What’s New

SIMULIA product 2021

Version 1.0 - 12/8/2020

What's new in SIMULIA product 2021?

Abaqus R2021x

1. Abaqus

1.1. General Enhancements and Modeling

1.1.1. Modeling enhancements in Abaqus/CAE

The following modeling enhancements are now available in Abaqus/CAE:

- You can directly import CATIA V5 files as parts on the Linux platform.
- You can specify multiple directory locations in the Abaqus **plugin_central_dir** environment file parameter rather than just a single path. The standard path separator (a semicolon on Windows platforms and a colon on Linux platforms) must separate the directories.
- For the SOLIDWORKS Associative Interface for Abaqus/CAE:
  - You can choose to export each body of a multibody part as a separate part in Abaqus/CAE.
  - You can choose to maintain all region boundaries on parts exported to Abaqus/CAE.
  - Abaqus/CAE considers the current assembly configuration in SOLIDWORKS when updating the model.
- In the composite layup editor, you can use an analytical field to specify the thickness of individual plies.

  For node-based submodeling, you can specify intersection only in the submodel boundary condition so that Abaqus ignores driven nodes found to lie outside the region of elements of the global model

1.1.2. Removing node limit for execution on a single computer node

1.1.3. Defining spatial distribution of material directions

1.2. Analysis Procedures

1.2.1. Multiple nonlinear load case analysis

A multiple nonlinear load case analysis is conceptually equivalent to executing each nonlinear load case as a stand-alone analysis, with any steps comprising the common base state repeated in each stand-alone analysis.

1.2.2. Performance improvement for element output in linear analysis procedures

1.2.3. Secondary base motion in modal transient and steady-state dynamic procedures

1.2.4. Trim line simulation for sheet metal forming in Abaqus/Standard

1.2.5. New GPU acceleration of the AMS eigensolver on Windows

The AMS eigensolver can now support GPU acceleration on Windows as well as Linux. In addition, enhanced GPU acceleration of the AMS reduction phase of the AMS eigensolver delivers improved performance for large-scale models on Linux with new NVIDIA P100/V100 GPUs.

1.2.6. Capacity improvements in Abaqus/Explicit

In this release several improvements are made in all analysis phases (analysis input preprocessor, packager, and Abaqus/Explicit) to allow running very large models. While the largest model that can be analyzed depends on the modeling features used, a complex model such as the one described below demonstrates more than doubling of the analysis capacity compared to a previous release.

1.3. Analysis Techniques

1.3.1. Adjoint sensitivities in a transient dynamic analysis

1.3.2. Enhanced quality check for substructures and generated matrices

1.3.3. Sequential reservoir modeling with predefined pore fluid pressure

1.3.4. Import enhancements

Abaqus now provides the capability to transfer nodal temperature and field variables from an Abaqus/Standard or Abaqus/Explicit analysis to an Abaqus/Standard or Abaqus/Explicit analysis when the material state is imported.

You can now transfer model data and results of elements sets or part instances multliple times from an Abaqus/Standard analysis to an Abaqus/Standard analysis.

1.3.5. Importing external fields in a sequential analysis

You can perform node-based submodeling using field import. This is a new and evolving approach to submodel analysis; therefore, functionality and performance limitations exist.

You can now import an external field to define distributions, initial conditions, and history-dependent fields (such as loads, boundary conditions, and predefined fields) in a sequential analysis. Abaqus reads the external fields from the output database (.sim) file of a previous analysis. Conservative mapping is performed automatically if the fields are evaluated at different locations or if the meshes are different.

1.3.6. Optimization enhancements in Abaqus/CAE

You can now

- Create a stress design response (SIG_SENS_MISES) in a sizing optimization task.
- Specify plane or cyclic symmetry with a nonsymmetric mesh in a shape optimization task.

1.3.7. Enhanced contact formulation for Eulerian analyses

- In some cases in an Eulerian analysis, Eulerian material may penetrate through the Lagrangian contact surface. This penetration can result in the leakage of liquid or gas through the Lagrangian contact surface. To prevent this leakage, you can activate an enhanced contact formulation to improve the contact between the Eulerian material and the Lagrangian contact surface. However, if you use the enhanced contact formulation, you cannot specify adaptive mesh refinement criteria. The enhanced contact formulation is more expensive than the default contact algorithm; therefore, you should activate the formulation only when required.

