export FE bodies as Parasolid files.
Note: This behavior is supported only for Windows.
- Support added to export wire and general bodies.

▼ Solver Input File Import

File > Import > Solver Input File

Abaqus

- Support added to import temperature dependent thermal expansion and thermal conductivity material definitions.

AcuSolve

- Added support to import orthotropic material definitions.

Nastran

- Support added to import CORD3R coordinate definition.

OptiStruct

- Support added to import CORD3R coordinate definition.
- Support added to import non-linear result monitor cards (NLMON) for Non-Linear analysis.
- Support added to read the convection load.
- Added support to import fluid grid definition.

Permas

- Support added to import the BIF file with Spring and RBE definitions.

RADIOSS

- Added support to import following definitions.
 - Contact:
 - TYPE24
 - Materials:
 - Elastic (/MAT/ELAST)
 - Elastic Plastic (/MAT/PLAS_TAB)
 - Properties:
 - Solid (/PROP/SOLID)
 - Shell (/PROP/SHELL)
 - Boundary conditions:
 - Fixed Constraint (/BCS)
 - Enforced Constraint (/IMPDISP)
 - Enforced Velocity (/IMPVEL)
 - Initial Velocity (/INIVEL)
 - Loads:
 - Pressure (/PLOAD)
 - Body Force (/GRAV)

LsDyna

- Support added to import shell element (TRI3 and QUAD4).
- Support added to import Ls-Dyna file with *.key extension.

Fluent

- Added option to import the fluent wall definition as shell bodies or solid element face sets.

▼ Graphics/User Interface

File > Preference > Applications

- Option added for the following operations
 - Middle mouse click to switch from one item to another
 - Spacebar button is used to select Apply
 - Enter button is used to select Ok

File > Preference > Applications > Keyboard Shortcuts

- Shortcut key "M" is used for Move tool.
- Support added to specify user defined shortcut for following tools
 - Face right click > Identical features
 - All tools under Inspect menu

File > Preference > Display > Color settings

- Option added to set the face edge color individually for CAD and FEM.

File > Preference > Display > Scale UI

- Toggle added to scale GUI in high resolution displays.

File > Preferences > Geometry > Chordal Deviation

- Supported chordal deviation-based tessellation for Parasolid, CATIA, SolidWorks, Step through spatial and Inventor models. By reducing the chordal deviation factor, geometry capturing will be finer.

File > Preference > System > Mouse Settings

- Mouse setting option is supported for Abaqus CAE, Ansys Workbench and SolidWorks.

File > Save as

- Support added to save the script in Python format.

File > Help > Function Usage Log

- Icons for each tool is added for easy identification of the tool.
- Dialog resize supported.
- Reset button is added to clear usage log.

View > Review Mode

- In CAD wire frame mode, pick through wire frame is activated. This helps to pick the mesh face directly.

Render Mode > Color > Feature

- Option added to display the torus. Also, added an option to display the faces which does not belong to any of the feature type.

Face Right Click > Color

- Remove color option added in set face color tool to remove the face color.

Body / Face Right Click > Select Feature

- Support added to identify the fillets based on arc length.

Face Right click > Show average results

- Support added to show average results for the selected faces. This is added only for AcuSolve results.

Face Right Click > Color

- Remove color option added in set face color tool to remove the face color.

Body / Face Right Click > Select Feature

- Support added to identify the fillets based on arc length.

Face Right click > Show average results

- Support added to show average results for the selected faces. This is added only for AcuSolve results.
- Support added to delete the nodes from point bodies.
- Option added in GUI to change the plot tolerance for field table contour plot.
- Supported Fillet selection filter for CAD.

Assembly Browser > Model Right click > Save Geometry

- Option added to save the CAD geometry. This will be mainly useful, if we regenerate the design parameters then we can save the updated CAD geometry.

Assembly Browser > Model Right click > Import Associated Mesh File

- Option added to import the associated mesh file for the CAD assembly. In case of assembly file, each part will be meshed by different users. Once the meshing is done for entire assembly, all the parts should be placed in the CAD assembly location. Then, by using this option, user can import the mesh file with assembly structure.

Assembly Browser > Sub-assembly/Body Right Click > Export in Original Coordinate

- Option added to export the FE Bodies in original coordinate.

Group Browser > Property

- Support added to display the group color in group browser.

