SIEMENS

Simcenter 3D 12.0 Caveats

November 15, 2017
(update to originally released Caveats)

Proprietary and restricted rights notice

Simcenter 3D 12.0 Caveats

Simcenter supported platforms

For information about operating system platforms supported by Simcenter, see *Operating system requirements* in the NX Release Notes.

> **Note**
>
> Simcenter does not support the Mac OS X.

Pre/Post

Caveats

Universal Connections

- (PR 8993464) The software currently issues an error message when you play a recorded journal file that contains a **Bolt Connection** type of **Universal Connection** that uses `CBAR` and `RBE2` type elements. The software does not issue an error message when you play a recorded journal file that contains a **Bold Connection** that uses only `RBE2` type elements.

- (PR 8994319) The software currently issues an error message when you play a recorded journal file in which a **Bolt Connection** type of **Universal Connection** is deleted and then recreated using the **Undo** command.

Load recipes

(PR 8345475) Simcenter currently crashes with a memory access violation when you export a solution that includes loads that reference a **Load Recipe** and the software cannot access the **Load Recipe** that contains the load data. This can occur, for example, if the file that contains the load values is moved manually, or when a Simulation file is copied to a workstation that cannot access the original data source file.

Materials

- The **Test Condition Parameter** and **Endurance Limit** properties, and the **Stress-Life Data** option **Slope Field** on the **Durability** page in the **Isotropic Material** dialog box are currently unsupported. They are used by a new durability solution that will be available in a future release.

- The **Stress-Life Data** option **Field** prior to Simcenter 12 is renamed to **Fatigue Life-Stress Field** to better distinguish between the options in the current and the future durability solutions.

NX Nastran FEM Acoustics and Simcenter Acoustics BEM

- In **Indirect Vibro-Acoustic** analyses, your model cannot contain isolated structural nodes (nodes that are not attached to any elements).

- (PR 8991440) NX Nastran computes the area correction factor (K) for transfer admittance when the `AREA` field on the `ACTRAD` bulk data entry is set to 0. However, in the **Transfer Admittance** dialog box in Pre/Post, the area **Correction Factor** (K) option, which corresponds to the `AREA`

field, does not accept a value of 0.0. To work around this issue, you must write out the NX Nastran input file and edit the `ACTRAD` entry to specify a value of 0 for the `AREA` field.

- To use either the **Fast Multipole** or the **H-Matrix** solution methods (available with the Simcenter Acoustics BEM environment in an **Indirect Acoustic** analysis), you must have the Microsoft Visual Studio 2008 dll redistributable installed, which is not currently part of the standard Simcenter installation.

- (PR 8944307) In the Simcenter Acoustics BEM environment, the software does not prevent a symmetry plane from cutting through the BEM model mesh, nor does it issue a warning message. The **Fast Multipole** solver fails when the model includes a symmetry plane through the BEM model mesh.

- In the Simcenter Acoustics BEM environment, neither the **Fast Multipole** nor the **H-Matrix** solution methods currently compute acoustic power.

- (PR 7936548) In the Simcenter Acoustics BEM environment, both the **Fast Multipole** and **H-Matrix** solvers fail if the name of the solver input file contains a space. To avoid this issue, do not include spaces in the names of input files or any associated directories.

- In the Simcenter Acoustics BEM environment, Simcenter does not validate the frequency range that you specify for the **Fast Multipole** solver, which has requirements for both the maximum-allowed frequency and the minimum-allowed frequency.

- (PR 8972329) In the Simcenter Acoustics BEM environment, the **H-Matrix** solver fails on the Linux platform when you use the **Number of Threads per Process** option in the **Solver Parameters** dialog box to specify processing on multiple threads. To work around this issue, use a single-thread process.

Post Processing

- (PR 8995564) When you have a multiple viewport layout, you cannot use the **Post Processing Navigator** to create a 3D post view in a viewport that contains a 2D plot that you created with the **Scenario Setup** command. To work around this issue, first use the **Return to Model** command.

- (PR 8351231) When you plot Rotation, Angular Velocity, and Angular Acceleration in 3D Post Views with scenario-based post processing, values that are in degrees are incorrectly reported to be in radians. This will be fixed in SC12.0.0 MP1.