1.3.8. Modeling thermal effects using XFEM

1.3.9. Special-purpose techniques for additive manufacturing

1.3.10. Thermomechanical analysis of powder bed–type additive manufacturing processes using the pattern-based method

1.3.11. Fluid exchange through ruptured surfaces

1.3.12. System-level modeling between Abaqus and Dymola

1.4. Materials

1.4.1. Enhanced modeling of short-fiber reinforced composites

For a short-fiber reinforced composite, if the matrix material is isotropic and the fiber material is either isotropic or transversely isotropic, you can now define its elasticity directly. You specify an orientation tensor (describing the dispersion of the fiber) and the elastic modulus of the composite when the fibers are aligned.

1.4.2. User control of shell element transverse shear stiffness in conjunction with user materials

1.4.3. Improving the accuracy in a consolidation analysis for materials with low permeability

1.4.4. LaRC05 and Hosford-Coulomb damage initiation criteria

1.4.5. Metallurgical phase transformation

1.4.6. Material enhancements in Abaqus/CAE

The following enhancements for modeling materials are now available:

- The **Section Manager** features a **Material** column to easily identify the materials associated with each section.
- You can create gap conductance, gap radiation, and gap convection material behaviors.
- For the creep model, you can specify the time type (total or creep) and the Anand, Darveaux, and double power laws.
- For the cap plasticity model, you can specify the time type (total or creep) and the power and time power laws.
- For the viscous model, you can specify the time type (total or creep) and the Anand, Darveaux, double power, power, and time power laws.
- For the Drucker-Prager creep model, you can specify the time type (total or creep) and the power and time power laws.
- For the plastic model, you can scale the yield stress and include the static recovery term with the nonlinear isotropic/kinematic hardening model.
- For the gap flow model, you can select the Bingham plastic or Herschel-Bulkley type to specify how you want to define the flow parameters.
- For the user material model, you can indicate that user subroutine `VUMAT` contains the effective modulus for an Abaqus/Explicit analysis and specify the hybrid formulation for hybrid elements in an Abaqus/Standard analysis.

1.5. Elements

1.5.1. Meshed composite beam cross-section enhancements

Abaqus/Standard now provides 3-DOF warping elements to model a meshed composite beam cross-section. These elements capture the effect of in-plane warping on the stiffness properties of a composite beam element using this cross-section. The conventional 1-DOF warping elements consider only the out-of-plane warping.

With the new warping elements (WARPF2D3, WARPF2D4, WARPF2D6, and WARPF2D8), you can now use all of the existing three-dimensional linear elastic material laws; for example, to generate realistic models for all kinds of laminates in a wind turbine rotor blade. In Abaqus/CAE you can recover the full three-dimensional strain and stress in the cross-section planes given the beam element deformations from subsequent analyses.

New options allow you to reflect the influence of a precurved and pretwisted beam geometry and of an inclination of the cross-section plane normal direction with respect to the beam centerline on the beam element stiffness properties.

You can now import the generated composite beam element stiffness and mass properties into Simpack and assign them to Nonlinear SIMBEAM elements for use in subsequent monolithic FEM-MBS analyses. In a subsequent Abaqus analysis, you can assign the generated composite beam element stiffness and mass properties to beam elements only when the coupling stiffness properties are fully characterized by centroid and shear center offsets.

1.5.2. Enhancements to general beam sections

You can use a material definition to define the material properties of a general beam section and associate these properties with the section definitions available in Abaqus/Standard and Abaqus/Explicit.

1.5.3. C3D10 element distortion control

- You can now define distortion control for C3D10 elements.
- Abaqus/Explicit offers element distortion control for C3D10 elements to prevent the elements from inverting or distorting excessively. Constraints are enforced using a penalty approach, and you can control the associated distortion volume ratio.
- In addition, the element stable time increment computation for C3D10 elements is improved whether the "improved" element time estimation method is activated or deactivated. The computation is now more robust for Abaqus/Explicit simulations including C3D10 elements and variable mass scaling.

1.5.4. Enhancements to composites modeling with solid elements

You can now use wedge (triangular prism) elements with a composite solid section definition. The following wedge element types are supported: C3D6, C3D6H, C3D15, C3D15H, C3D15V, and C3D15VH. For composite wedge elements, the material layers can be stacked only in the third isoparametric coordinates, parallel to triangular faces 1 and 2 of the isoparametric reference element.

1.5.5. Linear kinematic conversion

When elements are subject to large compressive forces, negative volumes can occur and result in a fatal error if nonlinear geometric effects are considered. Now, you can use linear kinematic conversion to allow the simulation to continue subject to approximation of linear kinematics for the highly distorted elements. This feature works as a form of distortion control, and it applies to all solid elements. In addition, it is also applied to membrane elements but only if they share nodes with a solid element for which linear kinematics has been activated.

