SIEMENS
Ingenuity for life

Femap Flow

Computational fluid dynamics (CFD) to accurately and efficiently simulate fluid flow

Benefits

- Allows investigation of multiple "what-if" scenarios involving complex assemblies
- Provides extensive set of tools for creating CFD analysis-ready geometry
- Allows troubleshooting of operational thermo-fluid problems prior to expensive physical prototyping time
- Enables advanced CFD simulation within Femap such as high-speed flows, rotational flows, non-Newtonian fluids, static motion of surfaces, particle tracking
- By default, all 2D and 3D solids transfer heat to the fluid they adjoin and serve as obstructions to the fluid flow; users can control the surface roughness and walls' convective properties globally and locally

Summary

Femap™ Flow software, part of the Simcenter™ solutions, is a computational fluid dynamics (CFD) solution that is fully embedded into the native Femap environment. It provides fast, efficient and accurate functionality to simulate fluid flow and heat transfer for complex parts and assemblies. You also can explicitly couple Femap Flow with Femap Thermal and Femap Advanced Thermal for comprehensive thermo-fluid modeling of a wide range of multi-physics problems. Femap Flow simulation applications are typically used in the following industries: aerospace and defense, automotive, consumer products, high-tech electronics, medical devices and instruments, power generation and process.

Basic functionality

Femap Flow models 3D fluid velocity, temperature, pressure and other industry-specific results by solving the Navier-Stokes equations for both steady-state and transient applications. It uses an efficient and robust element-based finite volume, multigrid solver enabling simulation of both low-speed and high-speed compressible flows.

Femap Flow technology enables users to model complex fluid flow problems. Solver and modeling features include:

- Steady-state and transient analysis (adaptive correction multigrid solver)
- Unstructured fluid meshes (supports tetrahedral, brick and wedge element types)
- Turbulent, laminar and mixed flows
- CFD solution intermediate results recovery and restart
- Heat loads and temperature restraints on the fluid
- Forced, natural and mixed convection
- Non-Newtonian fluids
- Fluid buoyancy
- Multiple enclosures
- Immiscible fluids and homogeneous gas mixtures
- Internal and external flows
- Complete and seamless coupling to Femap Thermal for simulation of conjugate heat transfer (handles disjoint meshes at fluid/solid boundaries)
- Losses in fluid flow due to screens, filters and other fluid obstructions (including orthotropic porous blockages)
- Head loss inlets and openings (fixed or proportional to calculated velocity or squared velocity)

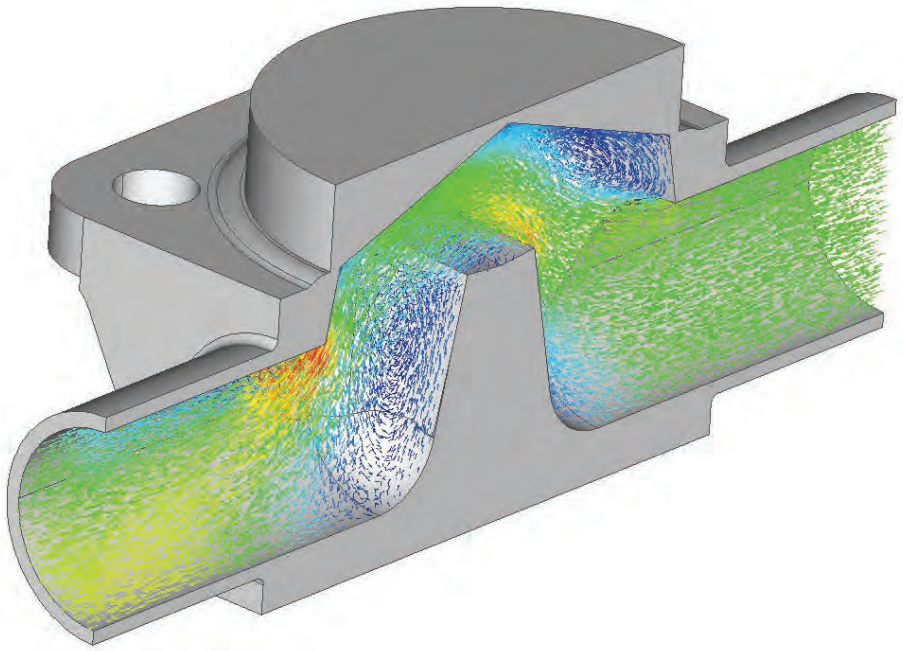
Femap Flow

- Fluid swirl and turbulence at inlet and internal fans
- Fluid recirculation loop with head loss or heat input/loss or fluid temperature change between unconnected fluid regions
- Automatic connection between disjoint fluid meshes for assembly modeling
- Altitude effects
- Nonlinear flow boundary conditions

Reliable and robust CFD solver technology

Femap Flow combines the versatility of finite element-based analysis technology with the power and accuracy of a control volume formulation.