Property Browser > Table Right Click > Create Table

- Redesigned the field table dialog by adding below enhancement in table creation mode.
 - Table type will be listed in tree view instead of combo box options.
 - Controls like insert row and delete row are moved to table right click.
 - Support added to display 500 rows in each page.
 - Support added to export table and plot contour from property browser right click.
 - Support added to plot graph for Field Table data.
 - Note: This plot graph for Field Table data can be accessed only by turning on 'Enable new graphing tool' toggle in File > Preference > Results.

Loads and Constraints Browser > Right click

- Support added to plot the contour for initial stress conditions using Show Contour option in right click of initial stress.

Loads and Constraints Browser > Simulation setup > Motion > nFX Animation

- Support added to animate in planetary motion.

Solution Browser

- Support added to include coordinates, contact, mass, set and connectors in Solutions.
- OptiStruct Topology and Free shape optimization setups are supported in Solution approach.

- Added support to define export settings for Solutions, right click "Create Export Options".

Results Browser > Results > Right click > Create Response

- Added support to create response support for Normal mode results.

Results GUI

- Improvements done to visualize the contour based on the user specified range in the legend bar. The values can be edited directly in legend bar by clicking on it.
- Top and bottom sliders are added in result legend to set the contour transparency.
- Supported an option in legend right click to set inside or outside range in transparent mode.
- Option "Restore Legend" and "Contour Options" added next to legend bar.
- Black lines are added in the legend bar for better visualization.
- Slider movement is restricted during animation
- Support added to display the results details in the graphics screen. Also, user can directly change the load cases, results component and plot type in graphics screen. It is supported only for OptiStruct results.

Note: To activate this new result panel, turn ON "Enable result panel" from File > Preferences > Results.

Move Tool

- Supported solid element filter for Normal option in Move Node.
- Copy option is supported for Move Node.
- Value remembrance support added for move tool.

Inspect > Nodes > By Region

- Support added to display nodes by defining sphere and box primitives.

Define box

- Following options are added to define the box
 - Rectangle selection
 - Left click on the mouse and drag will create a box.
 - 2 Planar faces
 - Select 2 parallel or 2 perpendicular planar faces will create a box

Note: After creating the box, user can adjust the box size and position the box by using move tool. Also, scaling option supported.

▼ Geometry

Face > Break

- Enhanced the tool to resolve the intersection within single face. Earlier, it will resolve the intersection across the faces.
- Break intersecting face tool is moved to the main ribbon. Earlier, this tool is under Face > Modify.

Body > Create > Cylinder

- Support added to create cylinder by using solid elements types (wedge 6, Tet 4 and Tet 10).

Body > Break

- Finite plane body break support added for Parasolid bodies.

Body > Show end faces

- Show end faces option added to identify end faces for given input bodies.
- The identified end faces will be added in a group.

Body > Create fluid domain

- Tool added to extract the fluid domain as a separate body and fill the inlet and outlet automatically. Extracted fluid domain body will be shared with the input body.

Transform > Translate

- Support added to translate face in a direction for user specified distance.

Transform > Match nodes

- Enhanced the display of movable body nodes and the fixed body nodes in different color.

Transform > Using LCS

- Support added to transform bodies from global to LCS (or) from one LCS to another LCS.

▼ Mesh

2D Create > Mesh controls > Defeature > Holes

- This is used to remove the holes during surface mesh.
 - Face input - Selected holes or slots will be defeatured.
 - Body Input - All radius holes will be removed or based on user defined radius range holes will be removed.

2D Create > Mesh controls > Defeature > Logo and details

- Remove Logo mesh control is renamed to "Logo and Details" and moved under Defeature grouping.

2D Create > Surface mesh

- Enhanced the bar body meshing to equivalence the nodes automatically on the interface region.

2D Create > Surface mesh > Quad mesh > HM Quad

- Option "Project to selected entities" is added to project the output quad mesh on the input CAD.

2D Create > Re-mesh

- Improvements done in local re-mesh to avoid uneven mesh seeding on the edges.
- Enhanced the tool to update the Permas contact springs while remeshing the contacts faces.

2D Create > Re-mesh > Quad mesh > HM Quad

- Option "Project to selected entities" is added to project the output quad mesh on the input CAD.

2D Create > Mid Mesh

- Enhanced mid mesh tool will consider body mesh control as input. Based on the element type specified in the body mesh control, it will create the output in mid mesh.
- Input body with shared faces is supported for mid mesh.
- Support added to give variable thickness body as input.