1.6. Interactions, Constraints, and Prescribed Conditions

1.6.1. Enhanced hard contact for general contact in Abaqus/Explicit

The default "hard" contact pressure-overclosure relationship for general contact in Abaqus/Explicit is enhanced to stiffen as the underlying material stiffens under compression. This enhancement often reduces penetrations when foam materials (such as low-density foams and elastomeric foams) are involved in contact. The contact pressure-overclosure curve evolves as penetration increases and remains in effect as penetration decreases. Therefore, the net work done by normal contact forces (aside from contact damping effects) is approximately zero after a complete contact cycle from initial touching to contact opening.

This enhancement has a noticeable effect after a foam stiffens by an order of magnitude or more in a contact region. Penetrations with default settings may be larger than desired even with this enhancement. Scaling the penalty stiffness by an order of magnitude or more (as discussed in Scaling default penalty stiffnesses) is recommended in such cases. However, this scaling tends to decrease the time increment size and, therefore, increase the simulation run time.

1.6.2. Defining initial conditions by importing results data from an output database file

For sequential multiphysics workflows, you must often import results from one analysis to a subsequent analysis. You can now define additional types of initial conditions by importing results data directly from an output database (`.sim`) file. If dissimilar meshes are used, the field data are mapped automatically between the two meshes. You can use either the original or the deformed configuration of the source mesh for mapping. This functionality is available only for three-dimensional continuum elements.

The following types of initial conditions are supported:

- Initial activation
- Initial values of element solution-dependent variables
- Initial values of solution-dependent variables
- Initial hardening with backstresses and with no backstresses
- Initial stress
- Initial temperature
- Initial ductile and shear damage criterion
- Initial plastic strain

1.6.3. Enhanced rotational constraint for distributing coupling

1.6.4. Evolving feature edge criteria with default angle thresholds

1.6.5. Reduced memory usage for solid erosion in Abaqus/Explicit

1.6.6. Enhancements for general contact in Abaqus/CAE

Several enhancements are available in Abaqus/CAE for defining general contact interactions.

When you edit individual contact property assignments, you can directly select materials in addition to surfaces for the contact pairs. You can similarly select materials for surface thickness assignments, surface offset assignments, and surface feature edge criteria assignments.

You can create new types of surface property assignments.
For Abaqus/Standard you can create surface beam smoothing assignments and surface vertex criteria assignments. For Abaqus/Explicit you can create crush trigger assignments and surface friction assignments.

You can activate the small-sliding tracking approach for interactions in Abaqus/Standard.

For general contact in Abaqus/Standard, you can also

- specify secondary feature edge criteria for surface property assignments and
- control the smoothness of the surface-to-surface formulation upon sliding for specific interactions and control the edge-to-edge contact formulations.

For general contact in Abaqus/Explicit, you can also

- specify secondary feature edge criteria and apply feature edge criteria statically or dynamically for surface property assignments and

- choose which sides of double-sided elements will be considered for node-to-face or Eulerian-Lagrangian contact with another surface for contact formulation.

1.6.7. Enhancements for mechanical contact properties in Abaqus/CAE

The following enhancements are now available in Abaqus/CAE for specifying geometric properties in surface contact interactions:

- You can specify the thickness that determines the contacting surfaces to be tracked.
- You can define the surface interaction model in user subroutine UINTER in an Abaqus/Standard analysis or user subroutine VUINTER or VUINTERACTION in an Abaqus/Explicit analysis.
- For a surface interaction model defined in a user subroutine, you can specify the number of state-dependent variables and the number of property values that are required.
- In an Abaqus/Standard analysis with user subroutine UINTER, you can use unsymmetric equation solution procedures.

1.7. Execution

1.7.1. Enhancements to the abaqus adams translator

1.7.2. Creating Abaqus user subroutines in individual dynamic link libraries or shared objects

1.7.3. New architecture delivers improved parallel performance of Abaqus/Explicit

1.7.4. Flexible body dynamics workflow enhancements

The following enhancements are now available for flexible body dynamics workflows:

- Improved interface for the FLEXIBLE BODY option.
- Creation of an EXCITE binary file.
- Additional command line options for the **abaqus toexcite** translator.
- Translation of matrix data from Simpack to Abaqus.
- Improved interface for the **abaqus substructurerecover** utility.

1.7.5. Enhancements for translating Nastran data to Abaqus files

The following enhancements are now available for translating Nastran data to Abaqus files using the **abaqus fromnastran** translator:

- New translation option for CSHEAR elements.
- Performance improvements.
- Thermal expansion translation for RBE2 elements.

