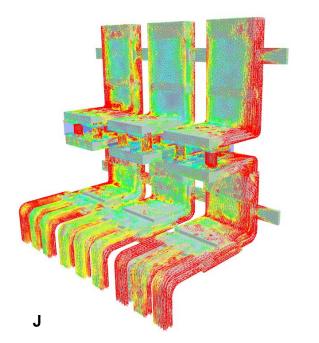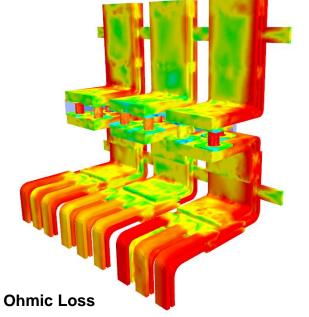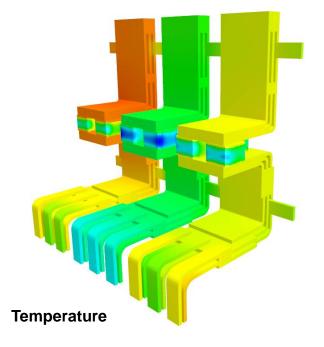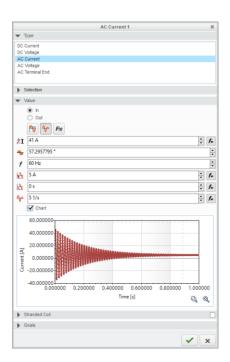# Electromagnetics in Simcenter FLOEFD

You can now simulate Alternating Current and Electromagnetics (low frequency) effects. Enabling electromagnetics allows for increased accuracy of thermal simulations by taking into account Ohmic and iron (core) losses due to AC and electromagnetic phenomena such as proximity, skin effect, etc. An electromagnetics analysis is defined together with a CFD analysis and a CFD-EM co-simulation takes place by exchanging the power (Ohmic and iron losses) and temperature fields.
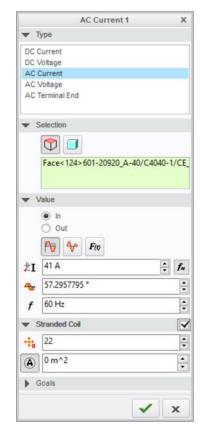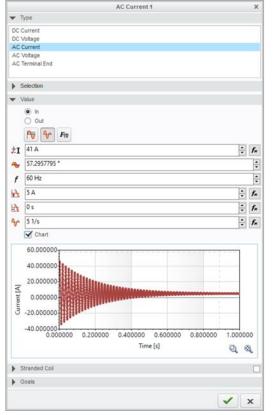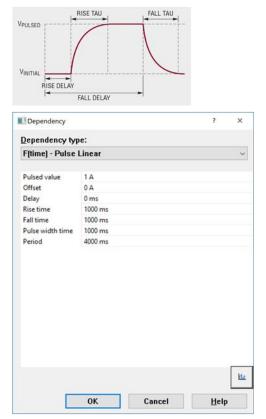

J

Ohmic Loss

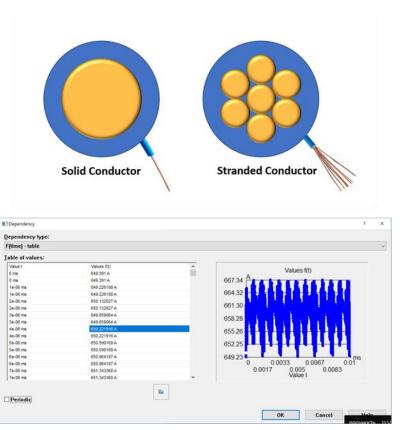Temperature

SIEMENS

Electrical Conditions and Sources

A coil can be solid or stranded (a bunch of wires). AC can be set as sinusoidal, optionally with damping and offset waveform, pulse function (linear and exponential), non-harmonic oscillations.
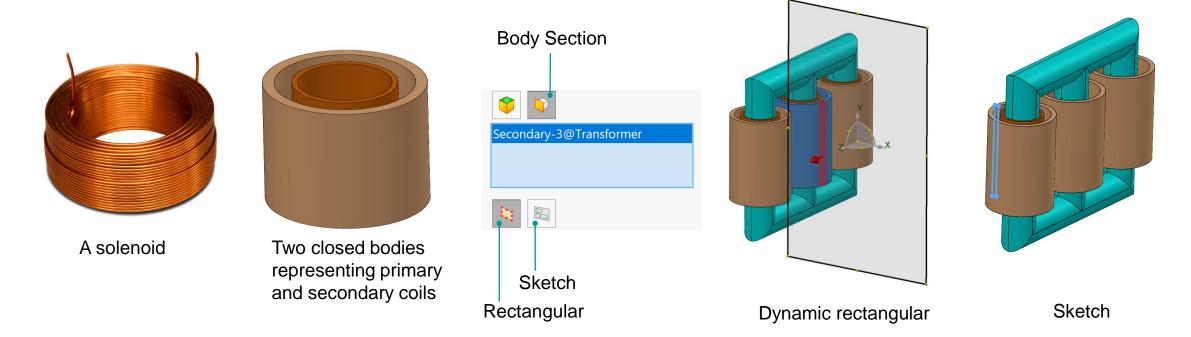
Body section

An AC conductor must be closed. A simple example of a closed conductor is a solid body representing a solenoid – a coil wound into a tightly packed helix. In that case you define an AC condition in a section of the coil body. You can define a section by either Sketch or dynamic Rectangular. Dynamic rectangular is easier while the Sketch can be used for design exploration in Parametric Study as it can be linked to a geometry and changed together with the geometry.

Body Section

Secondary-3@Transformer

Sketch

Rectangular

A solenoid

Two closed bodies representing primary and secondary coils

Dynamic rectangular

Sketch

SIEMENS

Body section

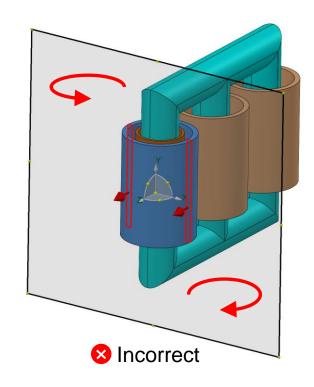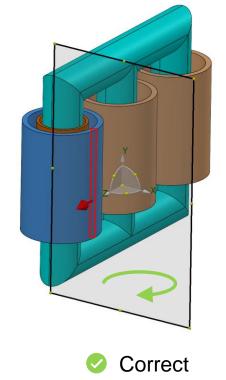Both the Sketch and Rectangle form the closed contour which intersects the selected bodies to get a body section to apply the AC condition to.

Note: Do not create two sections for the closed coil as it will lead to errors as the current direction is contradictory in the sections obtained.

❌ Incorrect

✅ Correct

SIEMENS

Terminal end

Another example of a conductor is a busbar. Unlike a solenoid body, the busbar is not a closed body, it has ends. For such a conductor you can also define an AC condition in a section, but the end of the busbar must be in contact with the boundary of the EM domain to make it "virtually" closed outside of the EM domain. In other words a busbar type conductor's end must not be in contact with the surrounding air but the EM domain boundary.

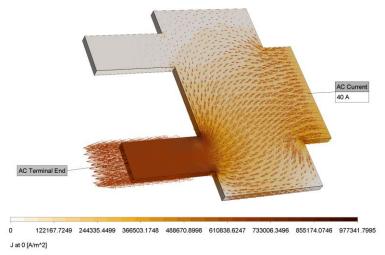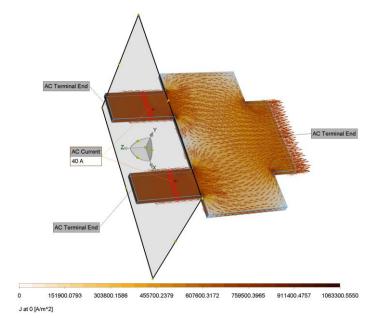❌ Incorrect   ✅ Correct   ✅ Correct

Terminal end

At least two ends must be defined for the conductor. If you define an AC condition at one end, for the other end it is enough to define an AC Terminal End condition. The AC Terminal End defines the direction of the current flow and the face where it goes out of the EM domain.

Once an AC face-based condition is defined (AC Current, AC Voltage, or AC Terminal End), Simcenter FLOEFD tries to automatically adjust the EM domain so that the AC face is in contact with EM domain boundary.
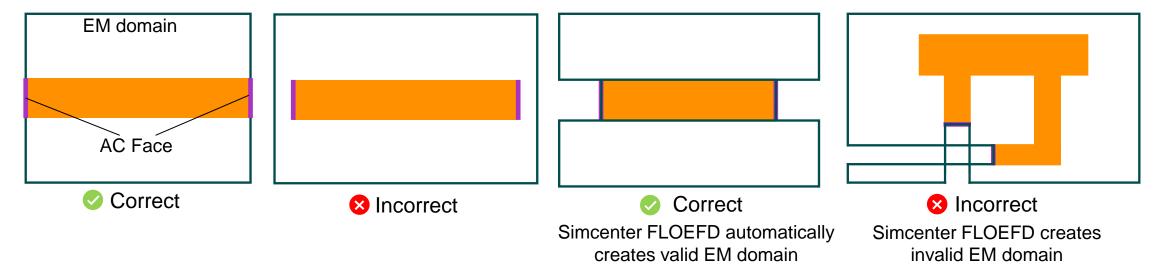
SIEMENS

Computational domain

There are two Computational Domains: CFD domain and EM domain.

A face where an AC Electrical Source is defined must not contact the air, it must lay at the boundary of the EM domain. By default the EM domain has the same size as the CFD computational domain with the exception that for every face-based AC electrical condition Simcenter FLOEFD automatically adjusts the EM computational domain so that the source face coincides with the EM domain boundary by cutting a hole in the EM domain. This must be taken into account when creating an AC face-based condition of a complex conductor with Automatic EM Domain as the "holes" must not intersect.

✅ Correct

❌ Incorrect

✅ Correct
Simcenter FLOEFD automatically creates valid EM domain

❌ Incorrect
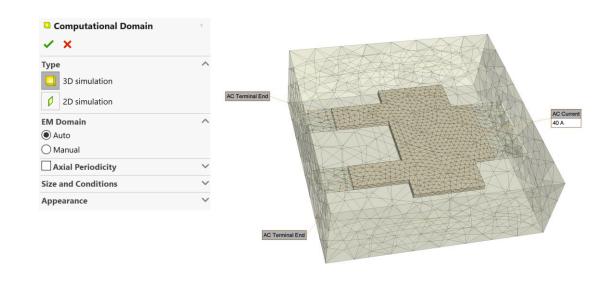Simcenter FLOEFD creates invalid EM domain

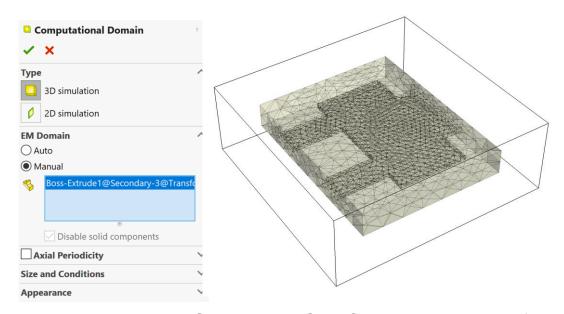SIEMENS

Computational domain

In case an automatic EM domain is not possible you can create a manual domain to satisfy the requirement to have AC faces coinciding with EM domain boundaries. For that you create a part (so that the part boundaries coincide with the AC faces) and select it in the Computational Domain dialog.
You can also switch to a manual EM domain to decrease the size (element count) of the EM task.

Automatic EM Domain: Same size as CFD CD with automatic holes for AC faces

Manual EM Domain: Smaller than CFD CD, the boundaries of the EM Domain body coincide with AC faces
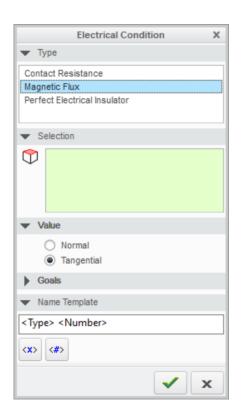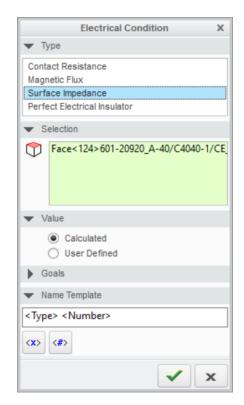
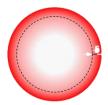SIEMENS

Electrical conditions

You can define the following Electrical and Magnetic Conditions on solid faces:

Electrical Contact Resistance (available before), Tangential Flux (default), Normal Flux, Perfect Insulator, Surface Impedance (Time-Harmonic only).
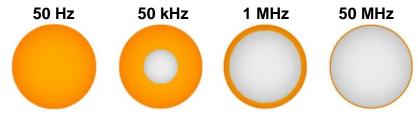

Surface Impedance can be used on conductors to reduce problem size for skin effect simulation (the interior of any conductor using surface impedance is not meshed).

In case the skin effect is significant the current flows in the very thin layer so the surface approximation can be applied (for Time-H,) reducing the size of the model by excluding the internal volume.
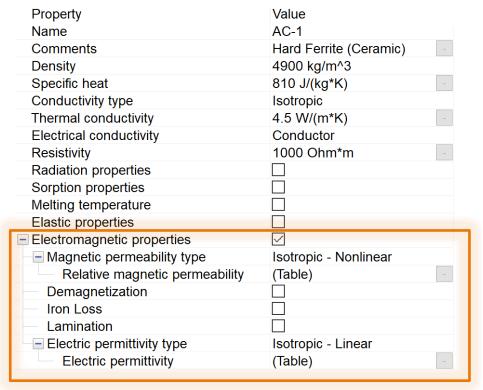
50 Hz   50 kHz   1 MHz   50 MHz

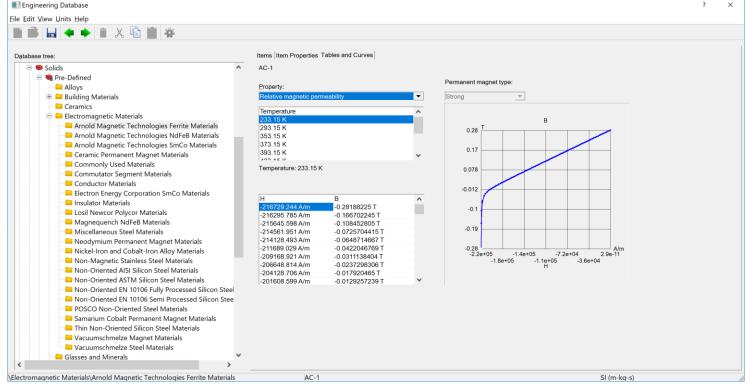Skin depth for conductor with diameter of 0.51 mm

SIEMENS

Extended material library

Material properties are extended with electromagnetics properties. More than 300 materials commonly used in electromagnetic analyses are added in groups.
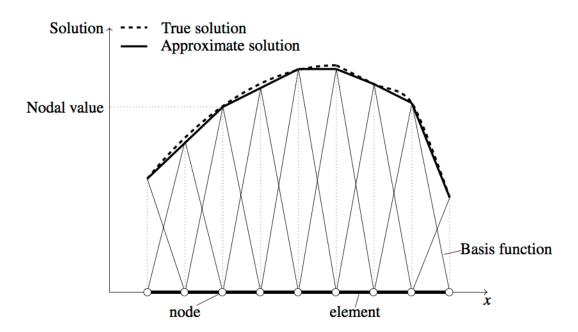
SIEMENS

Meshing

The entire domain under investigation is approximated as an assembly of discrete elements, so-called finite elements, interconnected at points common to two or more elements, so-called nodes. The approximate solution to the problem within the element is obtained as a linear combination of nodal values of the variables and the basis functions for the element.

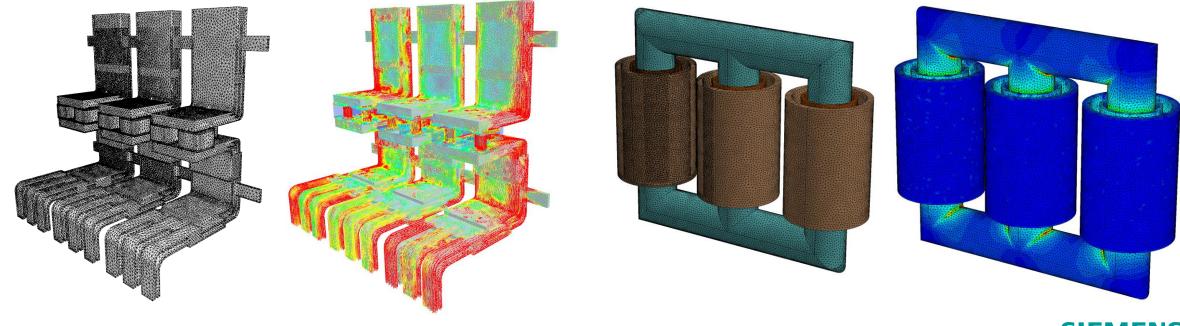

https://www.iue.tuwien.ac.at/phd/rovitto/node63.html

Schematic view of the linear discretization of the domain in elements and nodes. The true solution is represented as a continuous function (dotted line) and the approximate solution is described as a piecewise (1st order) polynomial (solid line).

Simcenter FLOEFD EM solution

Simcenter FLOEFD employs the finite-element method to solve the equation for the magnetic potential. With this method, the region of the problem is divided into a mesh of triangular elements, and the potential in each element is approximated by a simple function of the x and y coordinates. The simplest function is a linear variation with position; this gives first-order elements, where the potential inside a triangular element is obtained from the potentials at the three vertices or nodes. High-order elements use high-order polynomials and additional nodes to represent the potential.
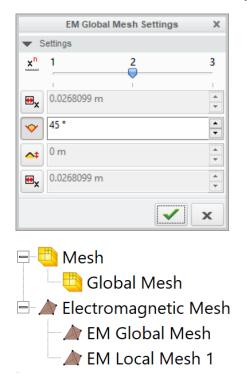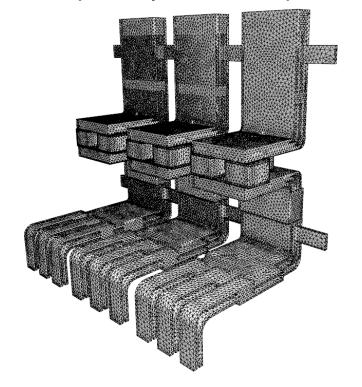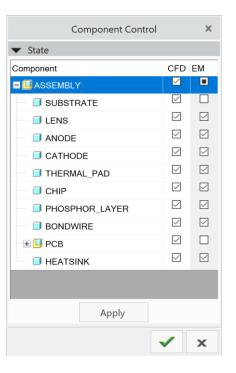
SIEMENS

Meshing

An electromagnetics task utilizes Simcenter MAGNET's Finite Element Mesher. A Simcenter FLOEFD project now has two meshes: CFD and EM. Like the CFD mesh, you can set EM Global (applied to everything) and EM Local Mesh Settings.

You can exclude components from EM or CFD tasks separately in the Component Control.
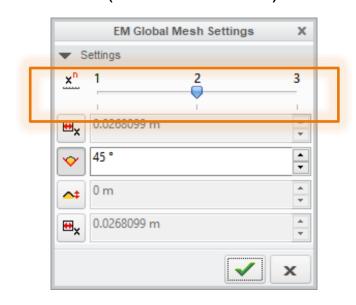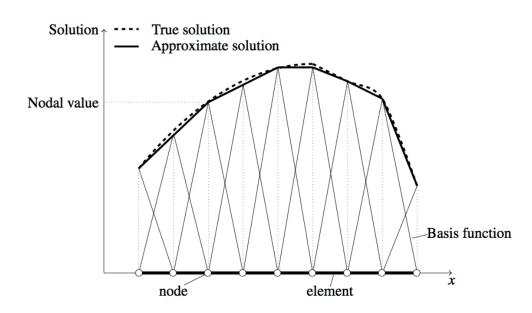
SIEMENS

Polynomial order

The magnetic potential in each element of the mesh is modeled as a polynomial in the spatial coordinates (x, y, and z). In general, higher orders give greater accuracy, but involve longer solution times. Order 1 gives a fast solution of low accuracy, and is useful for initial tests on a complex model. In most cases, however, good results will be obtained by setting the polynomial order to 2. In special cases, a high order must be used. Where solution time is an issue, the polynomial order should be increased only in the areas of interest by using the EM Local Mesh. Strictly speaking, polynomial order is not a mesh option as it does not affect the mesh but the solution (basic functions).
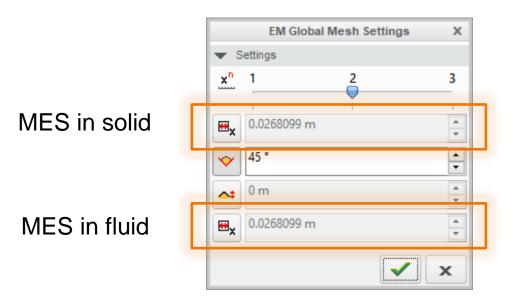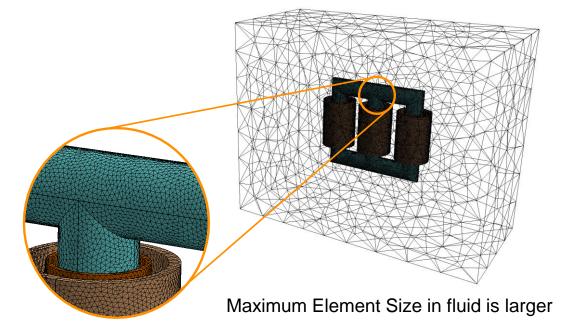
Maximum element size

Maximum element size is the maximum element edge length. This can be increased to start with a coarse mesh, or reduced to give a fine mesh. To decrease the mesh count often the Maximum Element size in the fluid (surrounding air) can be set to be greater than in the solid.