Verify > Quality

- Support added to retain the % failure element count and total element count column. This will help to avoid turn on these 2 columns every time during quality check.
- Undo support added for Tet clean up.

Verify > Quality check (New)

- Shortcut key "Q" is added to display the failed elements periodically based on defined quality criteria.
Note: Shortcut key will be activated only after compute and display the failed elements.
- Failed element display is improved.
- Undo support added for Tet clean up.

Modify > Fill hole

- Added a toggle "Retain all loops" to close all selected edge loops.
- Each loop will be created in separate bodies by turning ON the toggle "Fill alternate loops as a separate body".

Modify > By elements > Split

- Enhanced the tool to split the elements with better quality.
- Removed 'Tolerance' option from this tool.
- Message will be posted if zero area elements are created after the split.

Modify > Grid mesh

- Support added to create grid mesh for three sided faces. Mesh seeding can be changed interactively.
- Mesh transition is handled while changing the seed numbers in the mesh
Note: Left click for decreasing the seeding, Right click for increasing the seeding and middle click to enter the value of seeding.
- Option added to project the input face to CAD face during grid mesh.

Modify > Layers

- Option added to align start point along the principal direction.

Modify > Smoother

- Improvements done to get smooth mesh flow.

Modify > Hex/Wedge > Collapse

- Support added to collapse the hex element at mid-point of selected element edge.

Modify > Hex/Wedge > Split

- New option added to split Hex/Wedge elements.

3D Create > Volume mesh > Tet

- Support added to create a group by listing the surface elements which are modified during volume mesh. **Note:** This group will be created only if “Display modified element during volume meshing” toggle is turned on in File > Preferences > Mesh
- Surface hardpoints are respected in volume mesh tet clean up.
- Face and edge based hardpoints are supported for volume mesh and tet clean up.
- After the volume mesh a node group is created for hardpoint mesh control.

3D Create > Volume mesh > Hex/Wedge

- Hex/Wedge mesh will consider body mesh control as input. Based on the element type specified in the body mesh control, it will create the volume mesh.
- Input body with shared faces is supported.
- Support added to give variable thickness body as input.
- Support added to control minimum element size and aspect ratio.

3D Create > Element

- In case of Quad face as input, support added to have a shared face between layered element body and the input body.

3D Create > CFD Mesh

- Added volume mesh parameters in CFD mesh dialog. This helps in generating volume mesh directly after generating the boundary layer.
- Support added to create boundary layer on shared faces.

3D Create > nFx particles New

- During nFX meshing, the nFX motion definition and properties will be automatically transferred to nFX particles.
- Particles for moving solids (like gears), liquid /air and container could be generated in a single click.
- Auto detection of leaks for Liquid/Gas and Container option.
- Multiple leak detection (initial version) is supported.

▼ Assembly

Connect > Boolean > Parasolid

- Support to create general bodies as Boolean output. Earlier, after Boolean operation, the output will have only single volume whereas now, the same output body will have multiple volumes.

Connect > Bodies

- Option "Convert to penetrate" is added in connect bodies tools. This helps to visualize if all the elements are fully penetrated or not and then proceed to connect the bodies.

▼ Features

Modify > Morph

- Solid body support added for transform features tool

Defeature

- Remove ribbon is renamed as Defeature

Defeature > Hole

- Cylinder filter is made as default for remove hole tool
- Enhanced this tool to defeature the slots. Also, closed partial cylinders and cones will be defeatured automatically during remove hole operation and the options are removed in the dialog

Defeature > Logo and Details

- Remove Logo tool is renamed as Logo and Details.

Defeature > Flatten

- Flatten face tool will add the adjacent face to the base face. Due to this, mesh gets distorted for some of the cases. In this version, improvements are done to create side face from the base face to get the better output
- Flatten tool is moved under Feature ribbon. Earlier, this tool is available under Geometry > Face > Modify.

Attribute > Validate

- Support added to validate the Torus and Sphere attributes in selected FEM body.

Attribute > Update

- Support added to update the attributes for Torus and Sphere.

▼ FEM

Element > Modify

- Toggle added to update the associated RBEs to the mid nodes during lower to higher order element conversion.
- Also, support added to give RBE bodies as input for element order conversion.