- Negative structural damping translation.
- DMIG-to-SIM improvements.

1.8. Output and Visualization

1.8.1. Contact output enhancements

The following output variables are now available in Abaqus/Explicit for general contact analyses:

- CNAREA: nodal contact area.
- CFRICWORK: nodal frictional work.

1.8.2. Requesting the Lode angle term

1.8.3. New Abaqus/CAE plug-in to create a smaller output database

The **ODB Reducer/Builder** plug-in allows you to take portions of an Abaqus output database to create a new, smaller output database. The ability to choose among many options for data to include in a new output database makes this tool useful for a number of applications, including:

- Sharing portions of results files with downstream engineers, other teams, and external partners.
- Decreasing database load times.
- Reducing storage footprint by paring down the output database files to store only certain relevant results.

1.8.4. Selecting the element output position in Abaqus/CAE

By default, variables are output at the integration points where they are calculated. In Abaqus/CAE you can now choose to output the variables at the centroid of each element or to extrapolate the element integration point variables to the nodes of each element independently, without averaging the results from adjoining elements

1.9. User Subroutines, Utilities, Plug-ins, and the Abaqus Scripting Interface

1.9.1. Utility routine to access physical constants from Abaqus/Standard user subroutines

You can now call utility routine GETPHYSICALCONSTANT from any Abaqus/Standard user subroutine to obtain values of physical constants defined in an Abaqus/Standard analysis.

1.9.2. Scripting command changes

Abaqus makes every attempt to be backward compatible and can execute most Abaqus Scripting Interface scripts from previous releases of Abaqus. However, backward compatibility is not guaranteed beyond several releases of Abaqus, and it is recommended that you upgrade your commands to the most recent release.

fe-safe R2021x

fe-safe

1.10.GA.1 Support for 2021 Abaqus ODB files

From fe-safe 2021 onward the latest ODB support will be included with the core fe-safe installation, and no longer a technical product selection.

1.11.Third-Party Copyright Notices

- Certain portions of fe-safe contain elements subject to copyright owned by the entities listed below.

© Battlelle
© Endurica LLC
© Amec Foster Wheeler Nuclear UK Limited

- fe-safe Licensed Programs may include open source software components. Source code for these components is available if required by the license. The open source software components are grouped under the applicable licensing terms. Where required, links to common license terms are included below.

IP Asset Name	IP Asset Vaersion	Copyright Notice
Under BSD 2-Clause		
UnZip (from Info-ZIP)	2.4	Copyright © 1990~2009 Info-ZIP. All rights reserved.
Under BSD 3-Clause		
Qt Solutions	2.6	Copyright © 2014 Digia Plc and/or its subsidiary(-ies) All rights reserved.

-

Tosca R2021x

2. Tosca

2.1. New Milling Constraint for topology optimization

2.2. Clearer material distribution in combined Stress-Frequency optimizations

2.3. Faster evaluation of design response groups for Abaqus based optimizations

2.4. Support of plastic strain as design response for Topology optimization

2.5. Support of rotational symmetry in sensitivity-based shape optimizations

2.6. Optimization problem stabilization—minimizing structural volume or mass

Minimizing structural volume or mass—the mathematical optimization problem can be ill-posed leading to convergence issues. Stabilization can be activated to improve convergence behavior.

2.6.1. Upgrade to Python 3.7.1

Tosca now ships with Python 3.7.1. All user scripts are called using the upgraded Python interpreter. Most user scripts written for Python 2.7 should automatically be converted without issues to run with the new version. In case of compatibility issues, please contact support.

2.6.2. Supported solver interfaces

- Abaqus 2021
- ANSYS® v19.2
- MSC Nastran® 2017

2.6.3. Supported life solver interfaces

- fe-safe 2021
- Femfat 5.3

Isight R2021X

3. Isight

3.1. The Universal Kriging Model

Universal Kriging models are flexible because you can choose from a wide range of correlation functions to build the model; however, Kriging models can be computationally expensive. The Kriging model has its roots in the field of geostatistics—a hybrid discipline of mining, engineering, geology, mathematics, and statistics (Cressie, 1993)—and is useful in predicting temporally and spatially correlated data.

3.1.1. New Customization Script Indicator

Similar to **Prologue**/**Epilogue** script indicator icons displayed on the left and right side of the component in the sim-flow diagram, an indicator icon is now displayed on the upper side of a component to indicate the existence of the customization script in the component.