Although maximum element size parameters for solid and fluid are optional it is recommended to define it for solid bodies. For fluid it can be kept undefined – the mesher will find the optimal value which usually works well.

MES in solid

MES in fluid

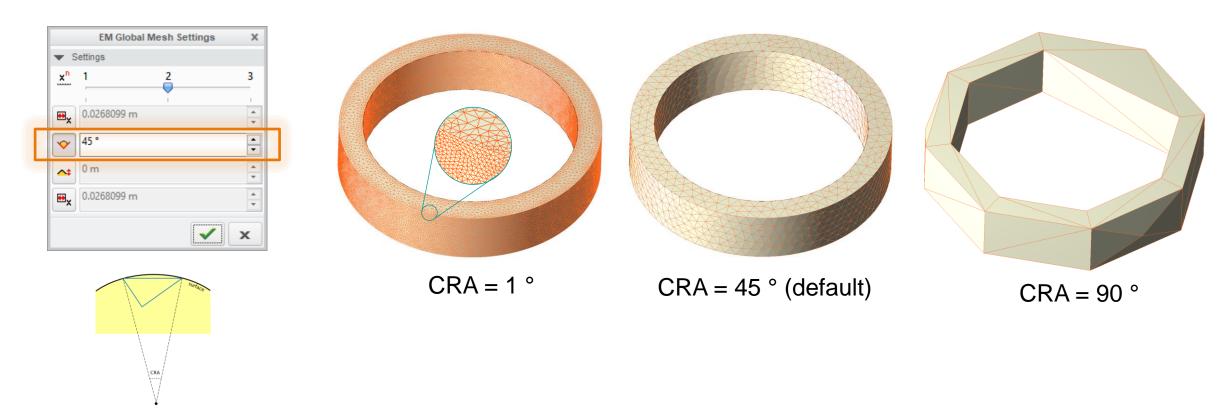Maximum Element Size in fluid is larger

Curvature Refinement Angle in Solid

The Curvature Refinement Angle (CRA) specified for a curved component face or edge corresponds to the maximum angle, at the local center of curvature, spanned by an element side in contact with the face or edge. CRA defines the size of an element at the solid/fluid interface of a curved surface, edge.

CRA = 1 °

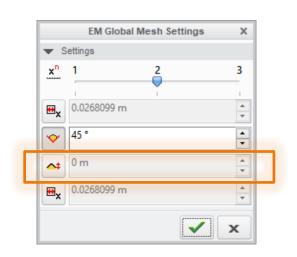CRA = 45 ° (default)

CRA = 90 °

SIEMENS

Curvature Minimum Element Size in Solid

The Curvature Minimum Element Size In Solid (CMES) specifies the minimum element length on curved component entities.

A model entity with very high curvature may cause too much local refinement. The Curvature Minimum Element Size In Solid sets a lower bound threshold to avoid over-discretization of a curved model entity, by preventing the default or specified curvature refinement from effectively decreasing the Maximum Element Size below the Curvature Minimum Element Size on curved component entities.

CRA = 1 °
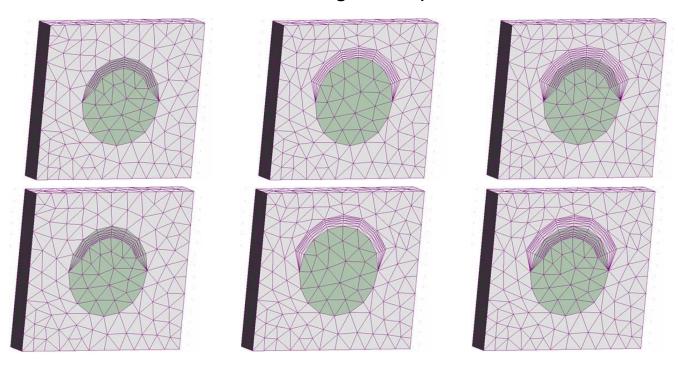CMES = 0 (default)

CRA = 1 °
CMES = 3 mm

Additional Local Mesh Settings: Mesh Layers

The Mesh layers feature allows you to create highly anisotropic elements near model surfaces and edges. This is especially useful for modeling skin depth or thin air gap regions where multiple layers of long, thin elements will suffice instead of an over-discretized mesh with traditional, good aspect ratio elements.

Uniform (top) and Logarithmic (bottom) mesh layers radiated
inward, outward and on both sides from the selected top semi-surface of the cylinder.
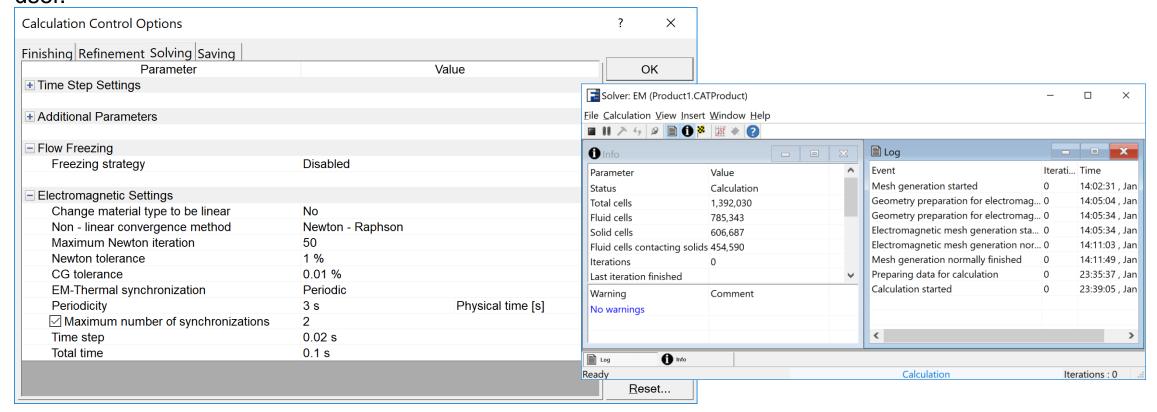
SIEMENS

Solution

SIEMENS

Solver: CFD-EM Co-simulation

Solver is Simcenter MAGNET's EM solver which starts automatically. It takes the temperature from Simcenter FLOEFD and sends the Ohmic losses back to Simcenter FLOEFD. The exchange frequency is defined by the user.
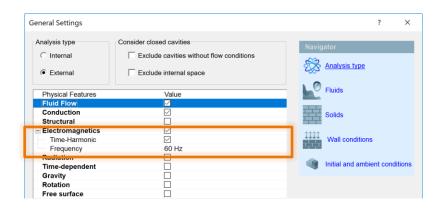
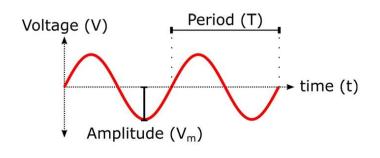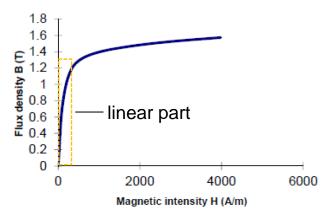SIEMENS

Solver: CFD-EM Co-simulation

Time-Harmonic

If all AC sources are true (no damping, delay) sinusoidal with the same frequency and you have linear material properties, then sinusoidally-varying sources will give rise to sinusoidally-varying fields and such analyses can be calculated with the time-harmonic solver so a time-consuming true transient analysis is not need. The sources and fields are represented by complex phasors.

Theoretically, a time-harmonic analysis is only possible when all the materials in the problem are linear. However, Simcenter FLOEFD time-harmonic 3D solvers are actually quasi non-linear solvers, taking into account the approximate material non-linearities by trying to find the operation point on a non-linear B-H curve, using its first few data points.
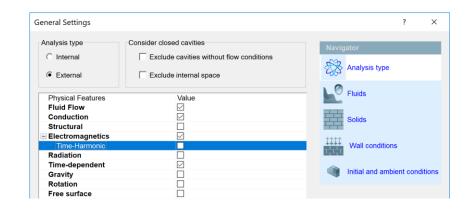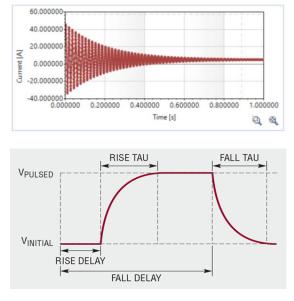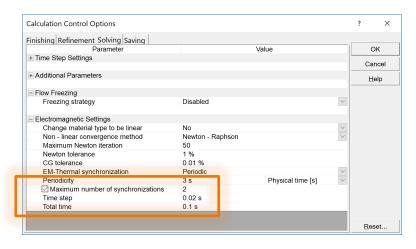
Solution Type: Time-Harmonic and Transient

Transient

If AC sources have non-sinusoidal waveforms or have different frequency or material properties that cannot be linearized you need to enable a transient electromagnetic simulation.

Transient EM solver is enabled when you choose Time-dependent analysis in General Settings and disable Time-Harmonic. In Calculation Control Options you define time step and total time for a transient EM analysis.
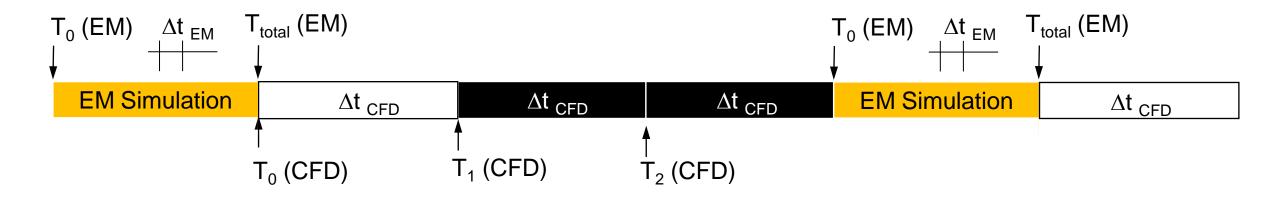
SIEMENS

It is assumed that an EM transient is much more faster than a CFD transient, so for each synchronization CFD time step (white period below), the whole transient EM simulation is calculated. The electromagnetic losses are averaged over the EM total transient time and the magnetic field is saved for the last EM time moment.

T_0 (EM) Δt_{EM} T_{total} (EM) T_0 (EM) Δt_{EM} T_{total} (EM)

| EM Simulation | Δt_{CFD} | Δt_{CFD} | Δt_{CFD} | EM Simulation | Δt_{CFD} |

T_0 (CFD) T_1 (CFD) T_2 (CFD)

In assumption that EM characteristic time is much smaller, a transient CFD task can synchronize with Time-Harmonic EM task (if appropriate for EM task).
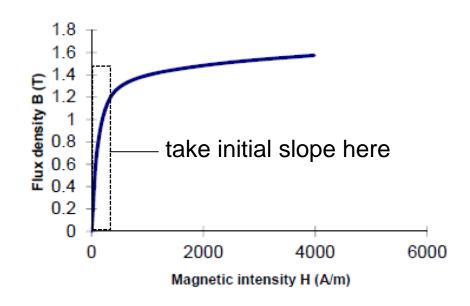
SIEMENS

Solver Options

Change material type to be linear:

- Yes. The non-linear materials are approximated as linear by the solver using the initial slope of the material's B-H curve. Linear solutions are useful for solution estimates as they require less solving time than non-linear solutions (no Newton iterations are performed). To obtain a faster solution, you can choose the linear material type even if the model contains non-linear materials. However, the solution does not take saturation into account.

- No (default). The non-linear materials are not approximated as linear by the solver. Models containing non-linear materials require that a number of Newton iterations be performed to converge to a solution.

- Yes, for Time-Harmonic only. This is the same option as Yes if the Time-Harmonic option is selected. Otherwise it equals No.

take initial slope here

Solver Options

Non-linear convergence method.

- Newton-Raphson. This method is used for non-linear solution types. Each step of the method solves a set of linear equations by the Conjugate Gradient (CG) method.

- Successive Substitution. If a non-linear problem has difficulty converging with the Newton-Raphson method, the Successive Substitution method can be used instead. The Successive Substitution method is slower than the Newton-Raphson method, but is generally more certain to reach convergence.

Maximum Newton iteration sets a limit for the number of iterations of a non-linear solution. The default is 50 iterations. The solver will abort if a solution does not converge within the maximum number of Newton steps.

Newton tolerance sets the maximum percentage of allowable change in the field from one Newton step to the next. The iterative solution process ends when the tolerance is met. The default of 1% is generally acceptable. If necessary, the tolerance can be reduced to obtain a more accurate solution, but this will increase solving time. The Newton tolerance only applies to non-linear problems.
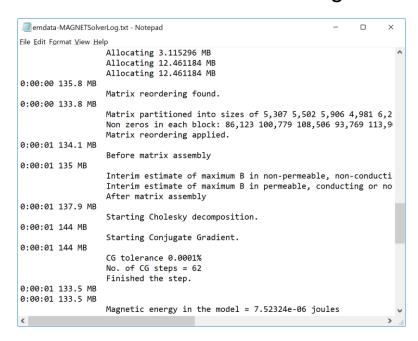
SIEMENS

CG tolerance sets the maximum percentage of allowable change from one Conjugate Gradient (CG) step to the next. The iterative solution process ends when the tolerance is met. The default of 0.01% is generally acceptable. If necessary, the tolerance can be reduced to obtain a more accurate solution, but this will increase solving time.
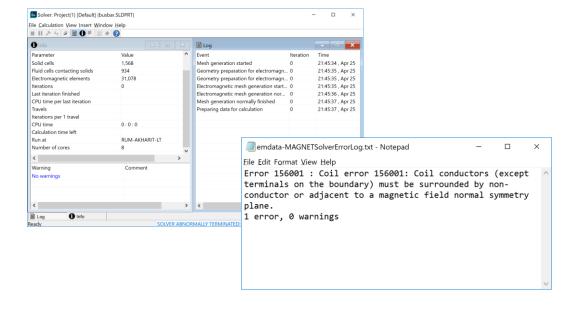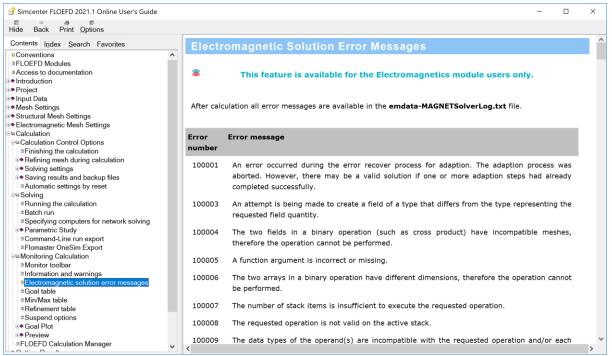Convergence can be seen in the emdata-MAGNETSolverLog.txt.

SIEMENS

EM Solver Log and error Log

In case of Solver error you can see the reason in the solver log: emdata-MAGNETSolverErrorLog.txt. The description of errors is given in the Help.

SIEMENS
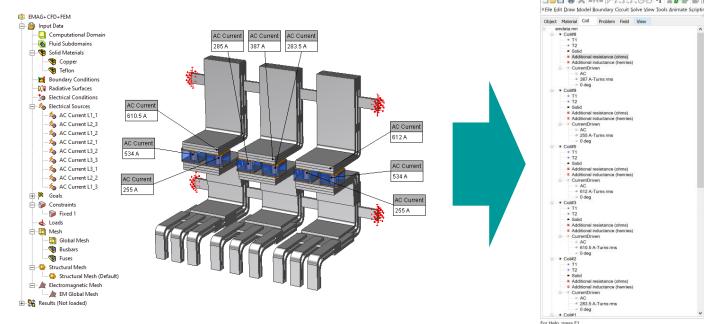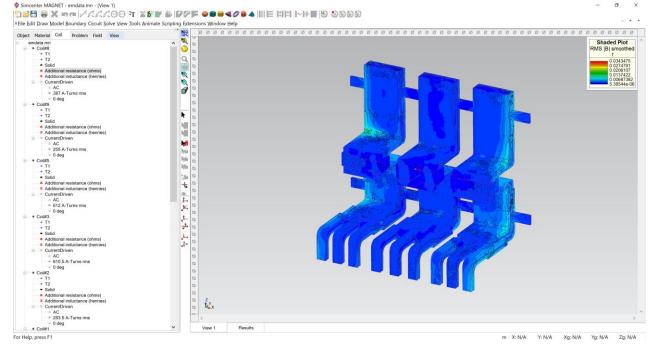
Simcenter FLOEFD Export to Simcenter MAGNET

Simcenter FLOEFD solver generates a Simcenter MAGNET (emdata.mn) file that can be opened in Simcenter MAGNET (from version 2021.1) for a deeper analysis. This emdata.mn file (saved in the project folder) contains all the settings, mesh and results but not the initial geometry. An EM expert can then inspect the EM solution with additional capabilities of Simcenter MAGNET and perform an advanced analysis.

