

SIEMENS

Ingenuity for life



**GEOMETRIC
SOLUTIONS**

Simcenter 3D Advanced Fluid Modeling

Speeding the preprocessing time for CFD analysis

Benefits

- Reduces time spent creating FE fluid domain models for CFD and acoustics simulation
- Rapidly creates fluid domain geometry from complex assemblies
- Accurately meshes fluid domain and boundary layers
- Quickly updates analysis models to changes in the design geometry
- Synchronizes analysis iterations with design to provide timely feedback on critical decisions
- Preprocesses and exports fluid models to prominent third-party solvers
- Full Assembly FEM support to extract air volume in complex assembled systems representations

Summary

Simcenter™ 3D Advanced Fluid Modeling software delivers state-of-the-art tools to efficiently create complex finite element (FE) fluid models used for either computational fluid dynamics (CFD) or acoustics simulation. Simcenter 3D Advanced Fluid Modeling enables you to rapidly create fluid domain geometry from complicated component or assembly models along with specialized boundary layer meshing capabilities that will deliver more accurate results. Being integrated in the Simcenter 3D platform also means that the fluid analysis model is completely associated to the design geometry, enabling rapid updates and quicker design-analysis iterations.

Simplify the fluid modeling process

Analysts have traditionally spent far too much time on the tedious task of creating the FE fluid model needed for CFD or acoustics simulation. That is because the fluid domain is represented by the void within a particular part or space within an enclosed assembly that might

have many components inside, such as an electronic device. The fluid domain is not something that is explicitly modeled by the design team, and instead requires the analyst to model it from scratch based on the part or assembly geometry.

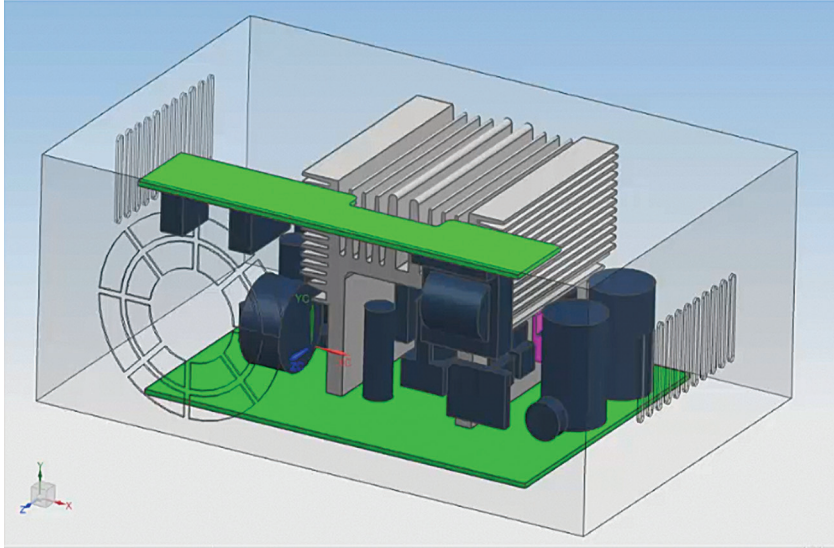
Traditional CFD preprocessors don't have the robust geometry capabilities to tackle fluid domain geometry creation, and computer-aided engineering (CAE) analysts don't have the computer-aided design (CAD) expertise to create the fluid domain geometry efficiently. So creating the fluid domain for complex parts or assemblies tends to become a multi-step, multi-tool and sometimes a multi-person endeavor that can last days or weeks. If the design changes, then the analysis model quickly becomes out of sync with the latest design iteration due to the lag time required to create the analysis model.

Simcenter 3D Advanced Fluid Modeling solves this issue and greatly simplifies the CAE fluid modeling process so that a single engineer can create a ready-to-run fluid analysis model in a matter of minutes, keeping the analysis model synchronized with the latest design to help an analyst provide critical and timely feedback.

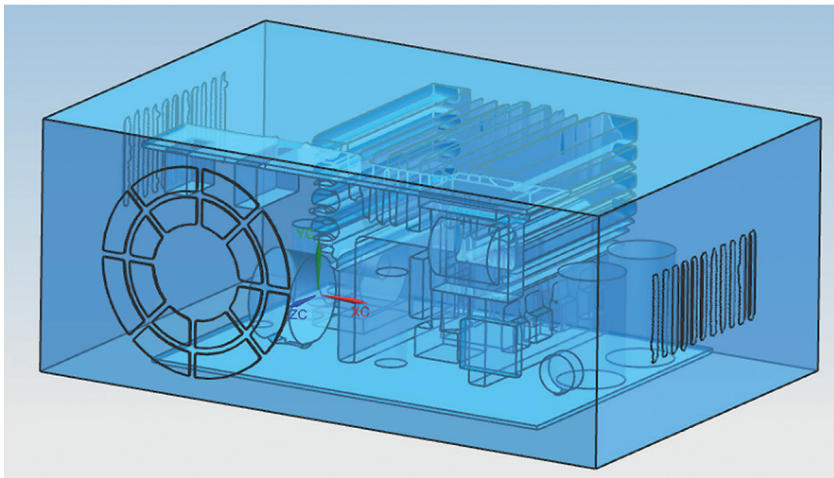
Creating fast fluid domain geometry

Simcenter 3D Advanced Fluid Modeling is built on the Simcenter 3D integrated platform, which means it leverages the powerful geometry foundation at the heart of Simcenter 3D, such as synchronous technology for fast and intuitive direct geometry editing. In addition to

Simcenter 3D Advanced Fluid Modeling



Design geometry of an electronic device with interior components.