3.2. Python DOE Technique

When you are configuring the DOE component, you can now select a **Python DOE** technique in addition to existing techniques. The **Python DOE** technique provides a convenient way for you to define your own DOE algorithm using Python outside of Isight and still make use of Isight's integration and automation capabilities.

3.3. Display Error Analysis Point

You can see **Error Analysis Point** numbers on a mouse hover of any design point displayed on any graph under the **Response Fit** or **Residual** tab. When you click any design point, the corresponding **Error Analysis Point** will be highlighted in magenta color on every graph for all responses under both tabs and the corresponding row in the **Error Analysis Points** table will be highlighted.

3.4. Storing Approximation Error Analysis summary

You can store the approximation error analysis summary in the output file parameter. For more information, see Configuring Runtime Options. In addition, you can export the approximation error analysis summary to a text file.

Simpack R2021x

4. Simpack 2021

4.1. Minor Enhancements Overview

4.1.1. GUI

- Improved performance when editing model

4.1.2. Solver

- Display of total CPU time in Job table
- Dissipative/strain energy and dissipative Power for Force Elements in Eigenmode Analysis
- Configurable Solver Termination (Stop run due to Warnings - Errors -- Fatal Errors)

4.1.3. Scripting

- Improved scripting performance for modifying many subvars
- Access to calculated Flexible Marker position and Node ID

4.1.4. DoE

- Added support for HPC (Directory structure for runtime directory was changed)
- New task option for Time Integration with Measurements

4.1.5. Wizard

- Support selection of directories in parameters
- View referencing elements (from parameter) within Wizard
- Copy verified component parameterizations to local project (for further editing)
- Various Wizard Usability and Wizard Scripting enhancements

4.1.6. Mathworks interface

- Support for MATLAB / Simulink@ R2019a and R2019b
- Extend S-Function blocks by adding Signal Names to the Output Ports

4.1.7. Flexible body

- Improved Performance for Flex Body Pre-Processing
- Support for ANSYS 2019 R2 and 2020 R1
- Improved contouring performance in post-processor
- Tutorial for nonlinear reduction of twist beam axles

4.1.8. General Modeling

- Optimized Result Element 52 ‚Radiated Power' Calculation for Linear Flexible Bodies

4.1.9. Automotive

- Update TMeasy to Version 5.3.1
- Standalone model support for external references used by Ftire

4.1.10. Rail

- Modal Flexible track representations "Linear flextrack"

4.1.11. Post

- Automatic frontbackplane clipping adaption during animation
- Harmonization of Animation Page into Autosize Page
- Marker Visualization for Mesh and Field channels

4.2. Resolve alternative licensing

Up to Simpack 2021 for some features an alternative licensing mechanism was implemented

- For example contact elements were licensed with Simpack Contact (KCT) or Simpack Contact Flex (KFC)or
- Certain control elements were licensed with Simpack Control (KCO) or Simpack Automotive (KAU) or Simpack Rail (KRL)

4.3. OLicense Support

Due to still strong usage of OLicense throughout our customer base we will not seize support in 2021x, but postponed to 2022.

Simpack 2021x will be the last Simpack version to support OLicense.

Support for OLicense will be set to, deprecated' in the UI and a warning message will be printed for each solver run when using OLicense.

4.4. Visualization Improvements

- Improved Visualization of (external) CAD data
- Improved Wireframe Visualization for CGR
- Basic 3D Mesh Visualization and Result Outputs for Belt
- Example of table

4.5. Nonlinear SIMBEAM option for belt/cable modelling

A new option allows to model belts, cables (and other structures) with Nonlinear SIMBEAM in the built-in configuration.

4.6. Simpack-Abaqus co-simulation with contact between Bodies in the FEM

In a Simpack-Abaqus co-simulation scenario where FE model parts can go in- and out of contact, these parts can now be connected to the MBS by using one Simpack Body per part.

4.7. Improved traction formulation for Rail-Wheel contact in zero velocity

- Previously tangential force law is switched off below certain value of reference velocity.
- A stick-slip law is added to enable simulation at zero vehicle velocity condition.
- Typical use cases include static loadcases and start from v=0.