Results Processing

You can visualize EM parameters: Electric Current Density - J, Magnetic Flux Density – B, Ohmic, Iron and total Electromagnetic losses. EM mesh and Magnetic Flux Density and Electric Current Density Vectors.
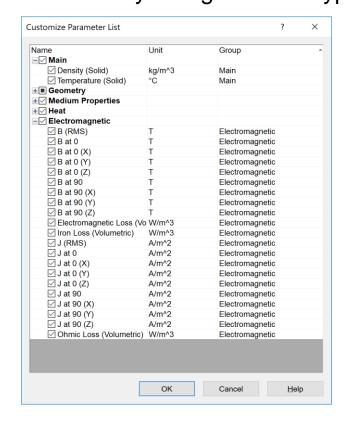

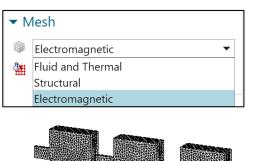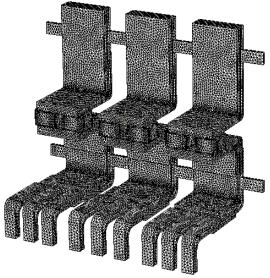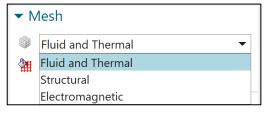**J**   **Ohmic Loss**   **Temperature**
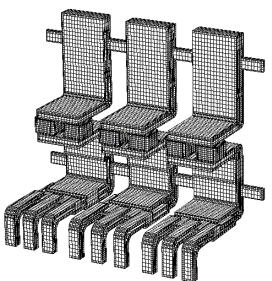
SIEMENS
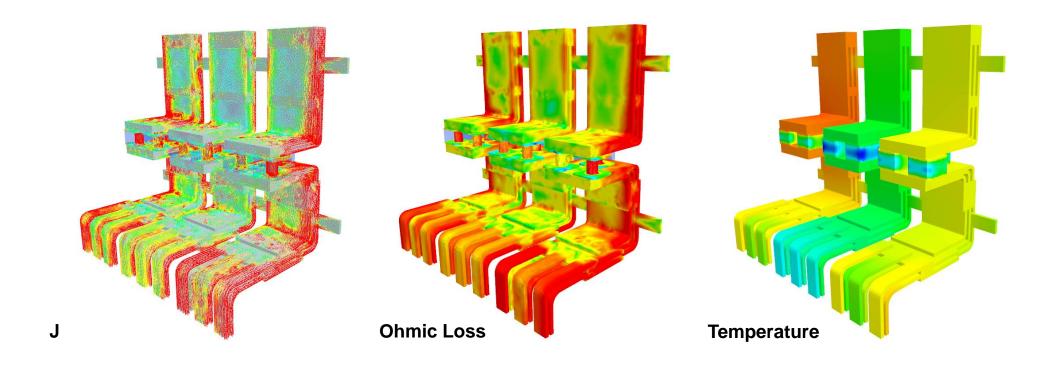
Results Processing Inside CAD

When displaying mesh-only in Cut plots and Surface plots, you need to choose between Electromagnetic, CFD (Fluid and Thermal) and Structural meshes. In case a contour parameter is added Simcenter FLOEFD automatically recognizes the type of mesh to display.
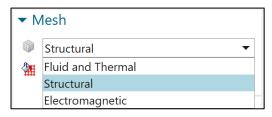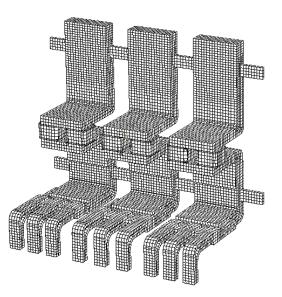
Capabilities and Limitations

Supported:

- AC
- Time-Harmonic solver
- Transient solver (EM simulation time is considered to be much smaller than CFD characteristic time)
- Surface Impedance (for effective Skin effect analysis)
- Permanent Magnets
- Iron Loss
- Demagnetization
- Linear and non-linear electromagnetic material properties
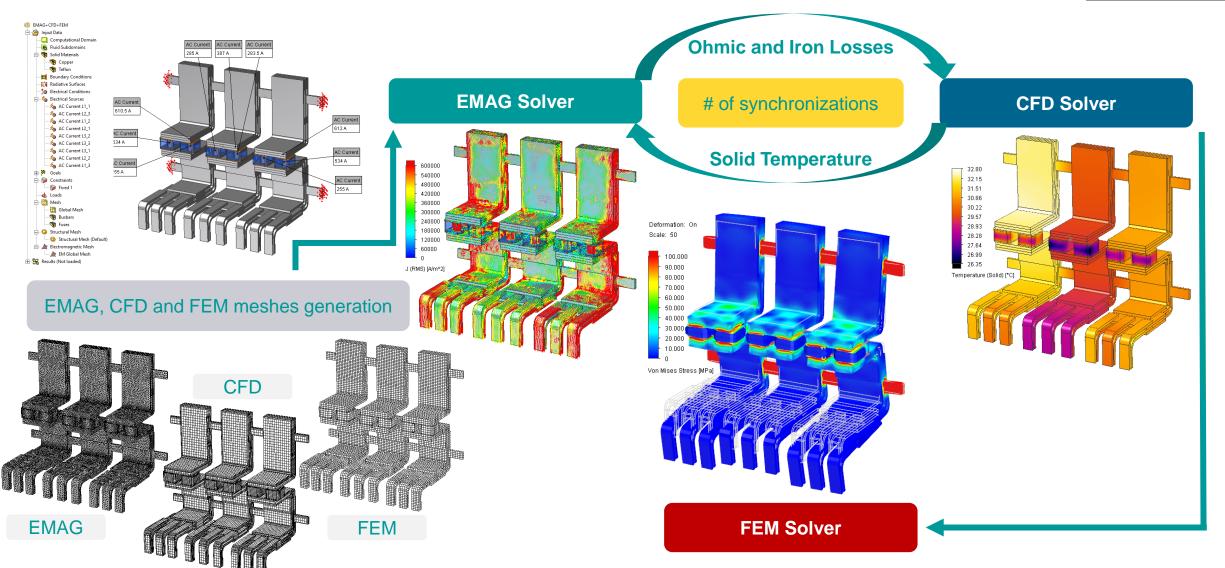- Visualization of J, B, Ohmic and Iron Losses

Not Supported:

- 2D
- Motion
- 1D Electrical Circuit
- Symmetry/Periodic conditions
- Materials:
  - Hysteresis
  - Anisotropic Magnetic Permeability
  - Complex Material Properties
  - Magnetizing ratio
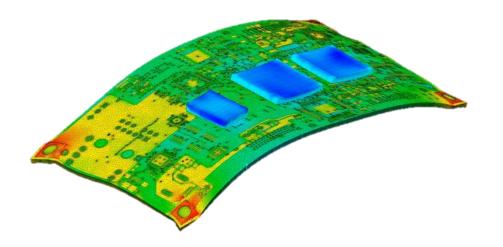- Visualization of H, Demagnetization
- Adaption

SIEMENS

Electromagnetics + Fluid Dynamics with Conjugate Heat Transfer + Linear Structural

Ohmic and Iron Losses

EMAG Solver

# of synchronizations

CFD Solver

Solid Temperature

EMAG, CFD and FEM meshes generation
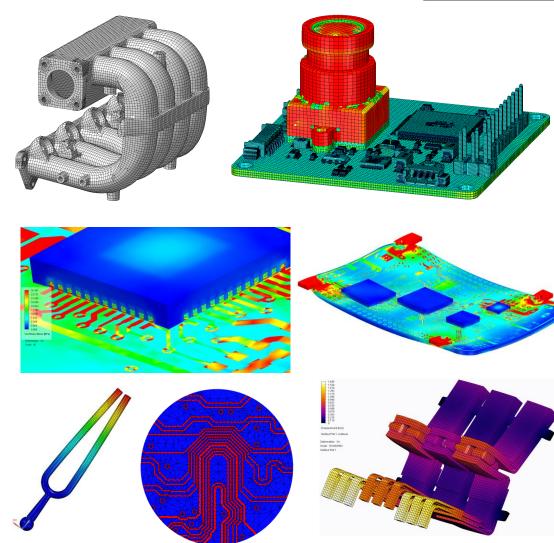
CFD

EMAG

FEM

FEM Solver

SIEMENS

Structural

Structural analysis capabilities

- Fully automated hexahedral dominant mesh generator
- Fully automated Smart PCB mesh generator
- Linear solver
- Modal frequency analysis
- Isotropic and orthotropic elastic material properties
- Transferred pressure and temperature fields from fluid dynamic analysis with conjugate heat transfer as pressure and temperature loads
- Multiphysics task
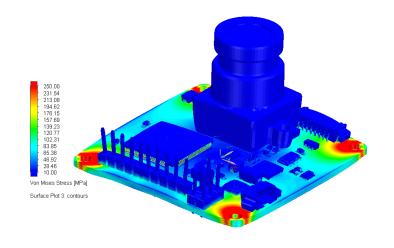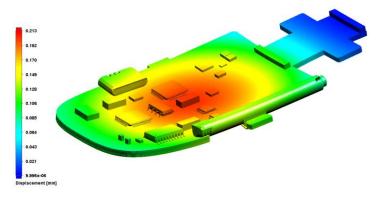- Export project to SC3D (FEM geometry and conditions)

SIEMENS

Structural finite element linear solver capabilities

Simcenter FLOEFD uses a finite element (FE) non-iterative solver for structural analysis. The range of analysis types in the FE solver includes: static response to loads, to thermal expansion and to enforced deformation and also determination of eigenvalues. Since the relationship between the load applied to an object and the response of the object is assumed linear, the set of linear equations is considered. A linear approach can be used when these assumptions are made:

• Deformations are small and do not impact the calculation result; set of linear equations are based on non-deformed geometry
• Temperature variation within the model is not considerable or material properties temperature dependency is not strong.

Von Mises Stress [MPa]

Surface Plot 3: contours
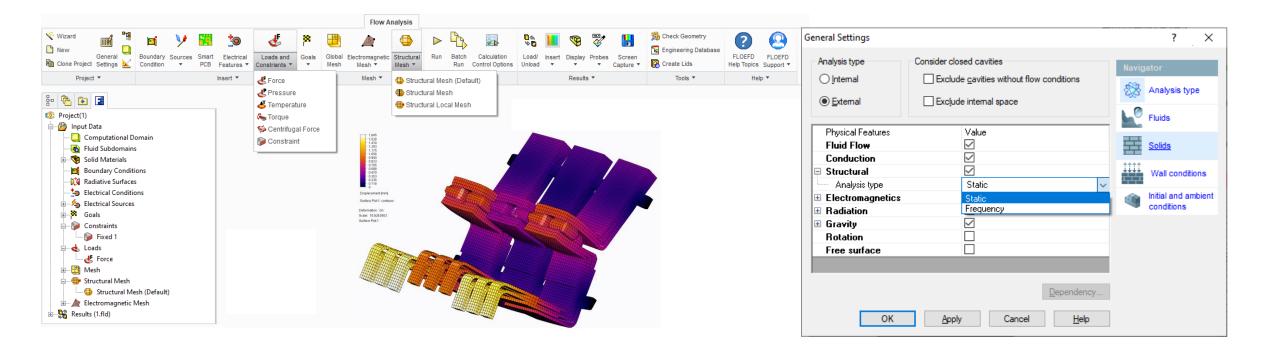

Displacement [mm]

SIEMENS

Structural: GUI summary

The Simcenter FLOEFD toolbar now has new groups of buttons to create loads, constraints and to set up a structural mesh. Options in the **General Settings** dialog are redesigned to simplify working with multiphysics projects. Structural options with Static/Frequency subtypes are added.
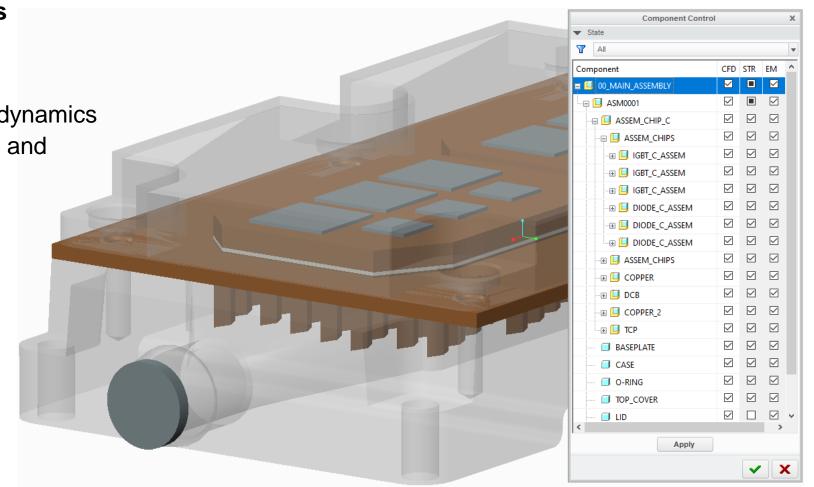
SIEMENS

Structural: component control

Component control for Multiphysics

Component status may be managed
independently for each analysis (fluid dynamics
with conjugate heat transfer, structural and
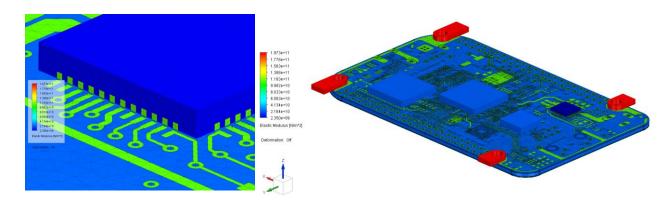electromagnetic)

SIEMENS

Structural: solid material properties

Solid material properties

Solid material properties are enriched with elastic options in the **Engineering Data Base**. Elastic properties can be isotropic or orthotropic, each value may be set as a constant or tabular dependency on temperature.

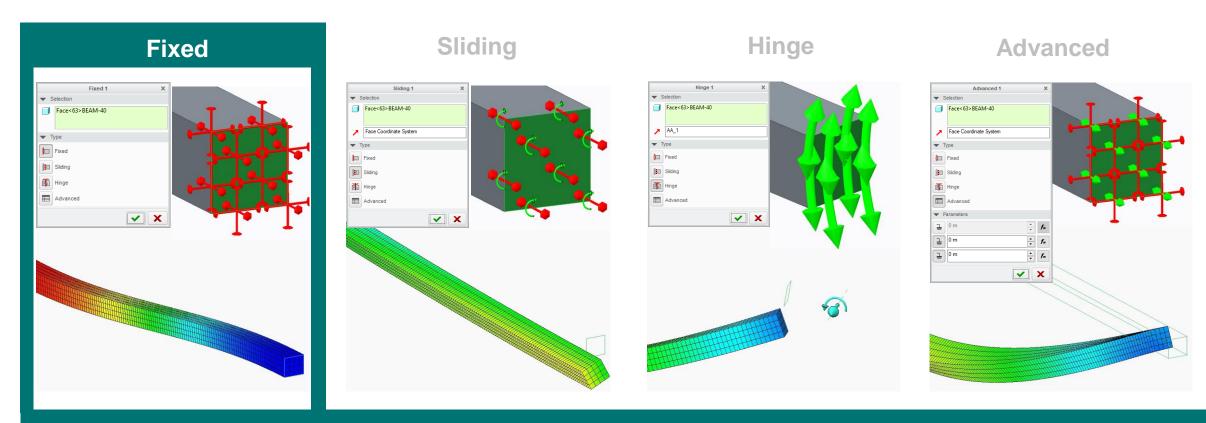Solid materials can be imported from CAD (NX, SE, SW)

Items	Item Properties	Tables and Curves	
Property		Value	
Name		Alloy	
Comments			_
Density		8100 kg/m^3	
Specific heat		1000 J/(kg*K)	_
Conductivity type		Orthotropic	∨
Thermal conductivity in X		15 W/(m*K)	_
Thermal conductivity in Y		15 W/(m*K)	_
Thermal conductivity in Z		15 W/(m*K)	_
Electrical conductivity in X		Dielectric	∨
Electrical conductivity in Y		Dielectric	∨
Electrical conductivity in Z		Dielectric	∨
Radiation properties		☐	
Sorption properties		☐	
Melting temperature		☑	
Temperature		1683.15 K	
Elastic properties		☑	
Elastic modulus in XX		1e+11 N/m^2	_
Elastic modulus in YY		1e+11 N/m^2	_
Elastic modulus in ZZ		1e+11 N/m^2	_
Poisson's ratio in XY		0.3	_
Poisson's ratio in XZ		0.3	_
Poisson's ratio in YZ		0.3	_
Thermal Expansion Coefficient in XX		2e-05 1/K	_
Thermal Expansion Coefficient in YY		2e-05 1/K	_
Thermal Expansion Coefficient in ZZ		2e-05 1/K	_
Shear Module in XY		5e+10 N/m^2	_
Shear Module in XZ		5e+10 N/m^2	_
Shear Module in YZ		5e+10 N/m^2	_
Electromagnetic properties		☐	

SIEMENS

Structural: fixed constraints

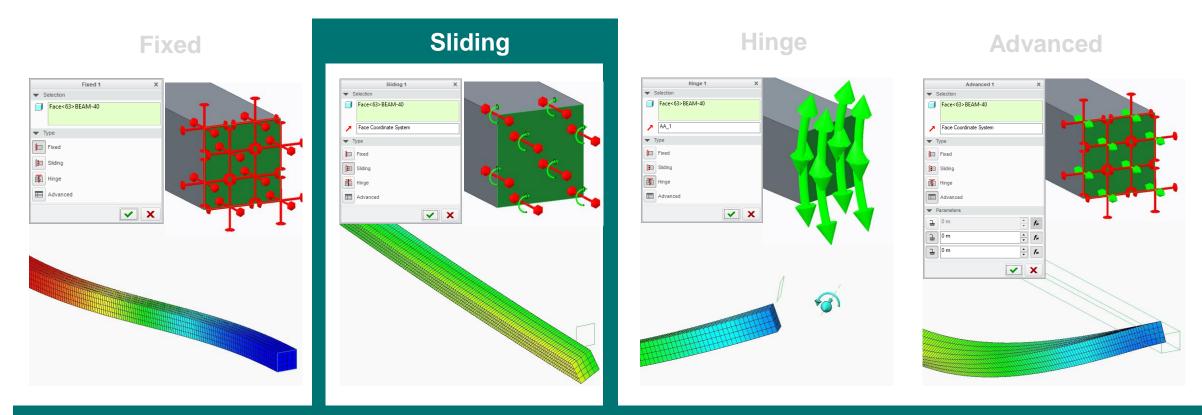**Fixed**

Sliding

Hinge

Advanced

<u>Fixed</u> constraint fixes all degrees of freedom of the selected geometry (points, vertices, edges, faces, bodies or components), does not require additional geometry or settings.
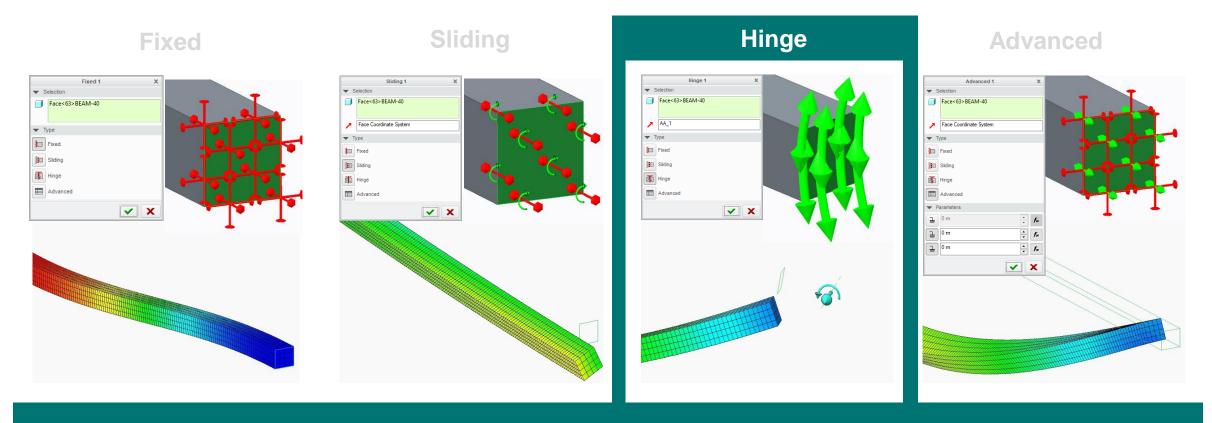
SIEMENS

Structural: sliding constraints

Fixed

Sliding

Hinge

Advanced

Sliding constraint allows any in-plane movements of the selected geometry (points, vertices, edges, faces, bodies and components). Requires user to specify axis, and selected geometry may rotate around the axis and slide in-plane normal to the axis.

SIEMENS

Structural: hinge constraints

Fixed

Sliding

Hinge

Advanced
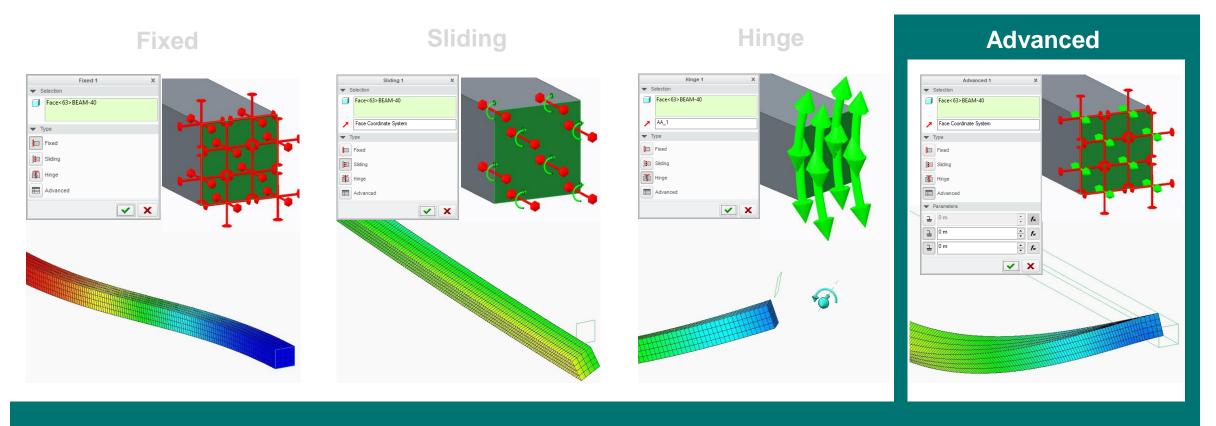
<u>Hinge</u> constraint allows rotating of the selected geometry (points, vertices, edges, faces, bodies and components) around the axis. Requires user to specify axis, and movement along the axis is prohibited.