Abaqus environment

Currently, you cannot import degenerate hexahedral elements into the Abaqus environment. A degenerate hex element has four nodes that have the same labels, so it becomes a pyramid element. This limitation will be addressed in a future release.

ANSYS environment

- (PR 8957875) When your FEM file contains cohesive elements, the software does not properly validate `SOLID272` and `SOLID273` elements, and a warning message displays when you exit the **Physical Property Table** dialog box. If your FEM file does not contain cohesive elements, then the software correctly validates any `SOLID272` and `SOLID273` elements.

- (PR 8936597) Performance and memory issues may occur when you export a large number of table fields, especially when the same table field is referenced by many different load steps. The software may hang or export incorrect results. To workaround this issue in a transient analysis, create a table field with a single time table that covers all steps, and then reference this table field in the first step only.

Nastran environment

In the **SOL 402 Multi-Step Nonlinear Kinematics** solution:

- The **Reference Displacement (REFE)** parameter on the **Equilibrium Iteration and Convergence** page in the **Nonlinear Control Parameters** modeling object is misspelled: it actually refers to an energy value (J) as its units suggest, not a displacement length.

- (PR 8351722) If you set **Restart Computation Flag (RSUB)**=*n* in the following types of subcases, the subcase must immediately follow subcase *n*. If subcase *n* does not immediately precede the subcase that calls it, the end state of subcase *n* is not used, and no warning or error message is written to the output file.

  o   **Subcase - Normal Modes**

  o   **Subcase - Cyclic Modes**

  o   **Subcase - Axisymmetric Fourier Modes**

  o   **Subcase - Buckling**

  The **Restart Computation Flag (RSUB)** parameter is located in the **Nonlinear Control Parameters - Subcase** dialog box, on the **Restart Options** page.

- (PR 8351714) There is a known issue with damage results when the software associates a **Damage Interface** (MATDMG) material with PSOLID elements. In the ODAMGPFD datablock identification, the type of law is erroneously set to 0 instead of 1 or 2. The software issues a warning message at the end of the .f06 file that indicates that Simcenter cannot properly display the damage results.

- (PR 8351716) Nodes from CELAS elements that do not belong to structural elements have no results when you solve a **Subcase - Normal Modes** step.

- (PR 8351718) In the **Contact Parameters - Multi-Step Nonlinear Pair** dialog box (BCTPAR2), the software ignores the specified **Constant Gap Distance** (GAPVAL) value when the **Initial Penetrations/Gaps** (INIPENE) option is set to **Overridden by GAPVAL**.

- (PR 8351719) Tensor results on deformed elements may not display well in Simcenter. This problem occurs because the content of the TRMBD/TRMBU datablocks, which allow the software to orient results on deformed elements, must be corrected.

- (PR 8351720) In **Subcase - Cyclic Modes** steps, the norm of eigenvectors must be corrected when you include a **Real Eigenvalue - Lanczos** (EIGRL) modeling object and select **MASS** from the **Method for Normalizing Eigenvectors** list.

- (PR 8351721) For solid elements, the software currently uses the wrong convention to write non-diagonal components of strain tensors other than total strain. The software currently uses epsilon (ε) instead of gamma (γ).

- (PR 8351723) The software currently writes the ply thermal strain data block twice in the .op2 file for multilayer shell elements.

- (PR 8351724) Stress results may be missing from **Subcase - Normal Modes** steps if the model contains composite elements.

- (PR 8352952) In a contact analysis that uses 3D elements, when you select **SEPDIS** from the **Separation and Slide Distance** list (**Contact Result** page in the **Structural Output Requests** dialog box), the software does not correctly calculate the total contact sliding distance and reports a value of 0. Additionally, in a contact analysis that uses 2D plane strain elements, the software does not reset the total contact sliding distance value when the contact is lost.

Simcenter Multiphysics

(PR 8348161) When your solution includes a **Deactivation Set Advanced** simulation object, Pre/Post correctly skips the deactivated elements when you export the NX Nastran input file. However, Pre/Post may also skip other entities that reference the deactivated elements, such as other simulation objects and loads. When this occurs, the software issues a warning message that notes that the entities were skipped because of subset export specifications.