4.8. Supporting Interface for electric machine

- Support run-up simulations
- Support EMAG Reduced order models

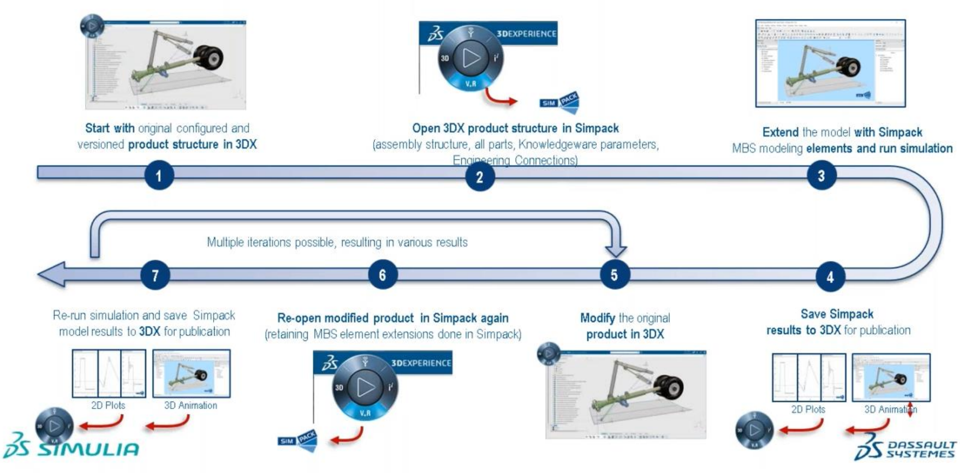
4.9. Modeling Elements Enhancements

- Consideration of Lubricant for Ball Type Bearings in Force Element 88 'Rolling Bearing'
- New Control Element 179 'EHD Piston Ring Gas Dynamics'
- EHD: Mass-conserving cavitation
- New Force Element 113 'Fluid Interface <prel>'
- Enhancements to Curve - Curve Contact (FE 45)
- Enhancements to Polygonal Contact (FE 199)
- Improve Contact Search for Force Element 198 'Cam Contact
- New Force Element 95 'Coulomb Friction prel Cmp'

4.10. Simpack POWER'BY: Model and Results

4.10.1. Connector for Simpack

You can now select a model on the 3DEXPERIENCE platform and open it in Simpack.



4.10.2. Support for Engineering Connections

Support conversion of Engineering Connections as Simpack Connections when opening a 3DEXPERIENCE Product Structure in Simpack.
Support is currently restricted to these types:

- Rigid
- Cylindrical
- Revolute
- Universal
- Planar
- Screw
- Free
- Prismatic
- Spherical

Simpack Markers will be automatically generated on the referencing Bodies. In case an engineering connection references a product (instead of a single part), the generated Simpack connection will reference the first Body in the respective submodel.

CST STUDIO SUITE R2021x

5. CST STUDIO SUITE 2021 – Release Notes

5.1. Release Highlights 2021

This list provides an overview of the important new features and improvements in the latest release of the CST Studio Suite® software. In addition, a lot of smaller changes and enhancements went into this release.

5.2. General Features

5.2.1. General | 3D Modeling

- Distributed Computing: Allow distribution of jobs to servers with inhomogeneous GPU count
- Added support to protect lumped elements in protected projects
- Wrap sheets and curves toward the faces of an arbitrary curved shape
- Project curves toward a shape along the surface normal of the target shape
- Enhanced healing and analysis of problematic shapes
- Convert Discrete Ports to Lumped Elements and vice versa
- Improved rendering performance for displaying the 3D geometry
- Support of DSLS Licensing
- Python API now compatible with Python versions 3.6 to 3.8
- New generic library format for custom library creation and distribution
- Update/extension of material library (Stacem materials, update of Preperm materials)

5.2.2. System Assembly and Modeling (SAM)

- Creating compact antenna array simulation projects by using array blocks
- New simulation project reference block to support antenna array and platform projects in simulation projects
- Fest3D blocks are supported as reference blocks for simulation projects

5.2.3. Meshing

- Automatic resolution and reporting of material overlaps for tetrahedral meshing
- Tool for full re-meshing of selected shapes to an intersection-free mesh shape
- Improved performance of mesh import and support for more keywords of Abaqus and NASTRAN files

5.2.4. Post-Processing | Results

- Report Tool: Copy/paste of report items, import of reports from other CST projects

- 2D/3D Result Plots: Separate 3D plot sheet, improved arrow plot customization
- Far field Plots: Component selection from ribbon, improved 2D plot features
- Template Based Post-processing: Copy/paste of defined result templates
- Post-processing: Improved monitor calculation after solver run

5.2.5. High Performance Computing (HPC)

- Support of multiple MPI versions
- Automated MPI-CPU setup for large projects (T)
- Improved support for many core systems