SIEMENS

Structural: advanced constraints

Fixed

Sliding

Hinge

Advanced
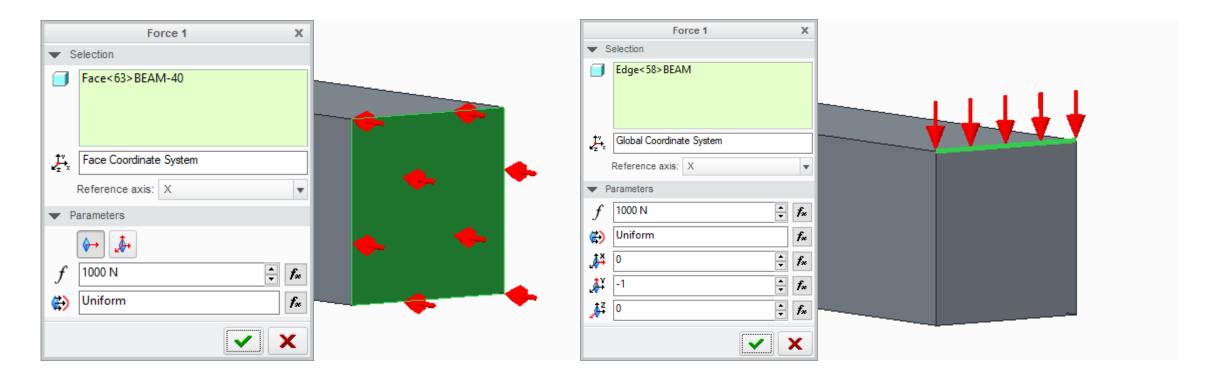
Advanced constraint allows management of movements in all directions independently. Requires user to fix or to set free the switches for direction X, Y and Z. Selected geometry may be pre-displaced in the directions which are fixed. Constraint visualization shows state of each direction with green and red colors.

SIEMENS

Structural: force load
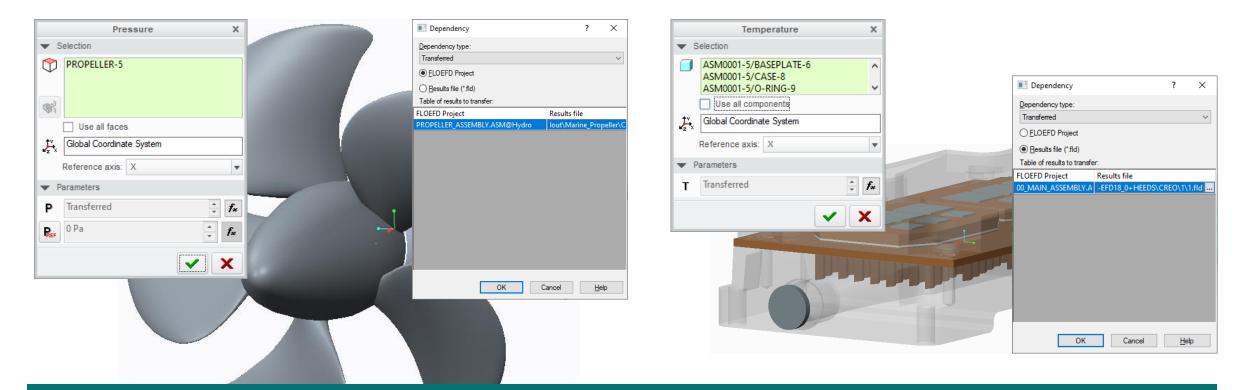
Force load can be applied to selected geometry (points, edges, faces, bodies or components), force direction can be set as normal to face for surfaces or through vector components for any geometry. Magnitude is constant or a dimensionless profile. Simcenter FLOEFD scales profile to keep the specified integral value (force module is to be integrated).

SIEMENS

Structural: pressure and temperature loads
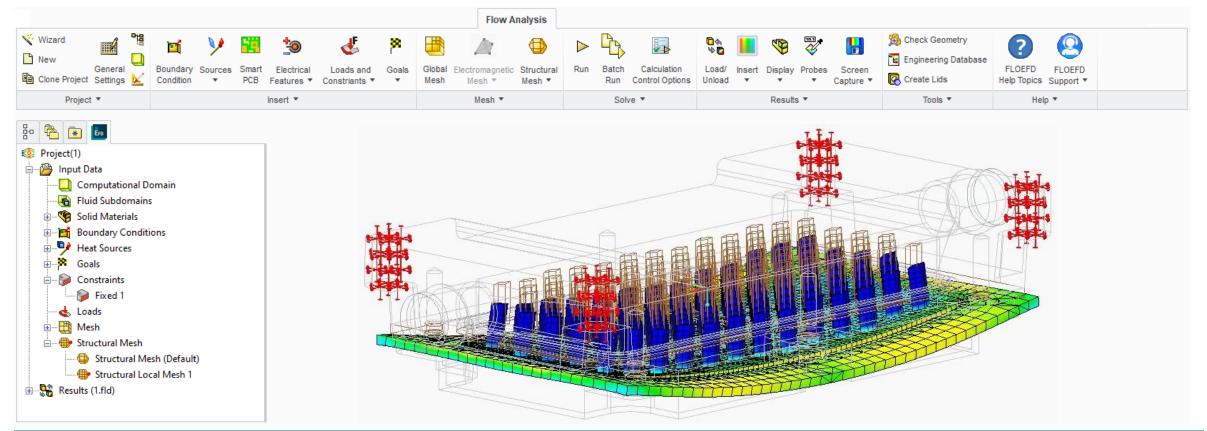
Pressure loads may be applied to a particular surface or component or applied to all surfaces in the project, Temperature loads may be applied to components. Values for Pressure and Temperature load can be specified directly or transferred from results of another Simcenter FLOEFD project through Dependency dialog.

SIEMENS

Structural: pressure and temperature loads automatic transfer for coupled projects
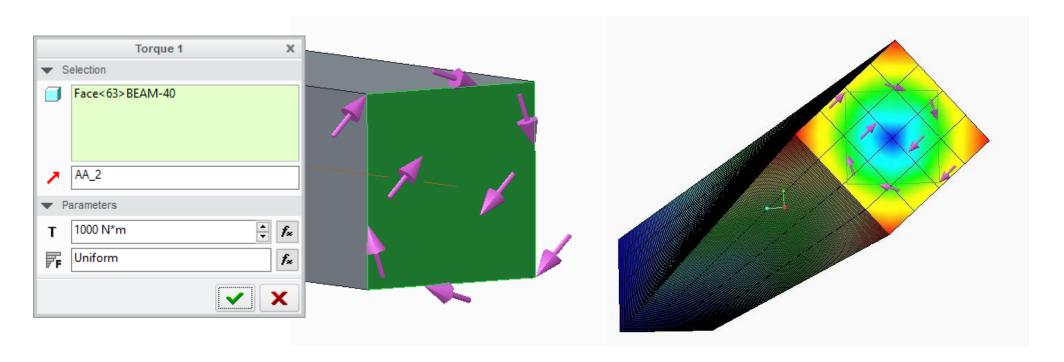
Several options (fluid flow with conjugate heat transfer and structural analysis) can be considered in one project. Pressure and Temperature loads are transferred automatically (no Load feature is necessary) from the fluid flow and heat transfer results in this case, and only constraints and structural mesh settings are required to complete the project definition.

SIEMENS

Structural: torque load
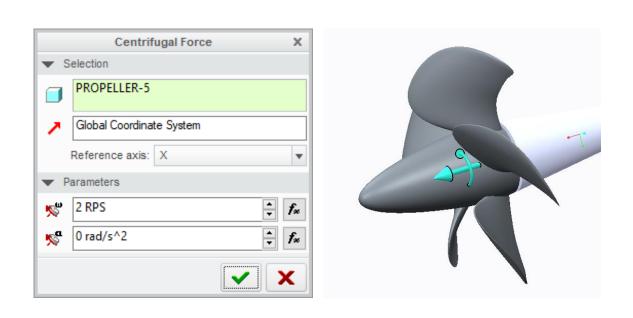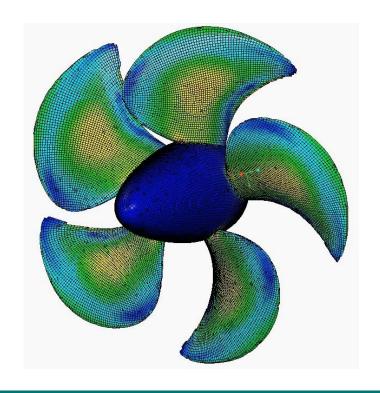
Torque load around an axis can be set on the selected geometry (points, edges, faces, bodies or components), then Simcenter FLOEFD applies the specific force to each element, force profile is linearly dependent on shoulder by default and might be changed manually. Simcenter FLOEFD scales force profile to keep the specified integral value.

SIEMENS

Structural: centrifugal force load

If <u>Centrifugal Force</u> should be applied to a rotating component, then the axis of rotation should be selected. Angular velocity or angular acceleration can be specified. If both options (fluid flow analysis with rotation and structural analysis) are considered in one project, <u>Rotation Region</u> will NOT be applied automatically to rotating component for structural analysis, so <u>Centrifugal Force</u> has to be set manually.

Structural: hexahedral mesh
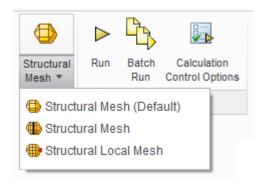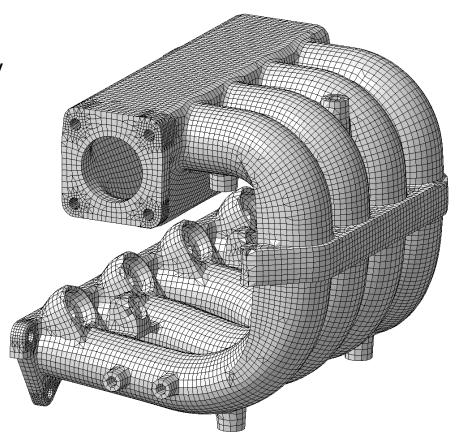
Hexahedral dominant fully automated mesh generator

Simcenter FLOEFD generates hex-dominant mesh automatically, only element size is required. The glue contacts between different mesh regions are created automatically.

Mesh settings can be set with three dialogs:
- **Structural Mesh (Default)**
- **Structural Mesh**
- **Structural Local mesh**

Structural: structural mesh (default)
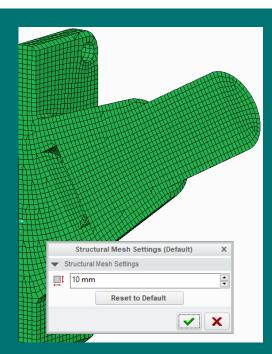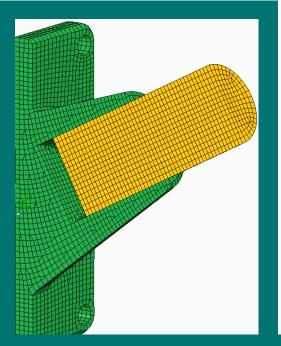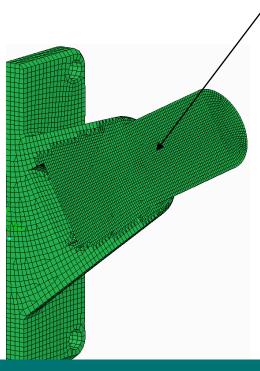
1. Default settings
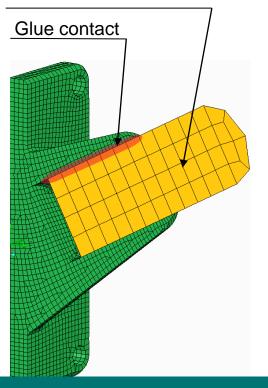
2. Different materials with default settings

3. Structural mesh

Glue contact

4. Structural Local mesh

Structural Mesh Settings (Default)

Structural Mesh Settings

10 mm

Reset to Default

Structural Mesh (Default) applies the Element size to all components. If components are made of the same material they are united by Boolean operations and mesh does not "feel" the boundary (picture 1). If materials are different, the meshes are automatically created "node to node" (picture 2).

SIEMENS
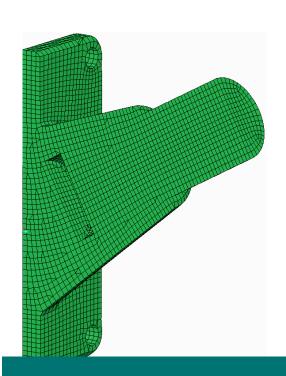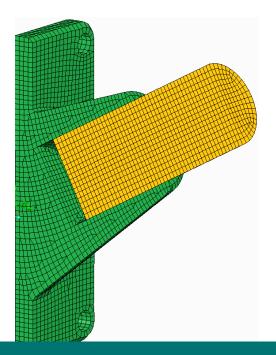
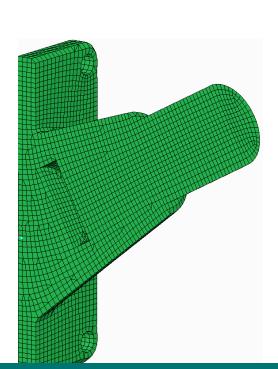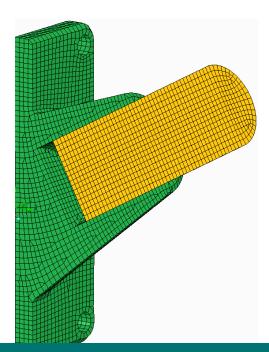# Structural: structural mesh
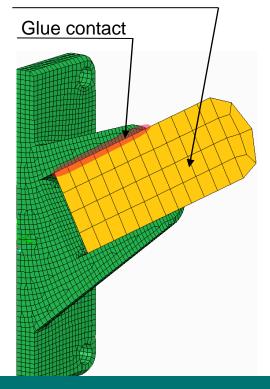
1. Default settings
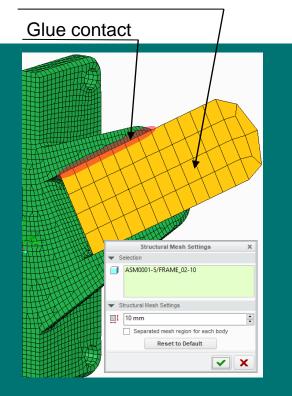
2. Different materials with default settings

3. Structural mesh

Glue contact

4. Structural Local mesh

Structural Mesh Settings

Selection

ASM0001-5/FRAME_02-10

Structural Mesh Settings

10 mm

Separated mesh region for each body

Reset to Default
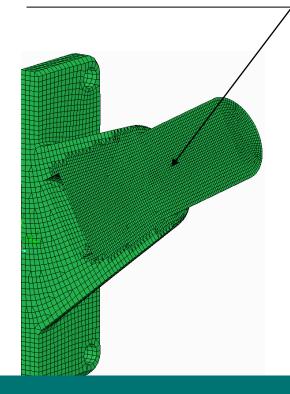
<u>Structural Mesh</u> applies the <u>Element size</u> to particular components and creates different mesh regions (picture 3). The rest of the components use default settings. Glue contacts between different regions are created automatically, and transition between elements of different sizes is not required.

SIEMENS

Structural: structural local mesh

1. Default settings
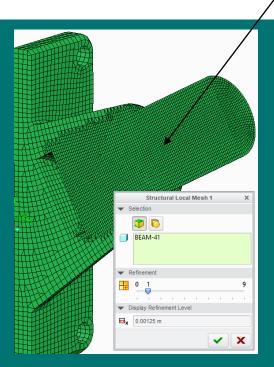
2. Different materials with default settings

3. Structural mesh

Glue contact

4. Structural Local mesh

Structural Local Mesh 1
- Selection
  - BEAM-41
- Refinement
  - 0 1   9
- Display Refinement Level
  - 0.00125 m

Structural Local Mesh applies the Element size to any type of geometry, and the mesh generator provides smooth transition between elements of different size. Mesh parameter is Refinement level, resulting element size divides the original size by 2, 4, 8 and so on in accordance with refinement level (picture 4).

SIEMENS
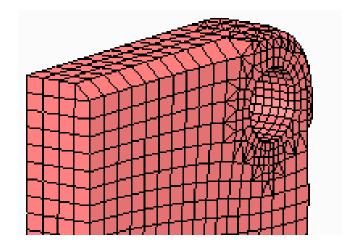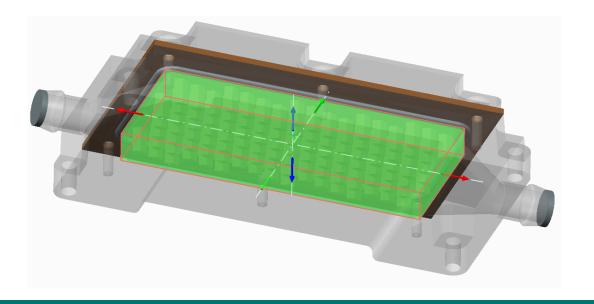
Structural: structural local mesh
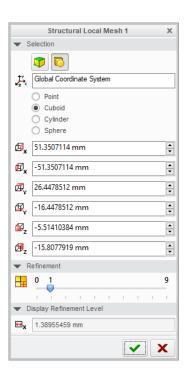
Local mesh on cylindrical surfaces within holes
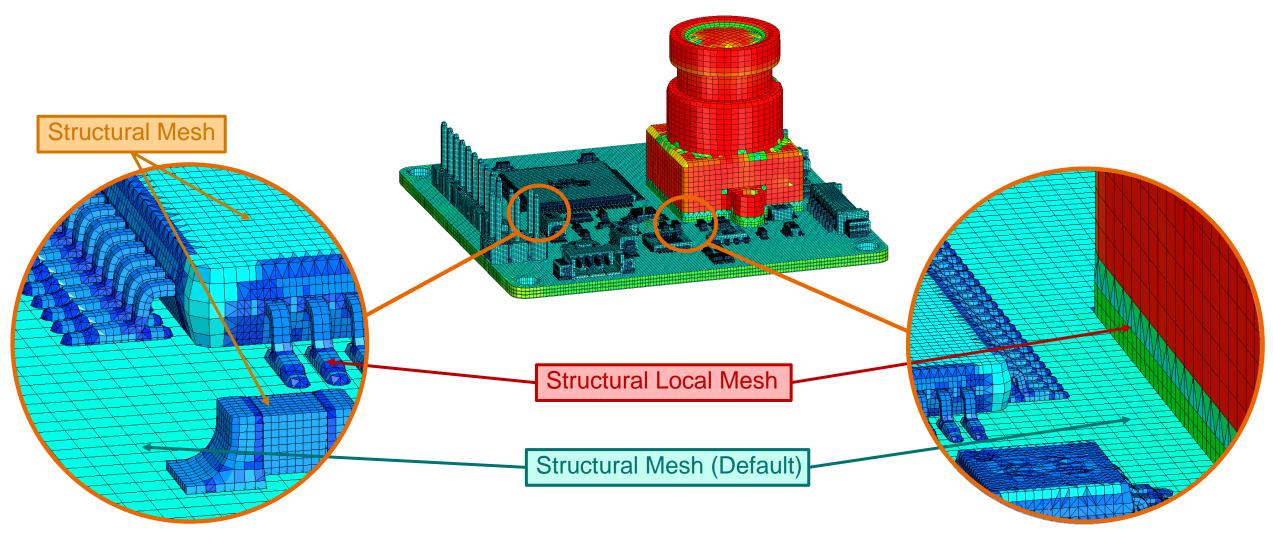
Local mesh in cuboid to refine all pins

Structural Local Mesh Panel:

- Structural Local Mesh 1
- Selection
  - Global Coordinate System
  - ○ Point
  - ● Cuboid
  - ○ Cylinder
  - ○ Sphere
  - 51.3507114 mm
  - -51.3507114 mm
  - 26.4478512 mm
  - -16.4478512 mm
  - -5.51410384 mm
  - -15.8077919 mm
- Refinement
  - 0   1   9
- Display Refinement Level
  - 1.38955459 mm

Structural Local Mesh can be set on any type of geometry existing in the model (vertices, edges, faces or components) or can be set on geometry primitives (point, cuboid, cylinder or sphere).

SIEMENS
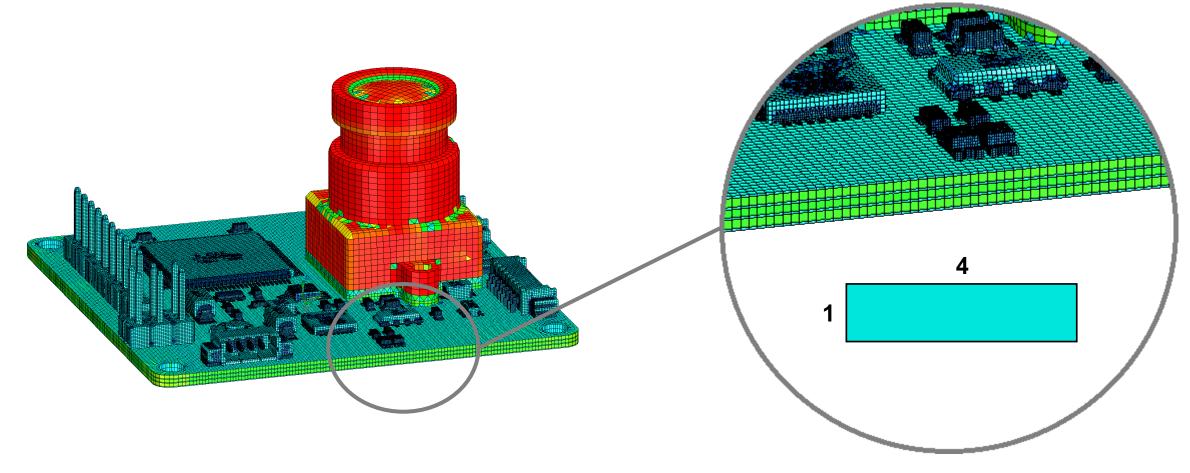
Structural: mesh settings example

Structural Mesh

Structural Local Mesh

Structural Mesh (Default)

SIEMENS

Structural: mesh settings example

Size ratio of the elements of hex-dominant mesh is limited to maximum 1:4.
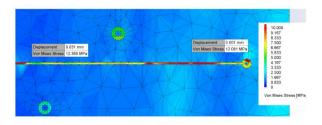(for triangular prismatic mesh in SmartPCB the size ratio is much bigger).