Fluid domain geometry of interior space of electronic device created using surface wrapping.

these CAD-level geometry capabilities, Simcenter 3D Advanced Fluid Modeling delivers tools specific to creating fluid domain geometry, such as surface wrapping.

Surface wrapping technology is used to easily create fluid domain geometry from complicated models, such as for air volume of models that comprise many solid parts and complicated geometry, like the interior of an electronic device. The surface wrapping algorithm quickly creates an airtight envelope based on the selected bodies (geometry or meshes with no underlying geometry) that define its surface boundaries and embedded bodies. This envelope can be generated as either polygon geometry or directly as a 2D mesh.

Advanced surface wrapping controls allow you to control the resolution of the resulting fluid domain geometry to better capture details within the domain when needed.

These controls include the ability to have tunable resolution of local features by either explicitly specifying a local resolution, allowing for automatic wrap refinement or assigning contact prevention. Contact prevention ensures that small gaps between two unconnected surfaces are maintained.

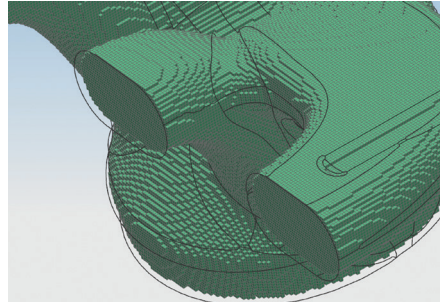
Ultimately, with Simcenter 3D Advanced Fluid Modeling, you can create fluid geometry in a fraction of the time than can be achieved with traditional CFD preprocessors.

Ensuring accurate fluid meshing

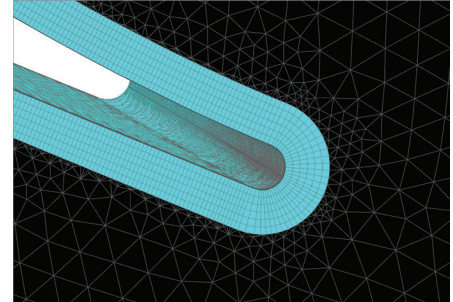
It is critical to accurately model the boundary layer for calculating friction and heat transfer for a number of applications, including heating, ventilation and air conditioning (HVAC), flow within a powertrain component and

external flows. Simcenter 3D Advanced Fluid Modeling delivers a complete set of tools to define the boundary layer mesh. With this product, you can control the mesh refinement level and first node location, as well as total thickness of the boundary layer mesh, which ensures optimal wall-adjacent meshes for turbulence modeling.

The boundary layer mesh can be further combined with a hybrid hexahedral-tetrahedral (hex-tet) mesh where hex elements are inserted wherever possible and connected to tets through pyramid elements. The use of hybrid hex-tet meshing, with a significant number of



Hex mesh throughout central volume of geometry.



Boundary layer mesh of an Airfoil.

structured elements, holds potential for improved accuracy and reduction in total mesh size to improve computational performance.

Speeding up analysis iterations

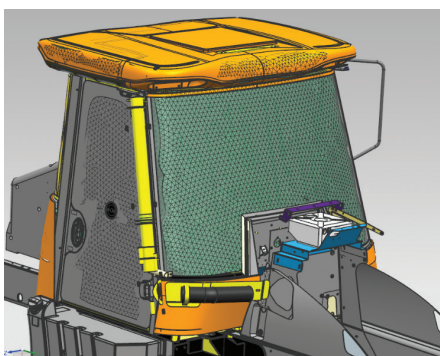
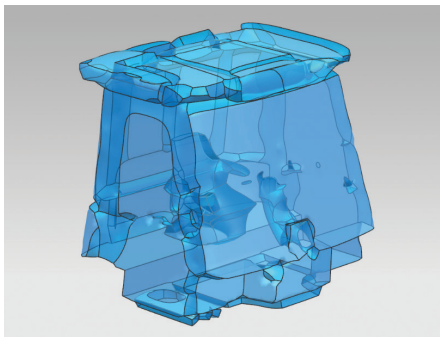
As is typical with all Simcenter 3D products, the fluid domain analysis model is associated to the base design geometry. If changes are made, the original part geometry, the fluid domain geometry, mesh and boundary conditions can all be automatically updated to reflect the new dimensions in a matter of minutes.

If the base design comprises of a complex assemblies the fluid domain model is also able to wrap assembly FEM which are fully associative to the underlying assembly, with one-click update to reflect changes in its structure.

This means engineers spend less time modeling and more time evaluating how design changes will impact flow or acoustics performance.

Providing multi-CAE support

While Simcenter 3D Advanced Fluid Modeling can be used in conjunction with Simcenter 3D CFD solutions, Simcenter 3D Flow, Simcenter 3D Advanced Flow and Simcenter 3D Electronic Systems Cooling, analysts can also leverage Simcenter 3D Advanced Fluid Modeling's benefits by using it as a preprocessor for other well-known, third-party CFD solvers. Simcenter 3D Advanced Fluid Modeling can be used to export analysis models in the CFD General Notation System (CGNS) file format that is supported by a variety of other CFD tools.



844-GEO-SUPT
support@geoplms.com
geoplms.com

Siemens PLM Software
www.siemens.com/plm

Americas +1 314 264 8499
Europe +44 (0) 1276 413200
Asia-Pacific +852 2230 3308

© 2017 Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. Simcenter is a trademark or registered trademark of Siemens Industry Software NV or any of its affiliates. All other trademarks, registered trademarks or service marks belong to their respective holders.
37205-A14 2/17 W