5.3. 3D EM Technology

5.3.1. High Frequency Simulation

- Djordjevic Sarkar fitting for constant loss material handling (T, TLM, F, I, A)
- Import of thermal fields for temperature depending materials from CHT Solver or Abaqus on irregular grid (T)
- Automatic absorption of higher order propagating modes (T)
- Added F-Parameters for waveguide monitors (T)
- Added plane wave to solver excitation list (T)
- Support of face lumped elements (TLM)
- Robustness improvements for cable meshing (TLM)
- Improved performance for initialization of complex models (TLM)
- Domain decomposition solver with MPI support for frequency domain calculations on tetrahedral mesh (F)
- Support of generalized port mode solver on hexahedral grids (F)
- Adaptive tetrahedral mesh refinement for the Eigenmode solver for general lossy problems (E)
- Inverse synthetic aperture radar (ISAR) analysis (A)
- Extended range of applicability for the normal material (lossless) for far field calculation (A)
- Added new preconditioner options for MLFMM (I)
- Extended range of applicability for the thin panel material (I)
- Improved setup of superimposed nearfield source (NFS) excitation during circuit co-simulation (T)
- Hybrid solver task (bi-directional) (SAM task):

- Support single plane wave excitation with bi-static RCS calculation
- Support protected projects in source domains
- Duplicate task functionality
- Limited support of mesh imports in platform domain. Transient solver to run source domains and Integral Equation solver to run platform domain. (T, I)

- Array task

  - Create full array simulation projects using more than one unit cell model
  - New option to select enclosure when creating full array simulation projects
  - Define mirrors of array element groups to define simulation zones

5.3.2. Low Frequency Simulation

- Authoring of coil segments from CAD geometries (MS)

- Added support for periodic sub-volumes (JS, MS, LT)

- Improved broadband calculation including DC point (LF FD TET)

- SAM Machine Simulation Sequence

  - Support of Synchronous Reluctance Machines
  - Improved calculation speed by reusing valid existing simulation results
  - Averaged values of flux linkage and torque as well as complimentary information
  - e.g. phase angles, units, machine parameters included in the export of the Functional Mockup Unit
  - Calculation of radial forces and export to the multi body simulation tool Simpack
  - Improved workflow and responsiveness of the user interface
  - Loss map drive scenario can be calculated based on a reduced order model
  - Export of dynamic machine characteristic / operating point function packaged in a Functional Mockup Unit
  - Skewing for d/q-drive scenario

- Temperature dependent permanent magnet model (SH)

5.3.3. Particle Simulation

- Added GPU support for E-Static PIC Solver

- Added new collision models for particle-particle interactions

- Particle Losses are now available as a solver results also before running a thermal simulation

5.3.4. Spark3D

- New High power breakdown analysis for pulsed signals

- Use of multiple SEY’s when importing the EM fields from CST Studio Suite

5.3.5. EDA Import and PCB Simulation

- Improved multi-editing of PCB elements like traces

- Improved and harmonized layout design view; view attribute manager now allows to set view options for individual components

- Report tool to document the design and the simulation results
- Load configuration of components (design variants) from ASCII file
- IR-Drop simulation now allows coupling to all available thermal solvers (THs, THt, CHT)
- SITD simulation now supports the new schematic eye diagram task including definitions of eye masks and shows an improved performance for the block arrangement on the schematic
- Support of new wizard-like DDR4 simulation workflow
- PI solver now supports component models of type Package Device
- Elevation of PI solver ports to package level
- Redesign of EDA import dialog UI with improved reporting
- New bond-wire profile editor
- Automatic creation of excitation nodes for Partial RLC solver
- Two-level PCB thermal model simplification inside/outside selection area

5.3.6. BoardCheck

- Improved performance of solver start by avoiding copying design data
- The design view together with violation results can now be exported to the report tool

5.3.7. Chip Interface

- Automatic creation of excitation nodes for Partial RLC solver

5.4. Antenna Magus

- Find by Value functionality has been improved to allow the specification of values for frequency, gain and bandwidth. Combined with the existing keyword search, a goodness factor is used to indicate the suitability of any given device.
- Constructions are more practical / realistic export assembly models. These are constructed using existing building blocks already contained in Antenna Magus.
- Variations, or helper models, are export models provided in addition to standard original export models for a given device. Variations provide the user with useful model variations, for example a horn with a lens variation, a patch with a superstrate variation or an absorber-lined cavity spiral with a cylindrical cover variation.
- Radome Library with fully parametric exportable geometries has been added
- A number of new antennas and transitions have been added to the database.