Structural: Smart PCB
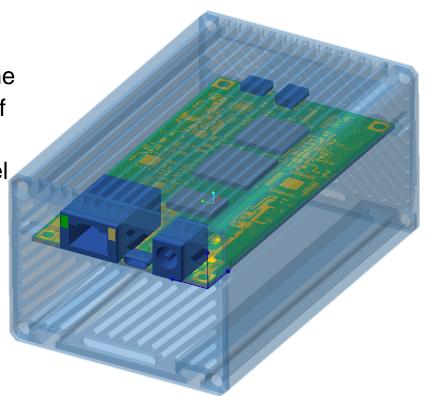
Smart PCB

Smart PCB – unique approach to simulate Multiphysics phenomena on the printed circuit board taking into account all nets with the maximum level of detail is introduced. Simulation models and calculation meshes are generated on EDA data without needing to convert EDA into a CAD model explicitly, so the resulting model is simple but lossless.

Smart PCB can simulate:

• Conduction with free or forced convection, radiation

• Conduction coupled with HyperLynx DC Drop Simulation

• Structural analysis (standalone or coupled)

A special SmartPCB FEM model allows for accurate and time effective stress analysis of a PCB taking all the details of the PCB's internal structure from an original EDA file (traces and vias are resolved without any simplification) into account without creating the explicit geometry in CAD.

SIEMENS
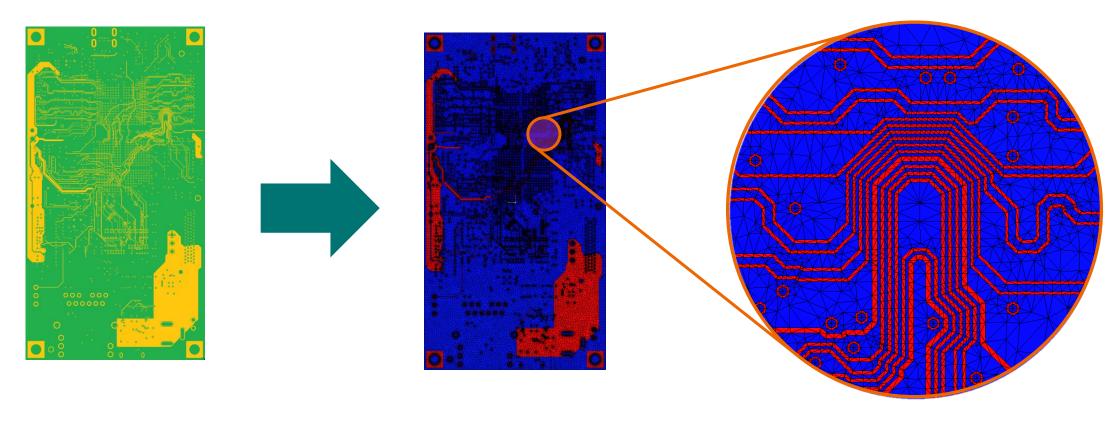
Smart PCB generates a mesh for conduction and structural analysis automatically from EDA data. FEM mesh is constructed from two-dimensional triangles which are extruded to the third dimension. This procedure is performed for each layer independently and resulting layer meshes are in glue contact with each other.
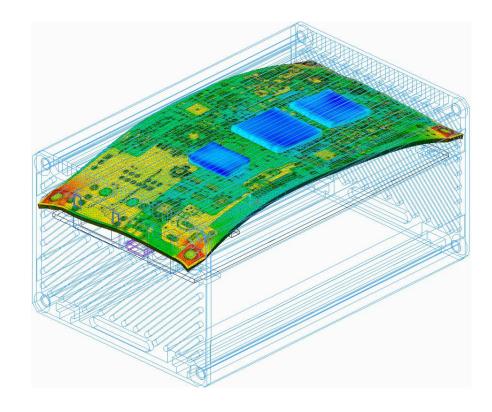
SIEMENS

Simulation times

The simulation model with different phenomena (fluid flow, conjugate heat transfer, radiation, structural analysis) is set as one project, mesh generation and calculations are conducted automatically and take ~ 2h (Intel® Xeon ® W-2235 CPU @ 3.80GHz 6 Cores).

Note: Because the solver is direct (non-iterative), a task with a large number of elements can be memory consuming (and if solver is abnormally terminated for a SmartPCB it may be due to it reaching memory limitations).
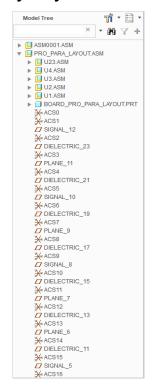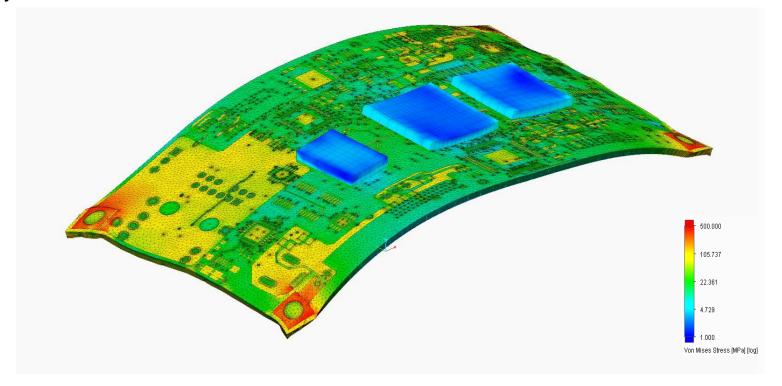
	CFD with thermal Smart PCB (60 200 cells, 5 280 592 nodes)	FEM without Smart PCB (28 700 elements)	FEM Smart PCB only (4 340 000 elements)
Mesher time	5 s	50 s	6 min
Solver time, s	50 min	1.5 min	60 min

SIEMENS

Structural: Smart PCB

Post-processing tools can be applied to the FEM Smart PCB the same way as for simple FEM results, including Surface plots on Smart PCB surfaces and Cut plots on any PCB section. Datum planes through the middle of each layer are created automatically, named as original PCB layers during EDA data importing, so Cut Plots for necessary layers can be easily created.
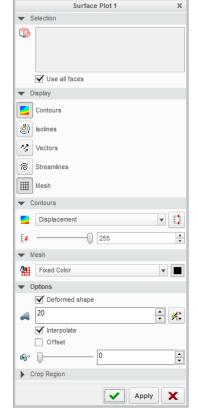
SIEMENS

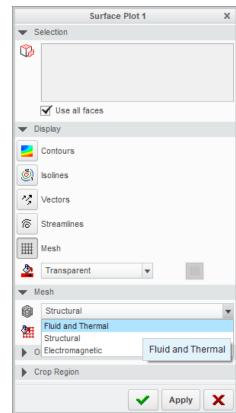# Structural: post-processing
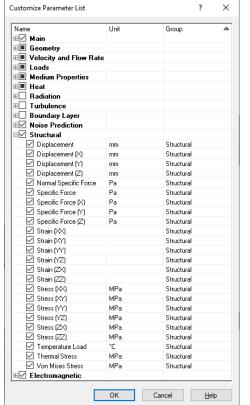
Post-processing tools for structural analysis

Results of all analyses are encapsulated within one data set and could be loaded simultaneously. Existing post-processing tools are capable of working with any type. The Parameter list is enriched with many new parameters.

Deformed shape options are in Cut and Surface Plot dialogs with the ability to set the deformation scale.

Mesh type is selected automatically in accordance with the plot parameter. If the parameter is not specified or is available for different mesh types the choice selection appears.

SIEMENS

Post-processing tools for structural analysis

New widget with plot deformation settings is added to the viewport area. <u>Deformed shape</u> option and <u>Deformation Scale</u> can be set for all plots interactively.

Deformed plots are playable with the context menu item Play!

SIEMENS

Export project to Nastran

- A Simcenter FLOEFD structural analysis project can be exported to a Nastran file. Mesh, constraints and loads (constant values or transferred from another Simcenter FLOEFD calculation) can be exported. <u>Export to Nastran</u> option requires a calculation to be run, so the most convenient method is to switch the option <u>Export project to Nastran</u> in <u>Calculation Control Option</u> dialog to invoke export automatically during solver.

- Simcenter FLOEFD can deal with Jacobians of 1e-10. Nastran solver is demanding of the FEM mesh quality and the <u>Jacobian filter</u> can be used to generate meshes keeping elements with higher Jacobian.
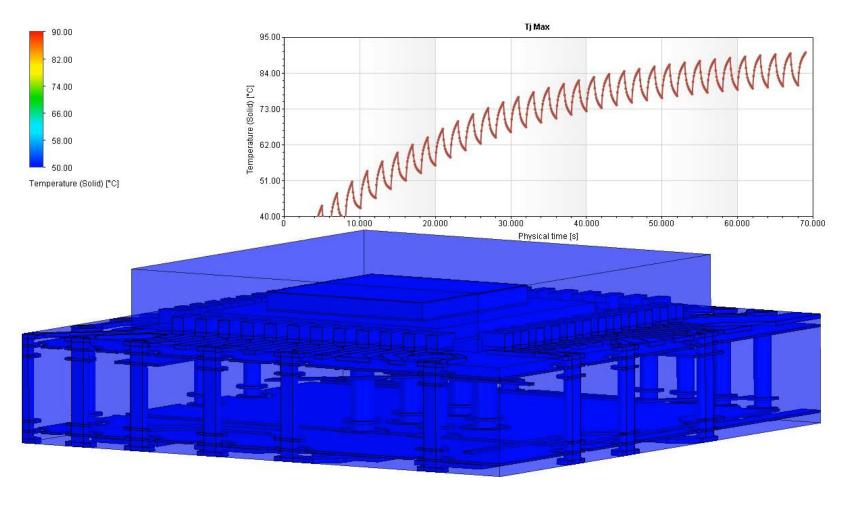
SIEMENS
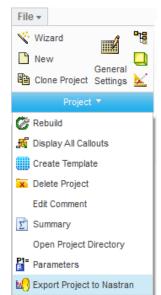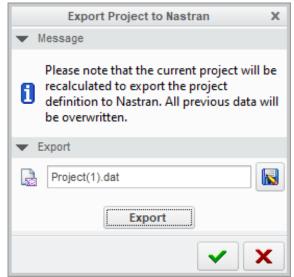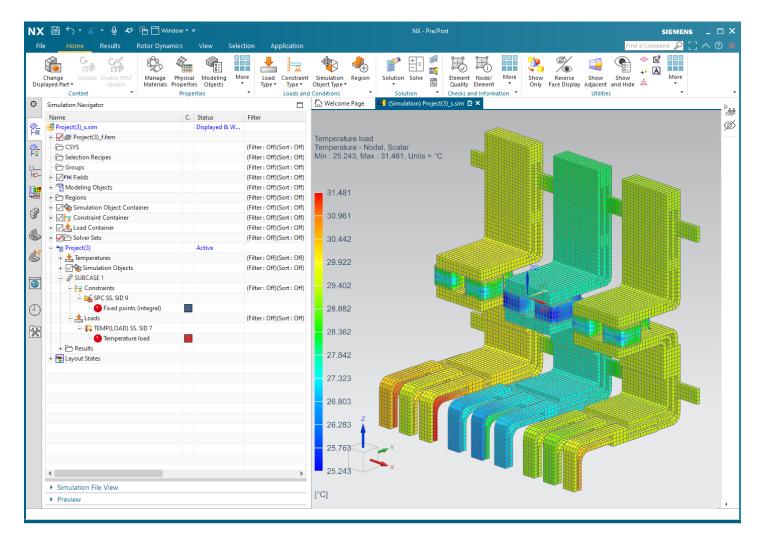
Structural: Export project to Nastran
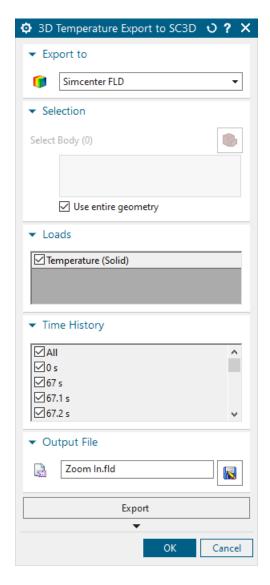
You can export a project to Nastran with the corresponding menu item in the Project menu group (dialog Export to Nastran appears). Then you can import resulting *.dat file to Nastran, change the project definition and run the Nastran solver.
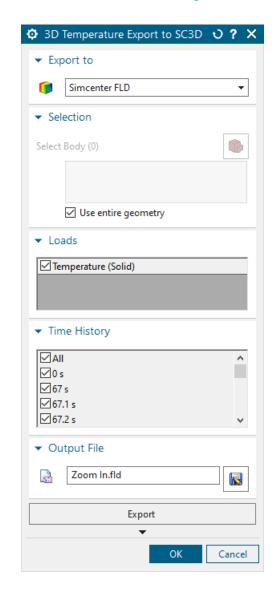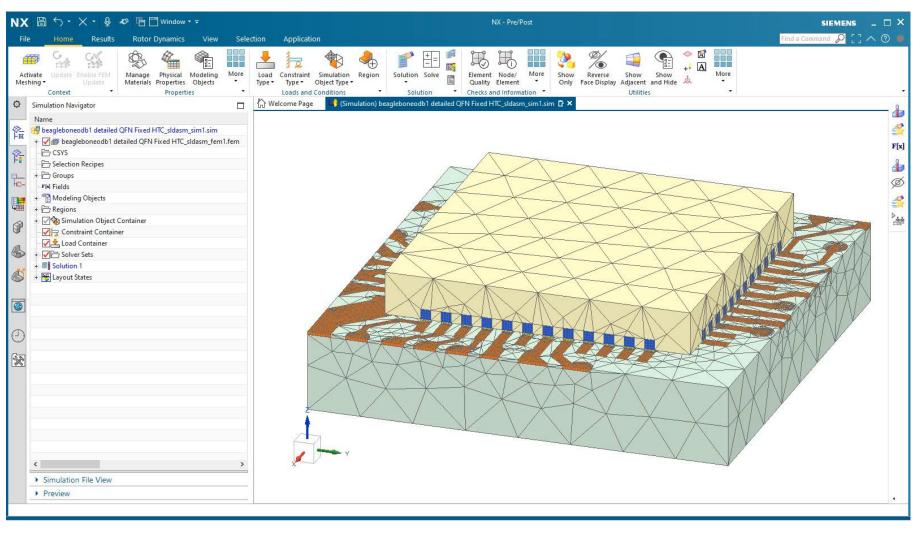
SIEMENS

Structural: export to field text format (FLD)

Simcenter FLOEFD can export time dependent temperature data to Simcenter 3D as a data field using the field (FLD) format to store steady state or transient temperature fields.

SIEMENS

Structural: export to field text format (FLD)

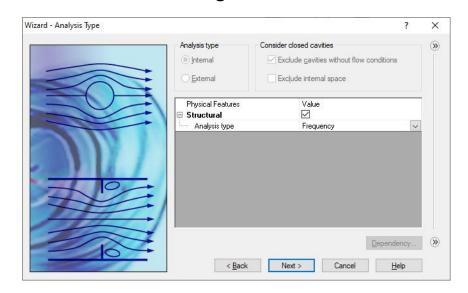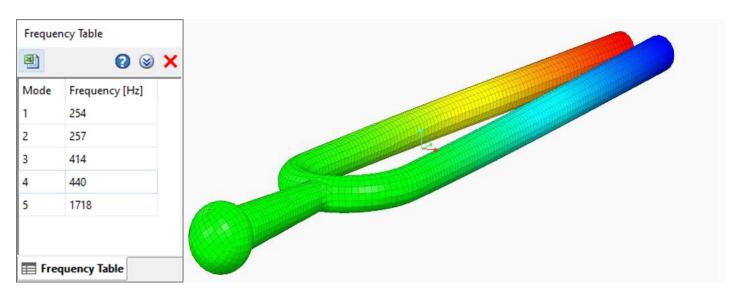Simcenter 3D allows importing the resulting FLD file to use for temperature loads

SIEMENS

Modal frequency analysis

The second available sub-type of structural analysis is <u>Frequency</u> analysis. Simcenter FLOEFD is capable of predicting eigenvalues of the non-loaded model, and to show dimensionless deformation fields for each mode. Resulting frequency tables can be exported to Microsoft Excel manually or automatically with the Batch Results Processing tool.

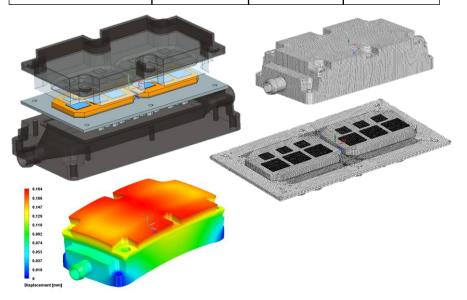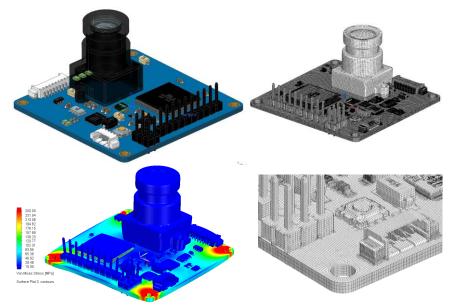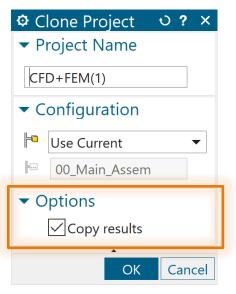Frequency table and deformation plot corresponding to the 4-th mode

SIEMENS

FE solver is much faster than FE mesher. When you need to analyze different load conditions you can significantly save time by reusing the created mesh and cloning project with results.

	Coarse	Medium	Fine
N of elements	112 615	287 845	645 792
Mesher time, s	154	248	556
Solver time, s	24	44	128

	Coarse	Medium	Fine
N of elements	343698	500898	1030429
Mesher time, s	458	776	1658
Solver time, s	20	44	135

SIEMENS

Structural: capabilities and limitation

Supported

- Linear solver only (displacements assumed to be small and the dependency of the properties on temperature is not strong)

- Stationary only

- Glue contact only, other types of contact are not supported yet

- Modal frequency analysis of the non-loaded model only, loads cannot be applied
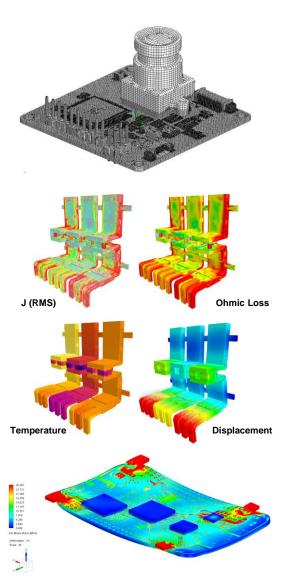
Not supported

- Non-linear phenomena

- Transient application

- Other types of contacts (sliding with friction or without, rough contact)

- Preloaded condition

- Modal analysis of the loaded model

SIEMENS

Customer Benefits

- **Automatic Hex-dominant mesh**: Accurate simulation of problems in which bending deformations dominate on rather coarse meshes due to minimizing the number of tetrahedral elements.

- **Auto Glue contact**: Auto 'glue contact' mesh technology minimizes the calculation time of projects with a large number of 'glued' surfaces and does not require specifying the contact surfaces manually, which significantly reduces the time needed to prepare for a calculation of a complex geometry.

- **All-in-one**: Direct integration of the structural analysis module and the CFD module allows you to carry out complicated simulations for a structural analysis by using the results of a CFD analysis calculation automatically, which does not require additional conversion of CFD results to external finite element analysis (FEA) software.