Samcef environment

- The following issues currently occur when you import a Samcef model in Simcenter:

  o (PR 7968311) `BORNES` bounds of analytical `.FCT` functions are ignored during the import and a default `BORNES 0 1` parameter is written in the Samcef deck file when you export the model again.

  o (PR 7969581) During the import, unused `.FCT` functions are written in the **User-Defined Text** epilog. However, they keep their original numbering, which can interfere with the functions output when you export the model again.

  o (PR 8970053) The import of `.FCT` functions with a "&" (ampersand) in their names fails.

  o (PR 8983412) When importing a multilayered 2D structural model, the **Tolerance Angle** property is not taken into account and the import fails to group composite data.

  o (PR 8974666) Eccentricities in **Axisymmetric and Other 2D Structural** models are not imported.

  o (PR 8969886) 2D plane strain quadrangular elements are not imported.

  o (PR 8969095) Coordinate systems (`.FRAME` commands) of the cylindrical type are not imported.

  o (PR 8968057) Constraints related to the non-uniform torsion in Samcef beams (`.CLM FIX COMP 8`) are not imported.

  o (PR 8967996) Beam non-structural masses (`.PHP LMAS` commands) are not imported.

- o (PR 8973539) 2D multilayered element non-structural masses (`.PHP SMAS` commands) are not imported.

- o (PR 7978558) Manual coupling conditions (`.LIA` commands) in which coupling directions (`.FRAME` commands) have been defined on both the independent and dependent nodes are not imported.

- o (PR 7977385) The import of fixed temperature loads (`.CLM TFX` commands) fails if the **Input File Units** import units system is set to Kelvin for the temperature, while the imported Samcef model temperatures refer to Celsius values.

- o (PR 8350113) Radiation conditions attached to `.PRITT` glue commands are not imported.

- o (PR 8970178) Multilayered parabolic hexahedral elements are not imported.

- o (PR 8350449) The import of a time dependent structural accelerations load fails if each component of the acceleration load refers to a different time dependent `.FCT` function.

- o Samcef commands that are not imported are written in the **User-Defined Text** epilog. These commands may refer to function ids that are imported in Simcenter, for instance because these functions are also used in imported data. However, when you export the model again, these functions may receive a different id in the Samcef deck file, so that the reference stored in the epilog may not point to any existing valid function.

- o If you import a model with 2nd degree Heterosis shells and if that model contains `.MCT` contact conditions, Heterosis extra nodes (aka `9th nodes`) that are automatically created by Bacon for the contact condition are by default imported instead of being discarded. You can remove these extra nodes by selecting the **Remove Orphan Nodes** option in the import dialog box.

- The following issues currently occur when you export a Samcef model from Simcenter:

  - o (PR 8350336) Samcef `RIGA` rigid body elements are badly exported if you specify a properties orientation CSYS.

  - o (PR 8931960) In the **Solution Step** properties of a **Thermal** analysis type, the unit of the **Maximum Temperature Variation (.SUB PRED)** property is incorrect.

  - o (PR 8952434) A composite criteria **Output Requests** of the tensor type (for instance the Samcef 6-components `9637` **Puck** criteria) are not output with the correct number of components it requires.

  - o (PR 8349379) The export of a **Convection** thermal constraint that is defined with a **4D Field** can take an unusually long time.

  - o (PR 8349784) The nodal **Force** is badly exported if you set a cylindrical coordinate system for the load.

  - o (PR 8983073) The orientation of the **Pressure** load on axisymmetric elements edges is badly exported.

- The following issues currently when you post-process Samcef results:

o (PR 7904954) When you import a Samcef axisymmetric model and its results, the axis of rotation displayed in the post view is wrong.

o (PR 8951359) The third component of the bush element results are displayed as **Moment-Z** while they must be **Force-Z**.

- Other issues:

o (PR 7744998) On Linux operating systems, the **Solution Monitor** is refreshed every 10 seconds (instead of every second on Windows platforms). You must wait during the refreshing cycle to view the information on the new tab you selected.

o (PR 8945166) When editing the representation of a large Samcef superelement, Samcef can run out of memory when checking the superelement and a *File is invalid* error message is issued.

o (PR 8962030) During a **Modal Analysis** run, the **Solution Monitor** no longer displays the effective masses table.