▼ Analysis

Loads and constraints > Contact

- Support added to define Friction Co-efficient table for OptiStruct contact under 'USE_CONTACT_PARAM' contact type.
- Support added to define below parameters for ADVK contacts.
 - Adjust to interface
 - Auto shrink
 - Use thickness
 - Search angle
 - Interface
 - Penalty scale factor
 - Critical penetration

Loads and constraints > Initial Condition

- Support added to define initial stress.

Tools > Mapping

- Enhanced the validation of input data for mapping by checking the location and size of input data with map to entities.
- Added support to perform force mapping and to create force load, by using force balance interpolation method. It will use OptiStruct in the background and distribute the forces in the reference points to the mesh without altering the force balance equilibrium.
 - It is supported for Data table and Result data + model mapping methods.
- Support added to create following loads in mapping.
 - "Initial stress condition" load in Stress mapping.
 - "Enforced constraint" load in Displacement mapping.

Loads and constraints > Connectors > Manual RBE

- Option added to create RBE by selecting dependent nodes, the independent node will be created automatically at the centroid.

Tools > Sets

- Trim option is added in create set pairs. The surface elements can be trimmed based on Slave or Slave and Master options. The created sets can be used as input for creating contacts.

Tools > Replace Bodies

- Enhanced the tool to transfer the contact springs while transferring Permas contacts.

Property > Material

- Enhanced classification and display of parameters in material dialog. Materials are now categorized as solid, polymer, fluid and nanoFluidX.

Property > Property

- Following Abaqus element types are added.
 - Improved stress visualization
 - Improved stress visualization – Hybrid

▼ **Export**

Nastran

- Support added to export CORD3R coordinate definition.

OptiStruct

- Direct Frequency Response analysis type is supported.
- Support added to define Axisymmetric elements for Shell Bodies through Property dialog.
- Support added to define Degrees of Freedom for RBE3 independent nodes through Property dialog.
- Supported to defining Bore distortion output request for static analysis. This can be defined through output request option on Solver Settings or Load Case Setting.
- Added support to export temperature dependent thermal expansion table and thermal conductivity table for Isotropic material.
- Support added to export CORD3R coordinate definition.
- Support added to define separate eigen value extraction method for Fluid for Normal Mode and Modal Frequency Response analysis types in OptiStruct solver settings.
- Support added to define non-linear result monitor cards for Non-Linear analysis type in OptiStruct solver settings.
- Support added to define fluid structure interface parameters in modal frequency response solver settings.

- Added support for below Param cards.
 - FLLWER, SHL2MEM, NLMON, PRTMGG, ENFMOTN, PRINFACC, REFPNT
- Added support to export below output request cards from OptiStruct solver settings.
 - GPFORCE
 - SPCFORCE
 - STRESS
- Support added to export contact parameters (CONTPRM) for Non-Linear Static analysis type.
- Support added to define large displacement Nonlinear static method (LGDISP) and initial load increment (DT) parameter from Non-Linear Static loadcase settings.
- New parameters are added for Lanczos (SHFACL, MAXSET & MAXTT) and AMSES (AMPFACT) normal mode extraction method.
- “Write as Non-linear steady state heat transfer analysis” is supported in Heat transfer Analysis to activate non-linear heat transfer analysis (NLHEAT).

AcuSolve

- Interface Boundary condition tool added to identify and define an interface surface.
- New tool is added to define fan component
- Added support to export orthotropic materials.
- Added support to perform data check from AcuSolve solution > Right click > Data Check.
- Support added to export the solver deck in AcuConsole format by tuning off the “Write mesh as sets” toggle in AcuSolve solution > Right click > Export Settings.

Ansys

- The default Ansys element type is modified to 180 series.

RADIOSS

- Tool added to define rigid wall surface (/RWALL).
- Added support to export following contact types, materials, Properties and Loads through solution approach.
 - Contact:
 - TYPE24
 - TYPE2
 - Materials:
 - Elastic(/MAT/ELAST)
 - Elastic Plastic(/MAT/PLAS_TAB)
 - Properties:
 - Solid(/PROP/SOLID)

- Shell(/PROP/SHELL)
- Boundary conditions:
 - Fixed Constraint (/BCS)
 - Enforced Constraint (/IMPDISP)
 - Enforced Velocity (/IMPVEL)
 - Initial Velocity (/INIVEL)
- Loads:
 - Pressure (/PLOAD)
 - Body Force (/GRAV)

Permas

- Support added to export the following in BIF format.
 - Spring definitions,
 - RBE definitions and
 - Surface sets of thread pretension.