- **SmartPCB FEM**: A unique technology of structural analysis for PCBs based on the Smart PCB technology is implemented, that allows you to perform static and thermal analyses on multilayer boards, taking into account the traces and via layouts without the need to have them explicitly added as solid geometry in CAD.

J (RMS)   Ohmic Loss

Temperature   Displacement

SIEMENS

Electronics Cooling & Thermal Management

SIEMENS

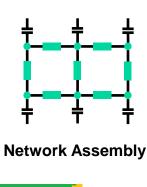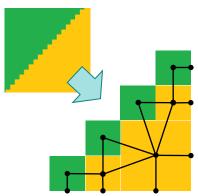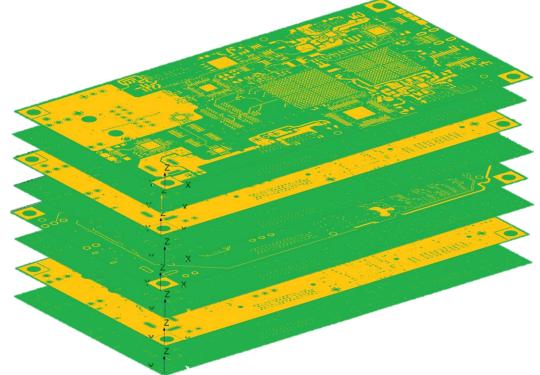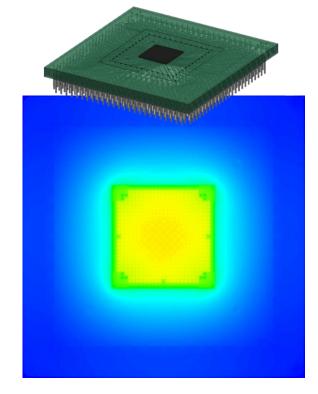# SmartPCB Enhancements

SIEMENS

SmartPCB - Overview

SmartPCB is unique approach to simulate multiphysics phenomena on the printed circuit board taking into account all nets with the maximum level of details. Simulation models and calculation meshes are generated using EDA data without the necessity to convert EDA into a CAD model explicitly, so the resulting model is simple but lossless. SmartPCB provides the high accuracy of a fully detailed (explicit) model for the calculation time of a compact model.

Network Assembly

SIEMENS
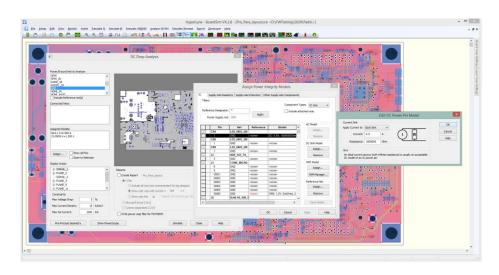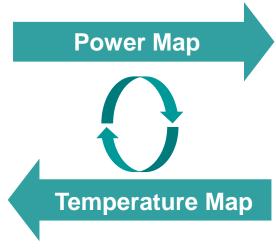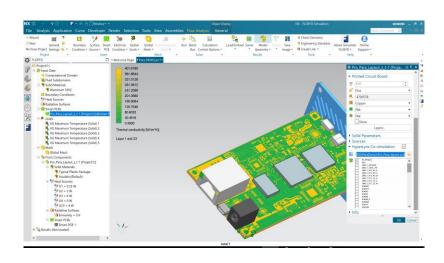
Joule Heating in PCB: Simcenter FLOEFD-HyperLynx SI PI Co-simulation

The SmartPCB model can now simulate joule heating via Simcenter FLOEFD-HyperLynx™ (v2.8.1 and newer) co-simulation. Simcenter FLOEFD can take a power map from a HyperLynx DC Drop Simulation and perform a thermal analysis, then it can return a temperature map to HyperLynx to update the electrical simulation and update the power map for Simcenter FLOEFD. Alternatively, you can just import a power map exported from a HyperLynx DC Drop Simulation into Simcenter FLOEFD for a one-way data exchange.

Power Map

Temperature Map

SIEMENS
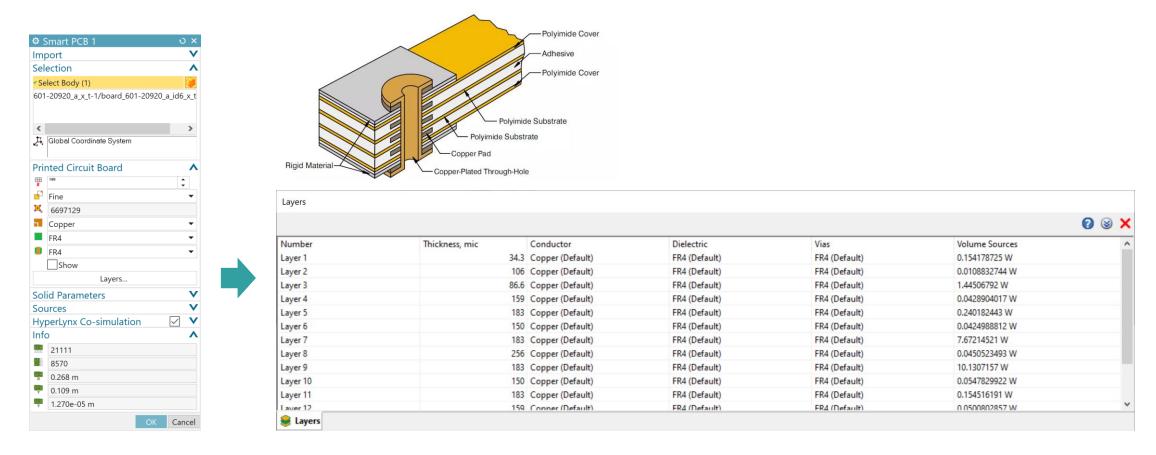
Smart PCB: Material definition for each layer

You can define materials of conductor, dielectric and via filling separately for each layer.
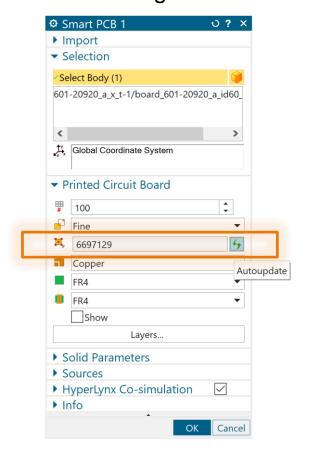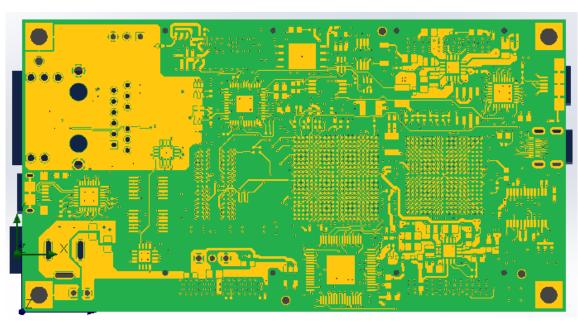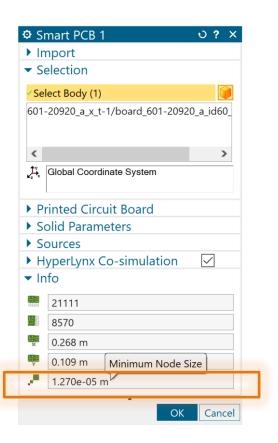The volume sources' power set for each layer is displayed.

Number	Thickness, mic	Conductor	Dielectric	Vias	Volume Sources
Layer 1	34.3	Copper (Default)	FR4 (Default)	FR4 (Default)	0.154178725 W
Layer 2	106	Copper (Default)	FR4 (Default)	FR4 (Default)	0.0108832744 W
Layer 3	86.6	Copper (Default)	FR4 (Default)	FR4 (Default)	1.44506792 W
Layer 4	159	Copper (Default)	FR4 (Default)	FR4 (Default)	0.0428904017 W
Layer 5	183	Copper (Default)	FR4 (Default)	FR4 (Default)	0.240182443 W
Layer 6	150	Copper (Default)	FR4 (Default)	FR4 (Default)	0.0424988812 W
Layer 7	183	Copper (Default)	FR4 (Default)	FR4 (Default)	7.67214521 W
Layer 8	256	Copper (Default)	FR4 (Default)	FR4 (Default)	0.0450523493 W
Layer 9	183	Copper (Default)	FR4 (Default)	FR4 (Default)	10.1307157 W
Layer 10	150	Copper (Default)	FR4 (Default)	FR4 (Default)	0.0547829922 W
Layer 11	183	Copper (Default)	FR4 (Default)	FR4 (Default)	0.154516191 W
Layer 12	159	Copper (Default)	FR4 (Default)	FR4 (Default)	0.0500802857 W

SIEMENS

SmartPCB: Update on demand and minimum node size.

You can update the evaluated number of nodes on demand. This prevents an auto-update and can dramatically save editing time for extremely complicated PCBs. Minimum node size gives you the actual resolution of PCB.

SIEMENS
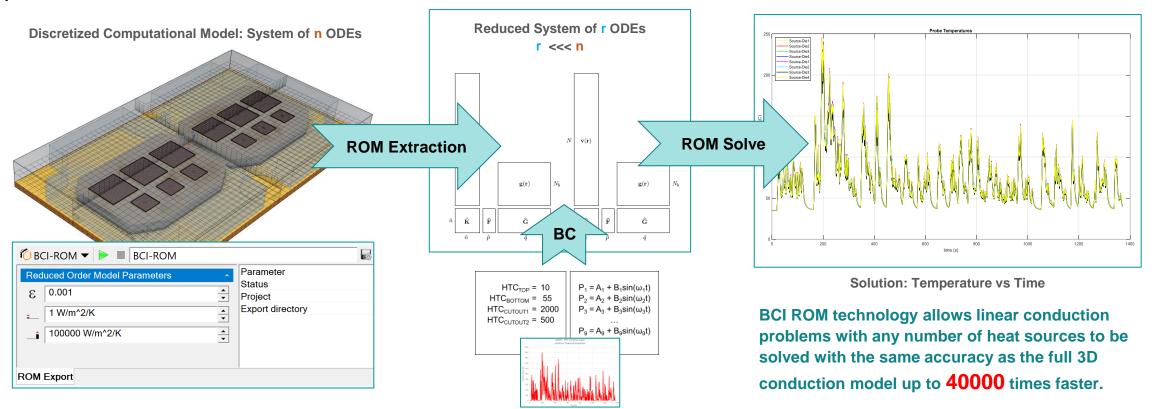
BCI-ROM Enhancements

BCI-ROM - Overview

Reduced Order Modelling is an approach to derive a dynamic compact thermal model from a thermal simulation model. The objective is to create a model that solves much faster, while maintaining predictive accuracy in space and time.

Discretized Computational Model: System of n ODEs

Reduced System of r ODEs
r <<< n

ROM Extraction

ROM Solve

BC

HTC$_{TOP}$ = 10
HTC$_{BOTTOM}$ = 55
HTC$_{CUTOUT1}$ = 2000
HTC$_{CUTOUT2}$ = 500

$P_1 = A_1 + B_1 sin(\omega_1 t)$
$P_2 = A_2 + B_2 sin(\omega_2 t)$
$P_3 = A_3 + B_3 sin(\omega_3 t)$
...
$P_9 = A_9 + B_9 sin(\omega_9 t)$

Solution: Temperature vs Time

BCI ROM technology allows linear conduction problems with any number of heat sources to be solved with the same accuracy as the full 3D conduction model up to 40000 times faster.

SIEMENS

Running BCI-ROM on Linux

BCI-ROM FMUs can now be solved on Linux:

- As an FMU model via any FMI 2.0 Co-Simulation compatible tool (32 & 64-bit).
- As matrices via Results Exporter (64-bit only).

Results Exporter provided for Linux platforms also allows to pack BCI-ROM matrices into an FMU file, which is compatible with Windows and Linux (32 & 64-bit).

SIEMENS

BCI-ROM VHDL-AMS export and import
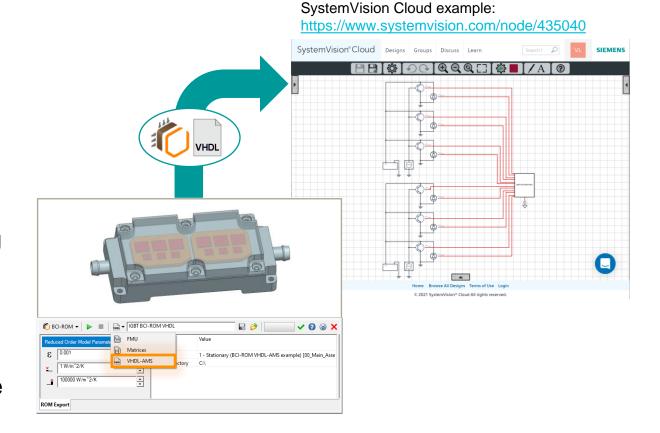
Now both reduced order models (Thermal Netlist and BCI-ROM) can be exported in VHDL-AMS format, which can be imported into SystemVision® Cloud.

To generate the model simply change Save As option to VHDL-AMS prior the export. Two files will be created as the result: *.vhd and *.csv.

Import BCI-ROM VHDL into SystemVision Cloud using the same method as with a Thermal Netlist and when requested to import a file, proceed to import the generated *.csv file.

Next, connect thermal power and ambient temperature ports, enter thermal resistance values for the external surfaces and run the simulation.

SystemVision Cloud example:
https://www.systemvision.com/node/435040

SIEMENS

Heat Sources

SIEMENS
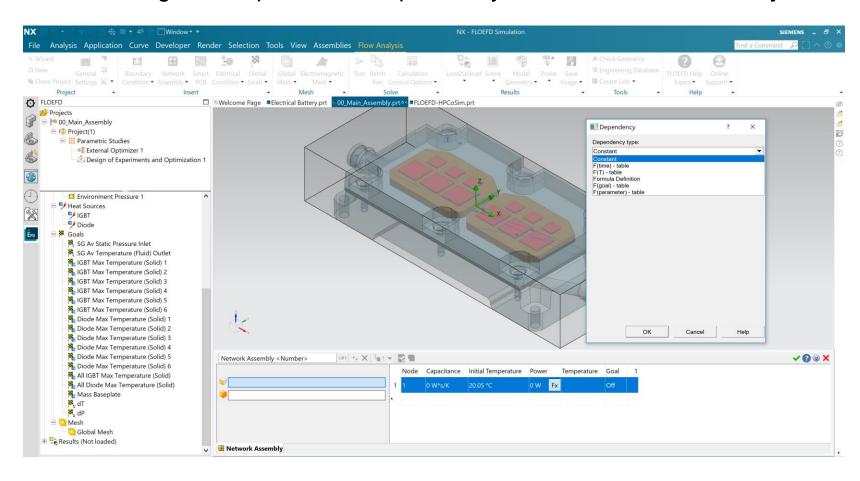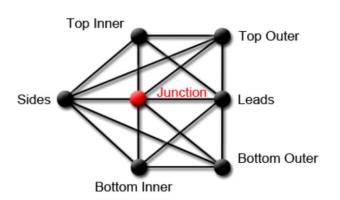
Added time, goal and parameter dependency for the Network Assembly's node heat power.

SIEMENS
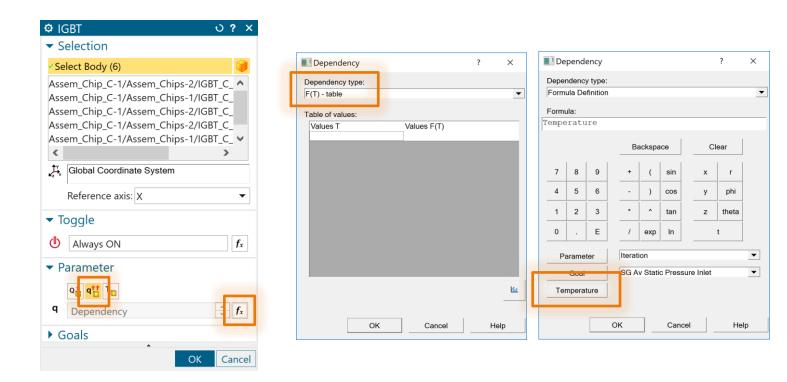
Heat Source: Dependency on local (cell) temperature

Specific power [W/m3] can be dependent on the temperature of the cell.

This dependence can be used for human body thermal analyses, for example with the Pennes bioheat equation.

SIEMENS

Dependency: Range function

A new range function allows you to get goal values at any moment of a calculation. This facilitates modeling complex behavior of a transient system, for example power derating based on a temperature sensor.
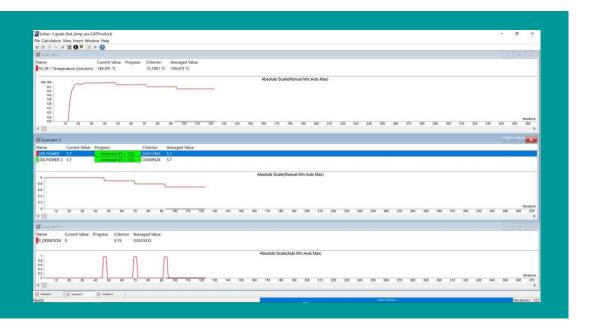
range (P1; P2(;P3;..PN); E1; E2) – array of values at the specified range. Parameters:

P1 – parameter on which to evaluate the array, i.e. {Time}, {Iteration}, {Travel},

P2…PN - goals, E1 – start period, E2 – end Period.

An example of transient derating with the range() function:

- Temperature goal is checked from iteration 20.
- Calculates average T at 20 .. 30 period = Tav
- Check deviation for each T(it) from Tav at this interval
- If all deviations are within 0.5 C then T is converged = Tcnv
- Check it Tcnv is higher than Tcritical (here = 150 C)
- If Tcnv > Tcritical, decrease Power for predefined step (0.1W)
- Do not check T for the next 20 iterations
- Check again and repeat

SIEMENS
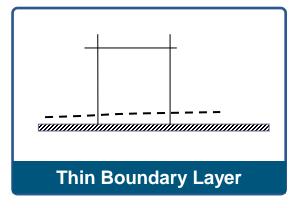
Technology Improvements

SIEMENS
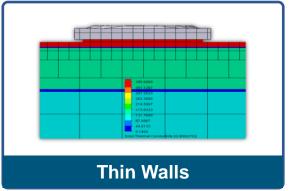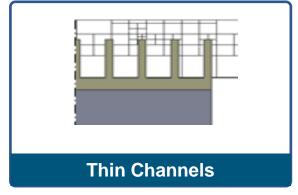
SmartCell Technology - Overview
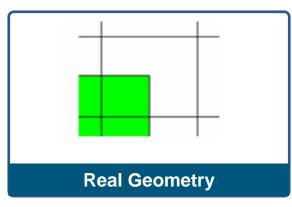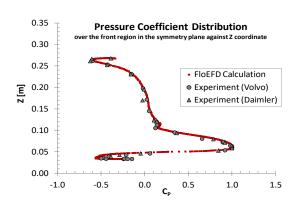
Simcenter FLOEFD automatically applies appropriate Analytical or Empirical solutions in places there mesh resolution is not enough for an accurate Numerical solution. As a result the mesh setting is easy and fast, and calculation time is dramatically decreased because accurate results can be achieved with a coarser mesh.

| **Thin Boundary Layer** | **Thin Walls** | **Thin Channels** | **Real Geometry** |

Pressure Coefficient Distribution over the front region in the symmetry plane against Z coordinate

- FloEFD Calculation
- Experiment (Volvo)
- Experiment (Daimler)

	1	**2**	**3**
N (mesh cells)	252 245	610 607	966 193
Δt_{water}	14.64	14.79	14.77
ΔP_{water}, Pa	270	277	276
Q_{water}, W	98.26	98.42	98.14

Static Pressure

Inlet Mass Flow

Accurate even with coarse mesh

SIEMENS

Improved SmartCell "Thin Channels" technology

The Thin Channel technology is improved.
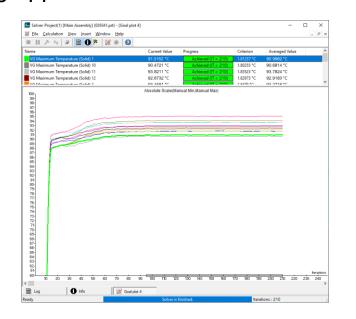
Case	Total time for the study	Mesh for design point
Fine mesh: Numerical only (No "Thin" models)	4 hours	3 000 000
SmartCell: Thin Walls and Thin Channels	**5 min (!)**	270 000

SIEMENS
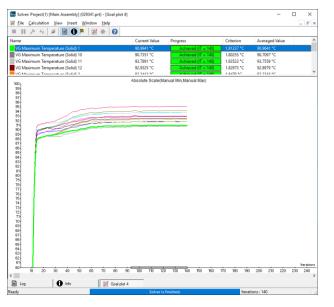
Defaults are optimized for EC applications: Travel

By default, when goals are used as finishing criteria for the solver, the goals are checked for convergence only after one Travel is reached. The Travel is a characteristic number of iterations a solution should achieve to ensure a good automatic strategy of stopping the calculation. The default travel is now optimized for electronics cooling applications to avoid over-calculating due to an unnecessarily high default maximum travel requirement. An electronics cooling application is defined as a task with conduction, gravity and with ambient velocity set to zero.
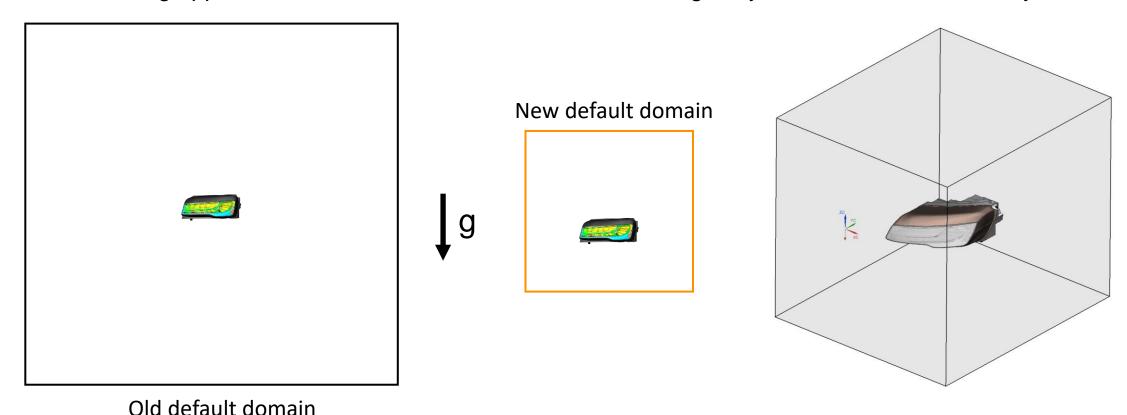
If goals are converged calculation of a EC task finishes earlier in 2021.1 (right) because of adjusted default travel criteria.