Simcenter Thermal/Flow, Space Systems Thermal, and Electronic Systems Cooling

Durability

- Durability static events do not recognize combined loadcases.

- Durability static events do not recognize companion results.

- The Durability solution process does not support stress-life data defined using **Slope Field** for isotropic materials.

- The Durability solution process does not support stress-life data and strain-life data defined using **Field** for orthotropic materials.

- Surface elements connected to RBE elements are ignored in Durability solution processes.

Laminate Composites

- The new woven material model is available only in the NX Nastran and Multiphysics solver environments.

- It is not possible to export laminates or layups containing plies with Samcef curing materials.

- Equivalent laminate orthotropic thermal expansion coefficients are incorrectly calculated.

- Importing .layup files created on Windows platforms my fail due to carriage return characters.

  Workaround: open the file in a text editor, and replace the 'CR-LF' characters by 'LF' characters, for example in Notepad++, change \r\n to \n.

- When you export the laminate spreadsheet report to a Microsoft Excel spreadsheet, the spreadsheet is limited to 30 tabs.

  Workaround: export to a .csv file.

- 3D inflation may produce unexpected results when global ply butt joints are defined.

- 3D extrusion does not recognize cutting faces that are defined on both sides of the extrusion surface. Only those faces in the positive normal direction are considered.

- Simcenter Laminate Composites 2D implementation of the Hill failure theory differs from NX Nastran. The conditions related to the denominator of the cross-term differ.

- The **Sandwich Inflation** option in the **Laminate 2D-to-3D Extrusion** does not work.

FE Model Correlation

Regardless of the nodal displacement coordinate systems defined in the test universal file, test shapes (dataset 55) are always imported in the global coordinate system. This behavior is consistent with .unv files created in I-DEAS. However, files from Test.Lab and other systems must have nodes and mode shapes consistently defined in the global coordinate system.

Test.Lab files (.lms)

If you want to import Test.Lab files into load recipes, or if you want to use Test.Lab results as Mode Sets or FRF Sets, you must have one of the following installed on the same computer as Simcenter:

- LMS Test.Lab 15A or 16A, 64-bit versions only

- Active Pictures, which is the read-only version of Test.Lab.

  If Active Pictures is not already installed, contact technical support (GTAC) for information on how to download it. Active Pictures does not require a license.

Shape optimization and topology optimization

- The documentation for Tosca-based shape optimization and topology optimization has been removed from the help. Topology optimization is now handled with NX Nastran SOL 200 Topology Optimization.

- All topologically optimizable elements in the design optimization area must use an isotropic material defined with a MAT1 bulk data entry. The model can include topologically optimizable elements that use other materials, but these must be excluded from the design area or frozen within the design area. Otherwise, problems might occur during the optimization process. Any elements that are not topologically optimizable may also use other materials, regardless of whether they are in the design area.

- For a SOL 200 Topology Optimization solution, if you create a checker boarding control (CHBC) manufacturing constraint, a **Checker Board Radius** box appears in the **Manufacturing Constraint** dialog box. However, this field does not operate as described in the documentation. When a radius value is omitted or set to zero or any positive value, NX Nastran automatically averages the material density values of adjacent elements, which is the default behavior that occurs if no CHBC constraint is created.

  To allow checker boarding to occur (that is, to disable the checker boarding control in NX Nastran), set the radius to a negative value.

  The **Checker Board Radius** field is equivalent to the OFF-FLAG field in the NX Nastran DMNCON bulk entry.

Flexible Pipe

- In the **Settings** dialog box:

  o   On the **Allowables** tab, the default value for the **Minimum value for contact success** option is incorrectly set to 9000%. The correct default value is 90%.

  o   On the **Database** tab, you do not need to specify a path for the **Optimization Result Database** option.

- While you can undo the creation of Flexible Pipe features, you cannot redo them.

- To access a subpanel from a tree item in any definition panel, you must double-click rather than press the **Enter** key.