5.5. Cable | Circuit | Filter Design | Macro Models

5.5.1. Cable Simulation

- Option to show the 3D view in same perspective as the selected cross-section mesh view
- Harness nodes can now be generated from picked points or imported via text file; they can also be picked as well as snapped to picked points
- Control hide and show of individual harness elements like cable bundles or harness segments
- Improved transfer impedance models: More robust circuit models and new export of transfer impedance curves in text format
- Current monitors in cable segments probe the current of each wire as well as the common mode current
- Electric boundary is now supported not only for bi-directional simulation but also for no-coupling and uni-directional setups
- Improved accuracy of lossless circuit models for the cable simulation
- Improved random bundling: Possibility to control the strength of the randomness and to define the position of certain cables as fixed

5.5.2. Circuit Simulation

- Improved creation of Busses and display of pins and pin groups
- Improved transient simulation performance
- Extended Clone Block: Support for parametric cloning of 3D Project Blocks
- Improved Touchstone Block: Parametric access to multiple Touchstone files
- Option to use IDEM macro modeling for S-Parameter based blocks
- Unified user interface for blocks
- New Eye Diagram task with mask violation detection and size measurement
- IBIS improvements: Option for IBIS waveform truncation to avoid overclocking
- Automation: Added scripting option to determine/modify circuit connectivity
- Automation: New scripting method for automatic block placement

5.5.3. Filter Design | Macro Models

- FD3D
  - New filter design process for automated 3D filter dimensioning
  - New supported filter topologies and new filter components in component library
- Fest3D
  - New automatic CST Design Studio project generation from synthesis modules

    - Enabled direct use of parameters and mathematical expressions in dialog boxes where elements are defined
    - New direct access to synthesis tools from CST Studio Suite main window
    - Enhanced Fest3D project exporting to CST MWS project
    - Performance improvement on coupling integrals numerical computation
    - Added coaxial/ridge T-Junctions and a general waveguide bend based on CST Frequency Domain Solver.
    - Possibility to use arbitrary waveguides as ports of the CST Studio Suite element library

- IdEM

    - Improved possibilities to compare different models
    - New option to calculate and visualize modeling error for different target configurations
    - Drag-and-drop of Touchstone and project files

5.6. Multi-Physics Simulations

5.6.1. Thermal Simulation

- CHT Solver

    - Support for liquid cooling
    - Full support for transient simulations
    - Support for Distributed Computing (DC)
    - Support for k–omega (k–ω) and Spalart-Allmaras turbulence models
    - Support for initial conditions on solids and fluid domains
    - Import of IR-Drop Losses

- Import of temperature fields generated by the CHT solver (JS, LF FD, F, T)

- Performance improvements for temperature field and loss imports

Opera R2021X

6. Opera 2021

6.1. Release Highlights 2021

SIMULIA Opera 2021 is a major release which introduces significant new features into SIMULIA Opera, including:

A new way to build and analyse electrical machines in Opera-2d.

New licensing options to allow SIMULIA Opera to be run from new license bundles.

Other minor changes have been made throughout the software to enhance its functionality and reliability. Some of these are also included in this document.

6.2. Front-end

- The 2D Pre- & Post-Processor has been enhanced following its initial release in the 2020 version.
- The geometric modeling software used in the 3D Modeler has been upgraded to ACIS 2020. This provides additional interoperability with other CAD formats, and performance improvements.

6.3. 3DEXPERIENCE

- Opera 2021 POWER'BY connection has been updated. For users who have 3DEXPERIENCE platform components pre-installed an additional button for POWER'BY is enabled on the Opera Manager toolbar, which allows users to push and pull Opera files and results to the platform so that they can be shared and used by other platform team members.

6.4. Python

- Opera 2021 includes a new Python integration using the Intel Distribution for Python. This allows a much stronger link to Opera and the opportunity to include a more complete set of packages.

6.5. Licensing

- Opera 2021 allows for three different licensing methods to be used. The existing LM-X licensing system continues to be used and supported but additionally, Opera 2021 can be licensed using SIMULIA CST license bundles using either the FlexNet or DSLS system.

6.6. Opera Machines Designer

- The Opera Machines Designer is an automated tool that assists in the design of electrical machines within the 2D product. Version 2021 of the Opera Machines Designer tool includes automated model creation for a wide selection of machine types:

- Brushless Permanent Magnet machine (BLPM)
- Induction (or Asynchronous) machine (IM)
- Wound Synchronous machine (SM)
- Switched Reluctance machine (SRM)
- Synchronous Reluctance machine (SynchRel)
- Permanent Magnet External Rotor (PMexternal)