SIEMENS

Defaults are optimized for EC applications: Domain size
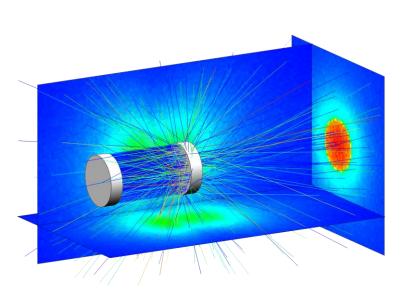
The default computational domain size is decreased to fit a typical electronics cooling external analysis. The change will make the calculation time shorter while keeping high accuracy if default settings are used. An electronics cooling application is defined as a task with conduction, gravity and with ambient velocity set to zero.

New default domain

g

Old default domain

SIEMENS

Optics and Lighting

SIEMENS
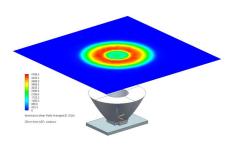
Lighting, optics, and radiation in one glance
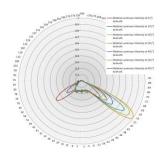
LED Electro-Thermal-Optical model

Near Field Illuminance
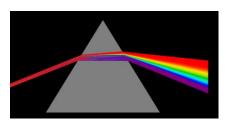
Far Field Intensity
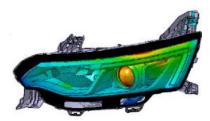
Solar Radiation
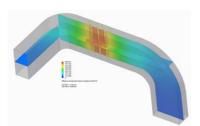
Lidar Thermal-Optical Analysis
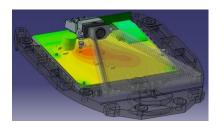
Optics

Lamps, Lights, Bulbs
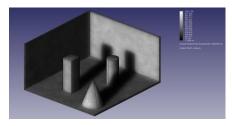
Ultra Violet Germicidal Irradiation
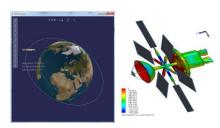
Camera Thermal and Fog Simulation

Light Guides

Light Distribution

On-Orbit Radiation

SIEMENS
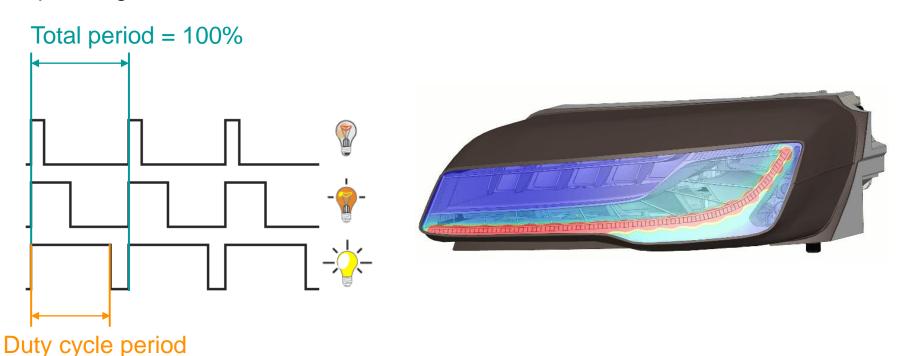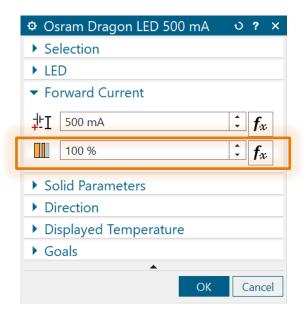
LED: Pulse width modulation and forward current goal

Pulse-width modulation can be modeled for thermal-electrical-optical LEDs. Pulse-width modulation is widely used in lighting to control the brightness of LEDs. You can set the duty cycle in percentage.

You can now set Forward Current as a LED specific goal to use the driving current as a parameter for other equation goals.
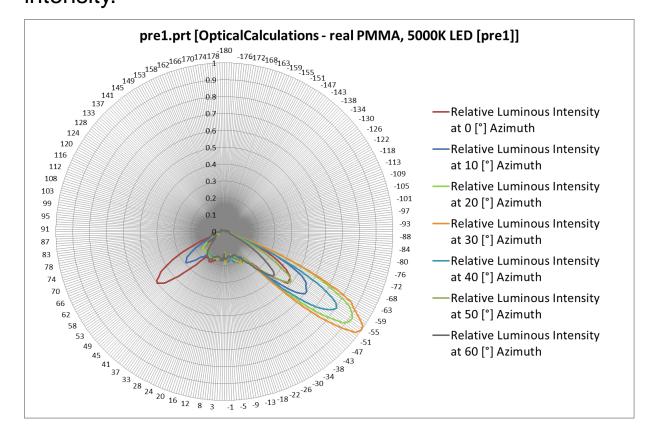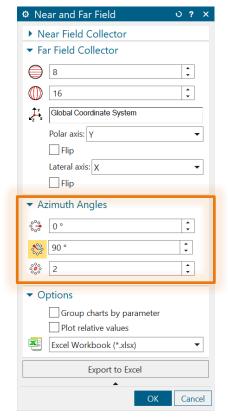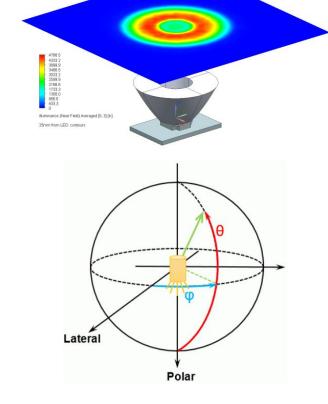
Far-Field plot: Multiple angle output and candela unit.

Relative or absolute luminous intensity values can now be plotted for many user-defined Azimuth (φ) angles. The candela (cd = lumen per steradian, luminous flux per unit solid angle) unit is added for the luminous intensity.
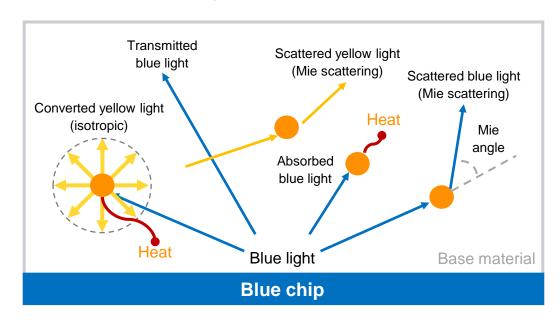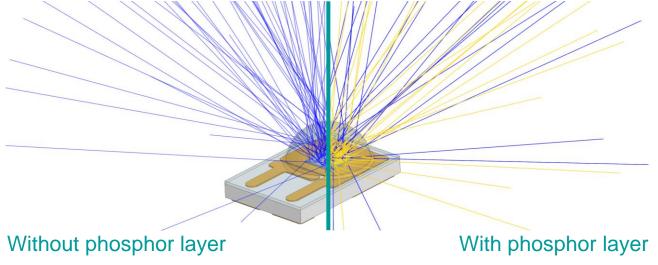
Phosphor Particles

Photoluminescence and Scattering (Mie scattering theory is used) of phosphor particles can now be simulated. Phosphor particles are often used to manufacture white LEDs, as they allow the conversion of blue light to yellow light. Photoluminescence is the process during which light of a specific spectral range is absorbed by a material and is then re-emitted with a different spectral range of longer wavelengths. Temperature dependency of the luminous efficiency is taken into account.

Without phosphor layer

With phosphor layer

SIEMENS
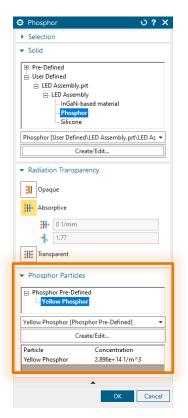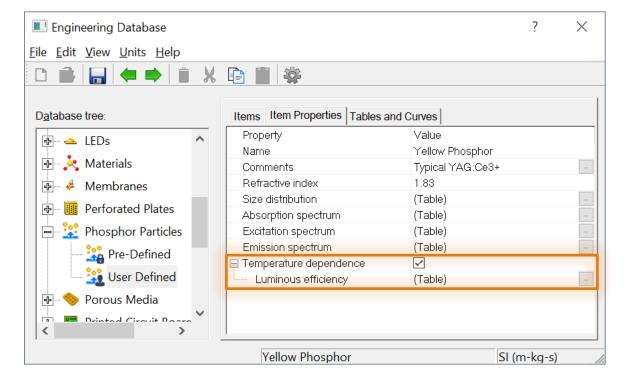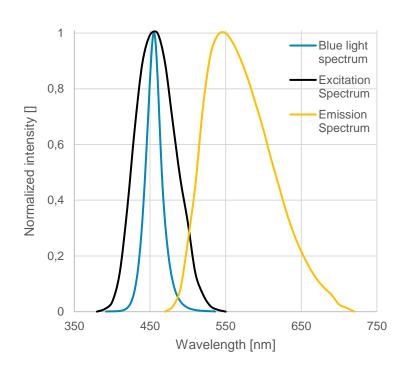
Phosphor Particles: Temperature Dependence

Luminous Efficiency is an optional parameter, which accounts for the temperature and excitation wavelength dependency. This parameter modifies the conversion efficiency of the excitation spectrum according to the actual solid material temperature and wavelength of incident light.
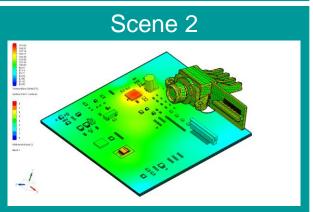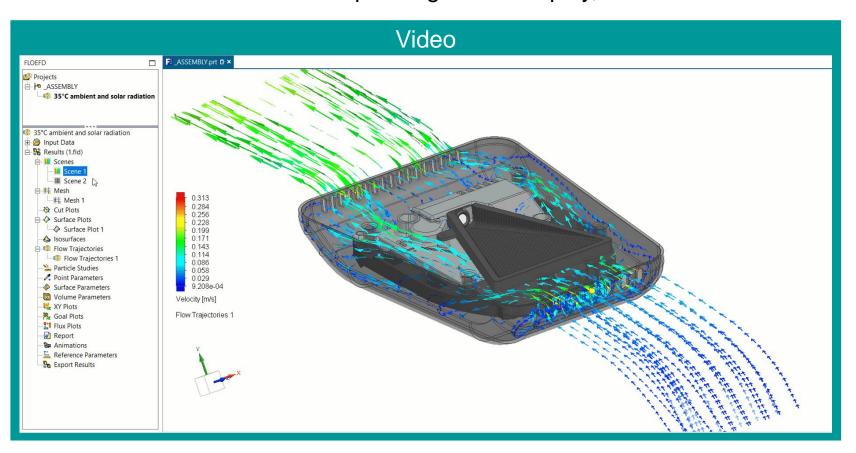
Usability

SIEMENS
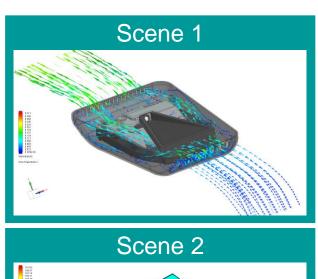
Scene Plot

A new scene plot remembers all displayed plots, model orientation, zoom and part visibility. Switching between scenes displays the plots saved in the scene and sets the corresponding model display, zoom and orientation.

Scene 1

Scene 2

Video
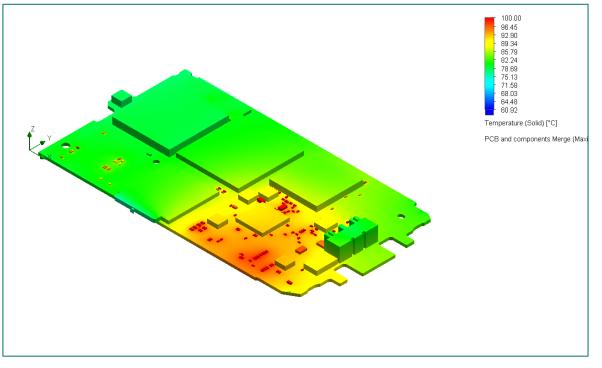
Compare: Merged plots

You can merge compared plots from different projects into one image to see critical results at a glance. Contour plots made for several design cases or several power modes might be merged, for example, to a merged plot based on maximum value which can show the maximum temperature for all design cases in one picture.

Original plots
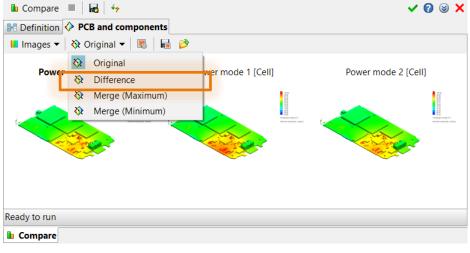
Merged plot (by Maximum)

SIEMENS
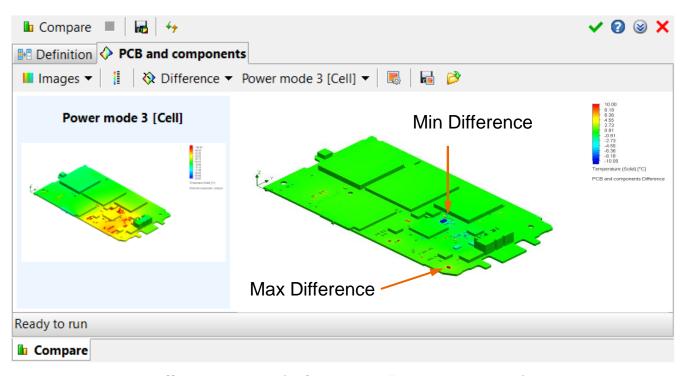
Compare: Difference plot

You can create a difference plot. The difference plot graphically shows the difference from one particular case to the reference case.

Original plots
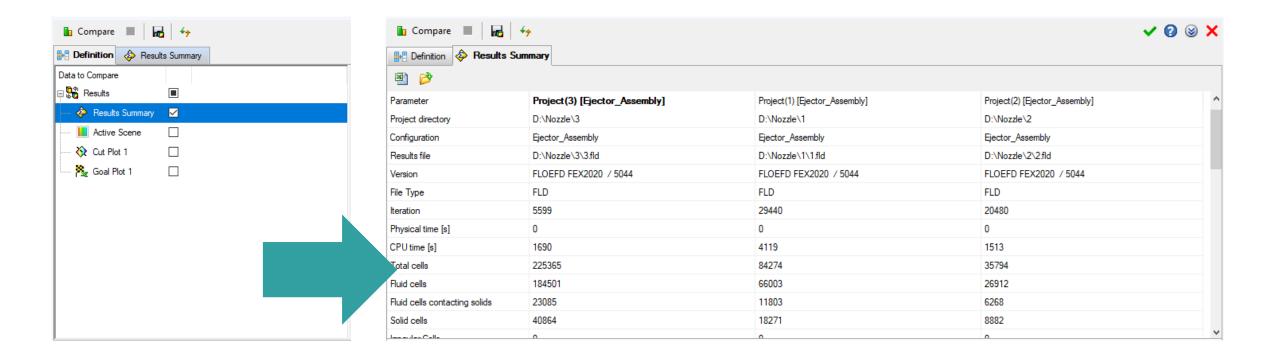
Difference plots (reference is "Power mode 3")

Results Summary comparison

Results Summary feature can be nominated in the Compare tool and Parametric Study now in order to compare mesh information, iteration count, calculation time and other parameters of all currently opened projects. The resulting table can be exported to the Microsoft Excel.

SIEMENS
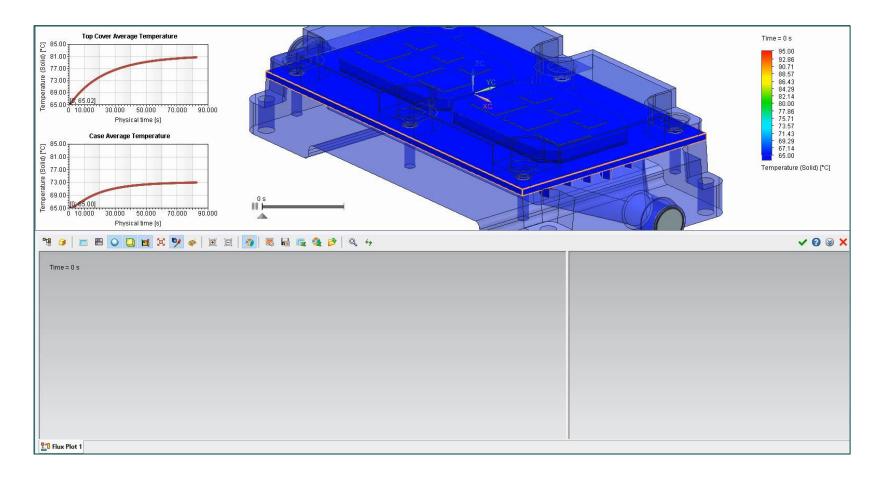
Flux Plot can visualize heat fluxes between components in Transient Explorer mode.

Updated Solver Info file

Solver info log (.info) file is reorganized, now has mesh information and can be created for Linux runs and in case CAD is closed.

Info file is now generated in two human and machine-readable formats:

- *.info
- *.info.json

Info files are updated at each iteration and include useful information, such as:

- Status of the solver
- Mesh count
- Time (Preparation, Solver CPU, Physical, etc.)
- Goals and their values
- Project details
- System information

SIEMENS

Other Improvements

- **Selection: Easy to handle lost or suppressed bodies**. Reference geometry (faces, edges, points) of missing or suppressed bodies can be automatically removed from the selection.

- **Surface Parameters: Crop Region**. Crop Region is taken into account when evaluating surface parameters. You can use Crop Region to evaluate parameters for a portion of a surface.

- **Goals: Adding Equation Goal after calculation**. You can now add an equation goal based on defined goals after a calculation.

- **Probes: Copy probes to project**. Probes are now copied to projects together with the plots used to define the probes.

- **Save As: Named Views and default settings**. Named Views are supported in Save As view orientation. You can adjust default settings for the background color of the saved images under Tools, Options.

- **New Microsoft Office support**. MS Office 365, 2019 and 2016 support (cancel support of MS Office 2010, 2007).