- To close a subpanel of a definition panel, you must click the **OK** or **Cancel** button. You cannot press the **Enter** key to close a subpanel.

- Values expressed in coefficients per unit length are displayed with an incorrect character (mm#).

Pre/Post Documentation notes

Bounding Volume Mesh Controls

The **Bounding Volume** type of **Mesh Control** is honored only by meshes that you create with the **3D Tetrahedral Mesh** command. 2D surface meshes and meshes that you create with the **3D Swept Mesh** command do not recognize **Bounding Volume** mesh controls. Because of this, if a **Bounding Volume** mesh control is associated with a **Bounding Volume** type of selection recipe that lies very close to or intersects a surface of a 3D meshed body, the mesh may fail. In this type of situation, you can combine a **Bounding Volume** mesh control on a body with **Size on Face** or **Point** types of mesh controls on the faces of the body to improve the results.

Face from Mesh

The enhanced Face from Mesh operation that identifies and creates multiple faces can take only 2D meshes as input; it cannot take a 3D mesh as input and automatically extract the 3D element free faces. If you want to use the **Face from Mesh** command with a 3D mesh, you must first use the **Surface Mesh** command to create a suface coat mesh on the 3D mesh. You can then use the **Face from Mesh** command to generate a face from the surface coat mesh.

Motion

Caveats

The following issues currently exist in Simcenter Motion.

Road visualization

If you enable visualization for an RSM2000 file and use a small grid density (such as 10 mm), you might experience performance issues.

Model definition files

*   When solving outside of Simcenter, externally modified expressions in MDF files are not evaluated.

*   User-defined forces and gear contacts are not supported.

*   Models with a speed-sweep element do not solve.

Submechanisms

*   The following functionality does not support submechanisms:

    o   RecurDyn solver

    o   Adams/Solver

    o   JT export

    o   PLMXML import/export

o   Capture arrangements linked from a submechanism

o   Create sequence to master parts and create explosion to master parts linked from a submechanism

o   Update design position

o   Change reference set

o   Import from subassembly

o   Teamcenter 4GD

o   Partial loading

o   Referencing a flexible link or spline beam from a submechanism

- You cannot copy bushing, tire property, spline beam property, or analytical contact property parameters from a submechanism.

- (PR 8346978) Deleting an assembly constraint in the **Assembly Navigator** does not delete the associated submechanism positioner in the **Motion Navigator**. You must manually delete the positioner.

General

Plot graphs in AFU tables have a 112-character limitation for the record name.

Motion Documentation notes

Solvers

The **NX Motion** solver supports the same functionality as the Simcenter Motion Solver, with the exception of the following:

- Tires and roads

- Flexible links

- Co-simulation

- Model definition files

- Adams and RecurDyn results in the Simcenter Results Viewer

When you see a reference to the Simcenter Motion Solver in the documentation, the same reference applies to the NX Motion Solver. With the exceptions listed above, if it is supported in the Simcenter Motion Solver, it is supported in the NX Motion Solver. Likewise, if it is not supported in the Simcenter Motion Solver, it is not supported in the NX Motion solver.

The assembly algorithm in the Simcenter Motion Solver is different from the assembly algorithm in Virtual.Lab. This difference could result in different positioning of the components when you send them to the solver. Therefore, for models that are defined in a non-assembled state and which have drivers, the results of assembly analysis may be different. To resolve this difference, you should introduce enough initial conditions to guide the assembly analysis to the desired assembled solution. You can also modify the model to define it in an assembled configuration.

Models that are defined with redundant constraints and friction elements with non-zero stiction coefficients may cause over-constrained systems that the software may not be able to solve.

Update Design Position

This functionality is removed from the Simcenter 12 release.

Siemens Industry Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Suites 4301-4302, 43/F
AIA Kowloon Tower, Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

About Siemens PLM Software

Siemens PLM Software, a business unit of the Siemens Industry Automation Division, is a leading global provider of product lifecycle management (PLM) software and services with 7 million licensed seats and 71,000 customers worldwide. Headquartered in Plano, Texas, Siemens PLM Software works collaboratively with companies to deliver open solutions that help them turn more ideas into successful products. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.