LsDyna

- Support added to export shell elements (TRI3 and QUAD4).

Fluent

- Added support to export the solid element face sets as fluent wall definition.

FLUX

- Electrostatic analysis can be setup with following boundary conditions.
 - Electrostatic
 - Tangential Electric field
 - Electric Potential
 - Region Physics
 - Symmetry
 - Periodicity

Molding

- Molding analysis can be setup with following boundary conditions.
 - Injection
 - Vent
 - Convection
 - Fixed Constraint

- Contact
- Symmetry

nFX

- Option added to export nFX motion bodies as *.stl file and referred in motion definition.

▼ Optimization

Optimization > Response

- Following response types are added.
 - Grid point Stress
 - Gasket Pressure
 - Contact Pressure
 - Bore Distortion

▼ Results

File > Import > Results

OptiStruct

- Added support to import materials (MAT1 and MAT2) and property (PSHELL and PSOLID) from h3d results.

Results > Contour Options

- "Auto" option added to format the legend bar values (Exponential or Float) automatically.

Results > Contour > Animate > Animation Settings

- "All frames" option is added to create and display an animation sequence that starts and ends with the original position of the structure. This is supported for Abaqus and OptiStruct results.

Results > Frequency Response

- Performance improved in computing the responses for multiple excitation and response nodes by 90%.

▼ Advanced

Bolt Modelling > 1D Bolt > Bolt Head > Face Based

- Added an option 'Connect bolt edge' for both top and bottom RBE of the bolt head. Turn on this toggle will consider the slave nodes only on the bolt hole circular edge.
- Support added to create template for 1D bolt head definition which can be further used to automate 1D bolt creation.
- This template will be listed under automation object in LBC browser.
Note: This is controlled by an env `SL_CREATE_1D_BOLT_USING_TEMPLATES = TRUE`

Bolt Modelling > 1D Bolt > Bolt Thread > Face Based

- Added an option 'Connect bolt edge' for top RBE of the bolt thread. Turn on this toggle will consider the slave nodes only on the bolt hole circular edge.
- Support added to create template for 1D bolt thread definition which can be further used to automate 1D bolt creation.
- This template will be listed under automation object in LBC browser.
Note: This is controlled by an env `SL_CREATE_1D_BOLT_USING_TEMPLATES = TRUE`

Bolt Modelling > 1D Bolt > Connect

- Added an option 'RBE' to connect bolt head and thread by RBE.
- Support added to create template for connection.
- This template will be listed under automation object in LBC browser.
Note: This is controlled by an env `SL_CREATE_1D_BOLT_USING_TEMPLATES = TRUE`

Bolt Modelling > 1D Bolt > Create Bolt

- Support added to create bolt by giving template for bolt, thread and connection as input which are listed in LBC browser.
Note: This is controlled by an env `SL_CREATE_1D_BOLT_USING_TEMPLATES = TRUE`

Bolt Modelling > 1D Bolt > Import Bolt

- Support added in bolt template (*.dat) file to connect RBE between bolt head and bolt thread.

Bolt Modelling > 1D Bolt > 1D Bolt Pretension

- Multiple edge input of head and thread are supported to create bolt pretention.

Electronics > PCB Hex mesh

- Improvements done to capture cylindrical holes while carrying out PCB hex mesh.
- Supported CAD body input and below element type inputs.
 - Hex/Wedge
 - Wedge6
 - Tet4

Electronics > BGA

- New tool Ball grid array (BGA) creation is supported for thermal and drop test analysis.

Pipes, tubes and wires > Pipes > Connect Clip

- Added support to input clip face that intersecting with pipe body. Intersecting clip face will be moved internally and projected on pipe body.

Power Train > Bearing

- Added a new rolling element type "Shell Quad" for all the bearing types.
- Option added to specify element size for rollers.

Weld > With Heat Zone > Create Weld Bead

- Redesigned the weld bead creation tool by listing the bead cross section type in the drop down list.
- Renamed the "simple" type bead cross section to "using weld throat thickness"
- Added new bead cross section type "Using weld leg length" for getting weld leg length as input.

Trials > Force computation

- New tool added to compute excitation load on the stator tooth faces by calling Altair Flux in background and automatically setup modal frequency response analysis in OptiStruct.

▼ Scripting

- Enhanced the delete mesh quality script function to delete all the mesh qualities.
- Added a utility function to pause the script execution and post a message. This will be helpful to pause the script, do some manual operations and then continue.