SIEMENS
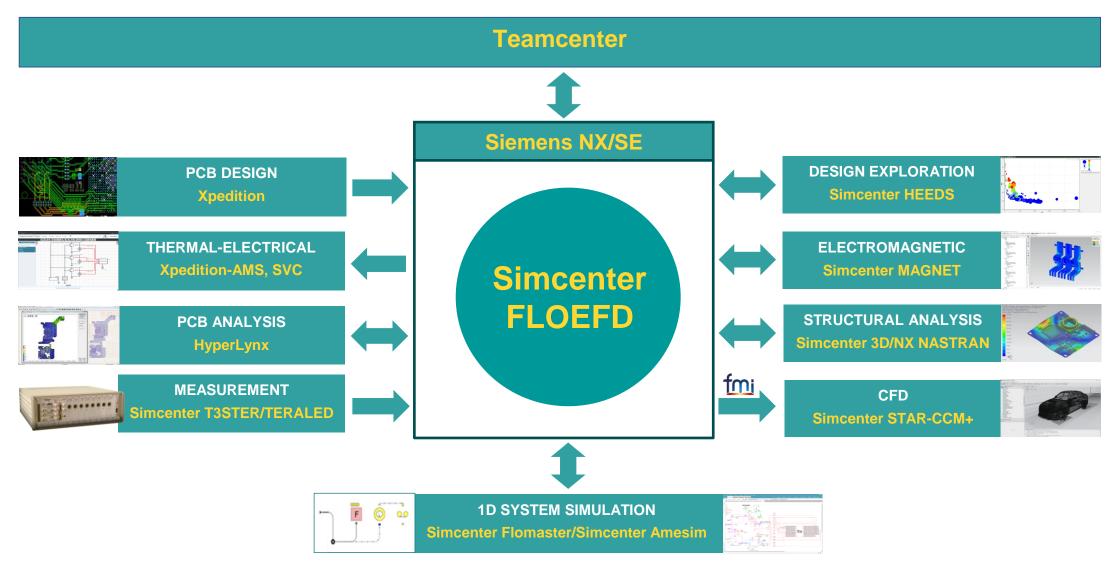
Interoperability

Simcenter FLOEFD is tightly integrated with other Siemens tools

Teamcenter

Siemens NX/SE

Simcenter FLOEFD

PCB DESIGN
Xpedition

THERMAL-ELECTRICAL
Xpedition-AMS, SVC

PCB ANALYSIS
HyperLynx

MEASUREMENT
Simcenter T3STER/TERALED

DESIGN EXPLORATION
Simcenter HEEDS

ELECTROMAGNETIC
Simcenter MAGNET

STRUCTURAL ANALYSIS
Simcenter 3D/NX NASTRAN

CFD
Simcenter STAR-CCM+

1D SYSTEM SIMULATION
Simcenter Flomaster/Simcenter Amesim
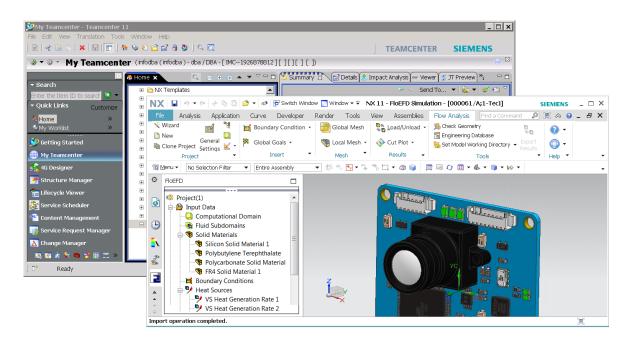
SIEMENS

Teamcenter

SIEMENS

Teamcenter (TcSim) Support for Simcenter FLOEFD for NX

Simcenter FLOEFD supports TcSim CAE Model to manage projects and results within Teamcenter.
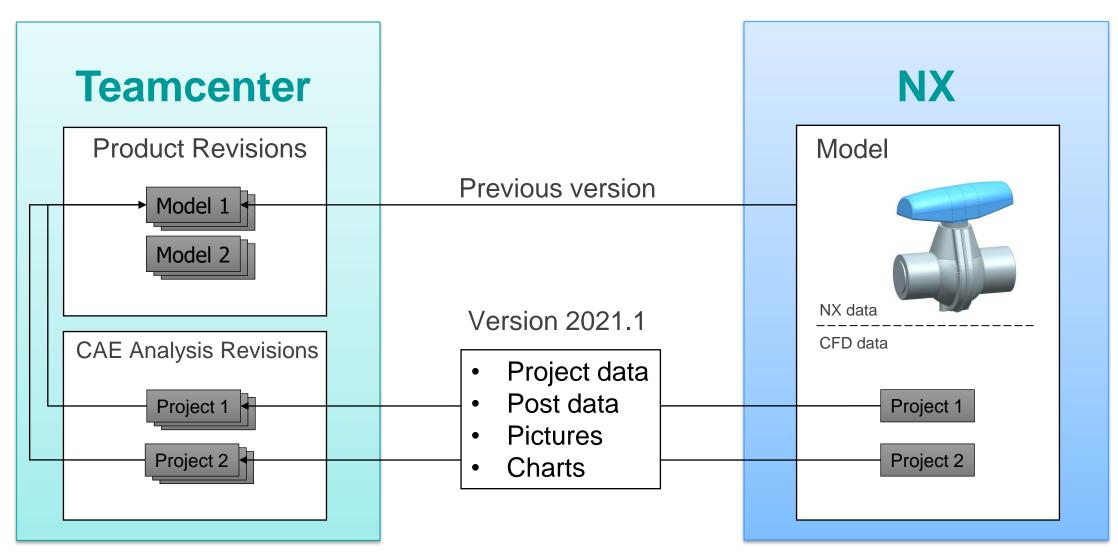
Simcenter FLOEFD projects were stored inside the CAD model files for all previous Simcenter FLOEFD versions but results files (binary, pictures, spreadsheets, docs) were stored in the local working directory. Further integration (Simcenter FLOEFD 2021.1) creates separate TC items for each Simcenter FLOEFD project to be able to track their revisions, to store results inside their data sets and to involve them into the design workflow.
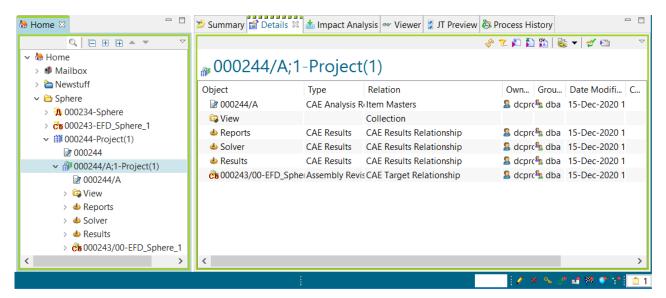
Simulation model

Simcenter FLOEFD projects are collected in the NX assembly item as they were in previous versions. To keep the original designer geometry unchanged a new assembly could be added here and the original assembly could be inserted into the new one as a subassembly or WAVE linked.

New items for Simcenter FLOEFD projects are common CAE Analysis items, which refers the main NX assembly with a **CAE Target relationship**.

SIEMENS

Teamcenter integration

Server side customization and limitations

Customization on the server side is not required, all changes were made inside the client application. There is no special item for Simcenter FLOEFD projects so here are some limitations:

- Simcenter FLOEFD projects can be created, edited or removed through Simcenter FLOEFD for NX only. It is not possible to do it through Teamcenter
- Simcenter FLOEFD projects cannot be opened from Teamcenter, the main NX assembly should be opened instead

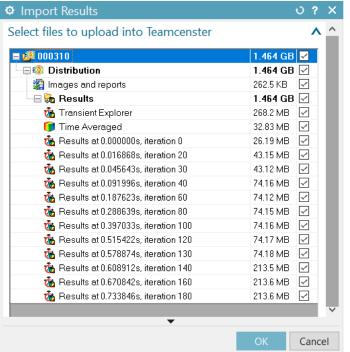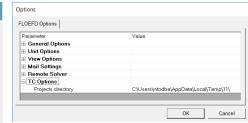SIEMENS

Teamcenter integration

Working directory

A local working directory is needed to store project data and results files while the project is open or being calculated on the client computer, but the workflow is enhanced as follows:

- It is not necessary to specify the folder every time, the default value is set from the global options dialog

- Files are stored in the local folder temporarily and would be upload into Teamcenter when model is saved

- The list of uploaded files could be customized with Import Results dialog which appears automatically if model is saved

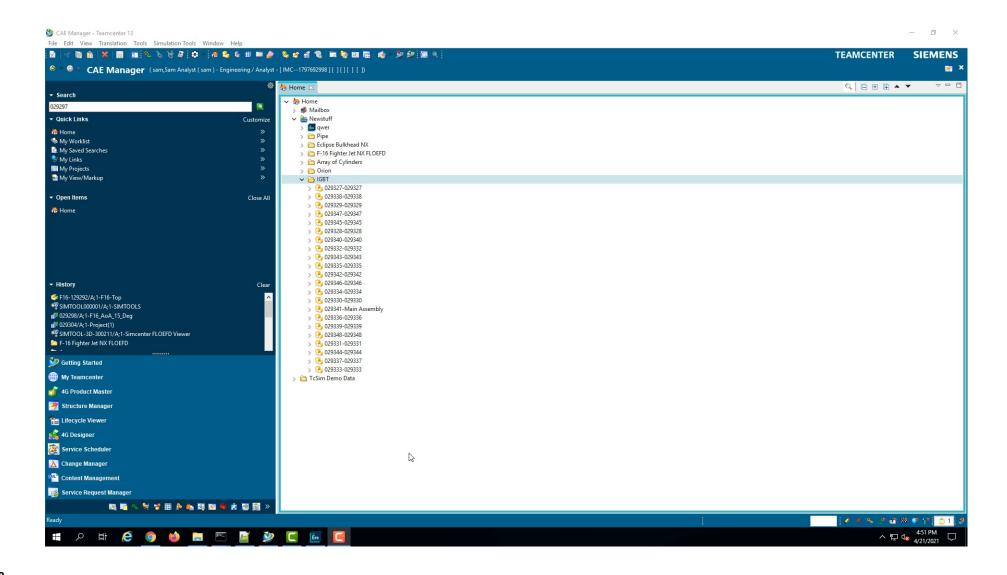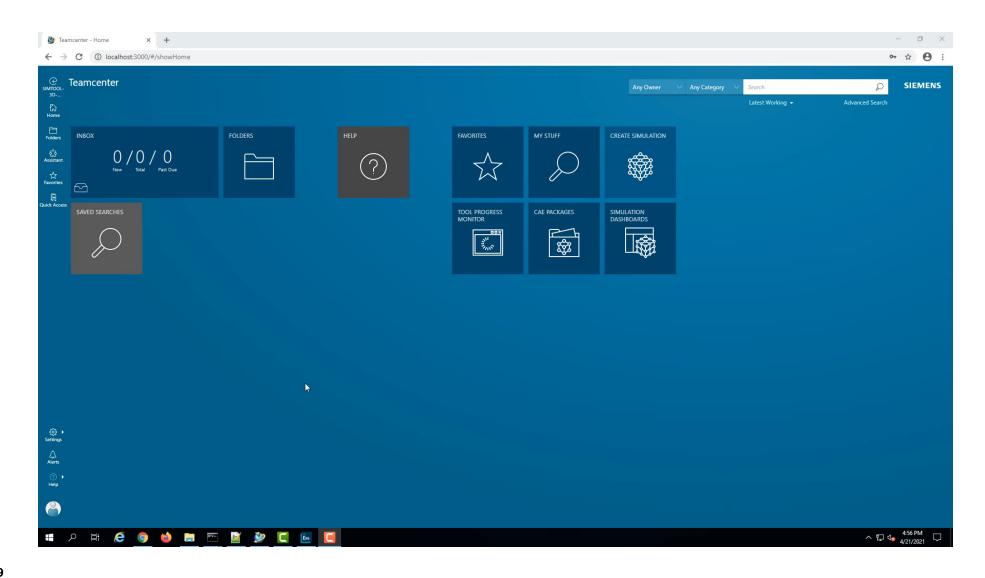SIEMENS
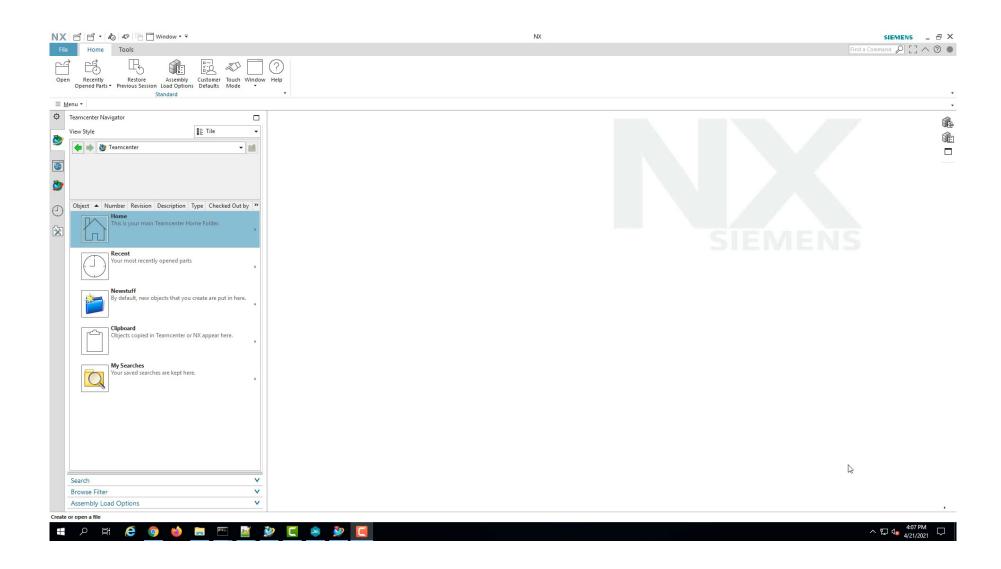
Teamcenter integration: example
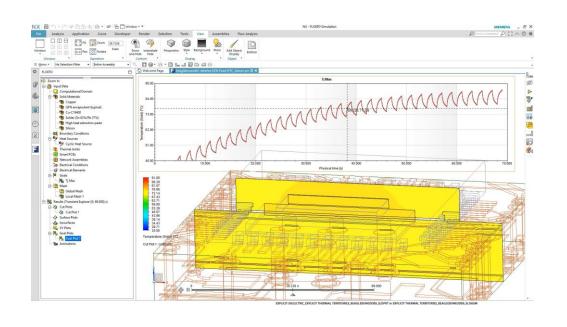
Teamcenter integration: example

Export to Field Text Format (FLD)
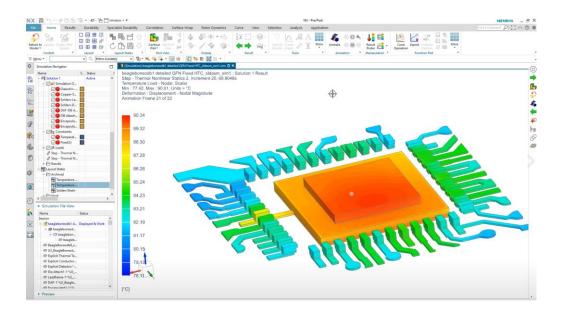
Interface to Simcenter 3D / NX Nastran

You can now export Simcenter FLOEFD steady-state and transient results into FLD format for performing advanced thermo-structural analyses in Simcenter 3D such as non-linear visco-plastic creep analysis.

SIEMENS

CAD Integration

SIEMENS
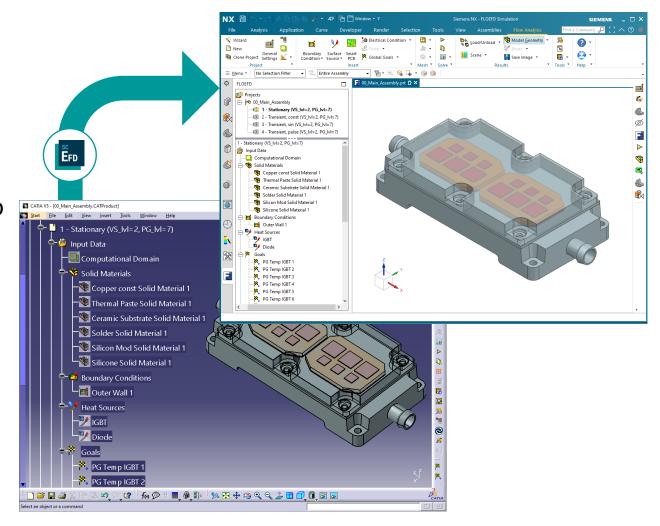
CATIA to NX model and project converter enhancement

Updated CATIA™ to Siemens™ NX™ converter is now capable of skipping the geometry conversion step. This allows you to easily copy projects from one product to another for already existing geometry.

To copy projects, simply add two new parameters to your conversion script:

- /skip_geometry_convertation – allows user to skip conversion step.
- /output_model_name:<path_to_NX_model> - directory to the Siemens NX model where the projects will be copied to.

SIEMENS

CAD specific enhancements

- **EDA Bridge: Save PCB parts into a folder for NX**. Parts generated by EDA Bridge can be saved into a specific folder. Under Tools, Options you can specify to use either the default folder or a custom folder. By default all parts are saved into the folder of 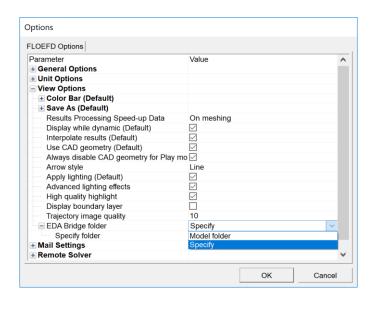the model. Using the specific folder requires adjusting NX settings as follows: Assembly Load Options, Part Versions, Load: As Saved; Scope, Load: All Components; Option: Fully Load. Leaving the specific folder empty will put all parts into the project folder.

- **Check Geometry improvement**. For the "Improve Geometry Handling" mode you can now create solid and fluid bodies (not available for CATIA V5).

- **Siemens NX new versions support**. Support of new versions: 1957, 1961, and 1965 versions.

- **Creo 7.0**. Creo 7.0 is supported.

- **CATIA V5 R30**. CATIA V5-6 R2020 is supported.

SIEMENS

New Product: Simcenter FLOEFD SC
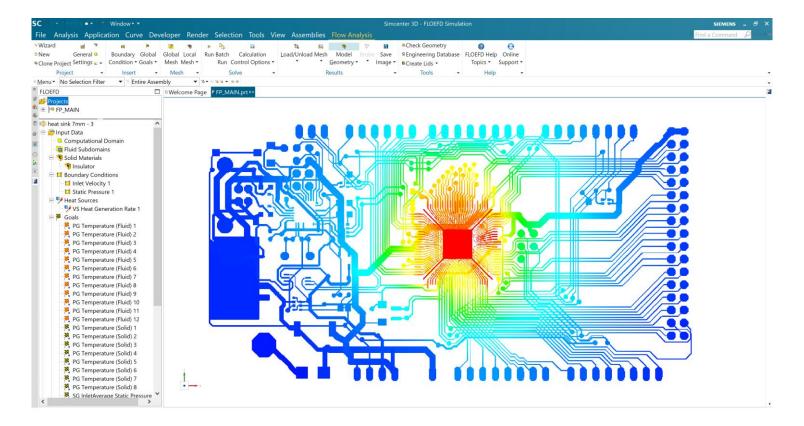
SIEMENS

Simcenter FLOEFD SC

"Simcenter FLOEFD SC" is a new Simcenter FLOEFD product available for Simcenter 3D users (starting from 2019.2-1880 version).

Product number: **MG284878FL**

License feature: **efdsc**
(not included in any flexx licenses except for Ultra_flexx)

SIEMENS

Licensing

SIEMENS

Licensing update

- **Licensing Timeout option.** For MLS Simcenter FLOEFD supports FlexLM timeout option, so the server automatically releases a license after a defined time (in seconds) of license client inactivity. A FlexLM option file can be defined by creating a text file in the same folder as the license file with the name of the vendor daemon and the suffix ".opt", i.e. "mgcld.opt". Among other option keywords TIMEOUT and TIMEOUTALL are now supported. Minimum time of inactivity is 900 seconds. Individual TIMEOUT settings can be defined (e.g. TIMEOUT efdnx 1200).
  For SPLM as an alternative to the server based timeout option a client based setting can be used by defining an EFD_SPLM_IDLE_TIMEOUT environment variable with the time in seconds, e.g. EFD_SPLM_IDLE_TIMEOUT = 1200.
  The timeout capability is not available for Simcenter FLOEFD for CATIA V5 due to CATIA limitations.
- **Licensing update.** The "Membrane" feature is added to the "Power Electrification" module. "DO radiation" feature is added to the "Advanced" module.
- **New module Simcenter FLOEFD EMAG (MG 285169FL)**. The module enables electromagnetics.
- **New module Simcenter FLOEFD Structural (MG285168FL)**. The module enables linear stress analysis.
- **New product Simcenter FLOEFD SC (MG284878FL)**. Simcenter FLOEFD available for Simcenter 3D users.

SIEMENS