- Algebraic multigrid solver technology
- Solver solution time linear with model size
- Co-located calculation points for momentum, mass and energy
- Momentum and mass equations solved simultaneously, not separately
- Expanded support for turbulence models: fixed viscosity, mixing length, k-epsilon, k-omega, SST, LES
- Near wall effects and convection handled for flow simulation with and without wall functions
- First- and second-order advection schemes
- Solver monitor with dynamic plotting of solution convergence and attributes
- Intermediate results display and recovery directly from solver progress monitor



Simulation results

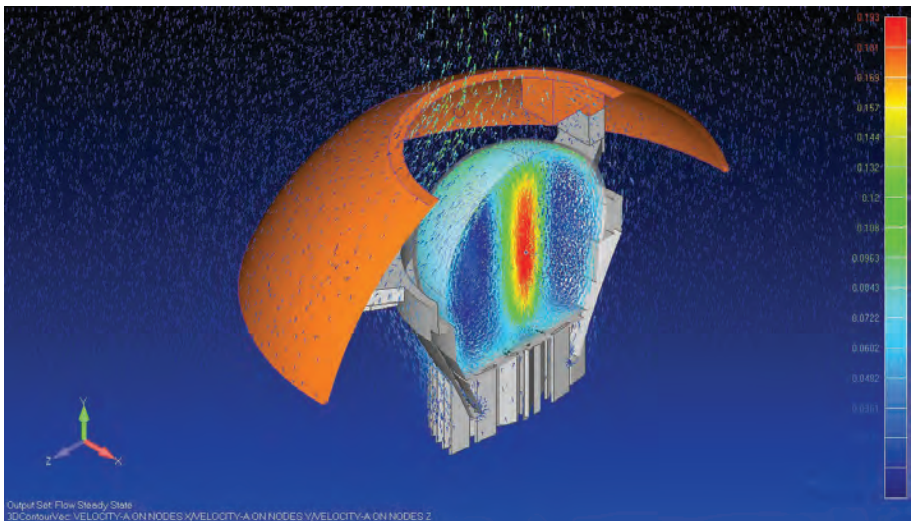
Simulation results can be displayed with graphical plots, charts and reports. The Femap postprocessing toolset makes it easy to generate images and reports to communicate the desired results to a design team. The following simulation results are available for postprocessing.

The set of flow results has been updated. The following results are now available:

- Temperatures
- Velocities
- Velocities adjusted
- Vorticity
- Static and total pressures
- Local and bulk convection coefficients
- Turbulence model quantities
- Non-Newtonian model quantities
- Fluid densities
- Shear stresses
- Roughness
- Y+
- Surface pressures
- Mass Fluxes
- Convective Fluxes
- Humidity, tracer fluids and mixtures
- PMV (predicted mean vote)
- PPD (predicted percent dissatisfied)
- Mach numbers
- Acoustic power density
- Pressure and shear resultants
- Condensation/evaporation

Features

- Support for disjoint meshes within an assembly
- Ability to leverage the Femap geometry modeling, abstraction and meshing toolset
- Ability to automatically transfer heat from all solid surfaces obstructing the fluid
- Handling of disjoint meshes at the fluid/solid boundaries for conjugate heat transfer
- Low-speed incompressible and high-speed compressible flows modeling
- Motion modeling for rotating and translating surfaces
- Multiple rotating frames of reference
- Model symmetry and slip conditions
- Support of unstructured mesh combination including tetrahedral, wedge and brick elements
- Volume porosity and flow resistances
- Disjoint meshing, support for flow across disconnected mesh-to-mesh volumes (automatic run-time fluid couplings created)
- High-order advection schemes
- Humidity and general scalars modeling for contaminants tracking



Feature-related advantages

Disjoint fluid meshes

The Femap Flow solver supports dissimilar fluid meshes at the interfaces between the different parts within a complex assembly. This allows the user to quickly investigate many "what-if" simulation scenarios involving complex assemblies. All parts within any design assembly context can be meshed independently.

The resulting disjoint fluid faces at the surface junctions between the different parts within the assembly can be connected to form a single fluid domain at solve time. Individual part changes can be re-integrated quickly within the assembly mesh, thereby avoiding the time-consuming task of remeshing the entire assembly.

The Femap CFD solution toolset of choice

Femap Flow is native within the Femap simulation product. The Femap application allows the skilled engineer and CFD specialist alike to avoid any additional transfer of input files or geometry conversions and manipulations. Integrity is assured by maintaining data associativity between model building, solving and results interpretation within a common working environment. Femap Flow provides the ability to model, catalog and share parts and material libraries among the entire design team, thereby minimizing tedious rework and modeling errors.

Thermo-fluid interactions

The fluid flow modeling capabilities offered by Femap Flow can be explicitly combined with the Femap Thermal and Femap Advanced Thermal heat transfer solutions to simulate fully-coupled thermo-fluid interaction problems. Whenever the Femap Flow and Femap Thermal products are combined, the thermo-fluid iterative solver automatically and seamlessly turns on within Femap at no additional cost offering both conduction and radiation modeling to be fully coupled to fluid flows through convective heat transfer at the fluid/solid interface. The coupled solver handles disjoint meshes at the fluid/solid boundaries. This unique technology of thermally coupling the disjoint solid and fluid meshes provides a complete solution for simulation of conjugate heat transfer problems.

Features *continued*

- Fractional step scheme
- Enhanced second-order discretization schemes
- Enhanced freeze-flow and restart options
- Static pressure and convective outflow boundary conditions
- Expanded output options for simulation results
- Enhanced CGNS export
- Execute up to 8 processes on a compute node via distributed memory parallelization (DMP)

Geometry modeling and complex geometry abstraction toolset

The Femap interface provides a complete set of tools for creating CFD analysis-ready geometry. A user can refine the mesh in critical areas and selectively control mesh density, minimizing or optimizing model size for rapid and accurate solution. The same Femap tools are leveraged by the engineer, removing the requirement for additional training of an additional pre/post interface. The engineer and expert can concentrate on the CFD design from the start.

Automatic convection from all surfaces to the fluid

In addition to handling disjoint meshes at the fluid/solid boundaries, an automatic solid surface convection option can be used to avoid unnecessary creation of flow solid surface entities (and convecting flow blockages) that have identical surface properties. When this option is selected, all the solid surfaces obstructing the flow (so called flow surfaces) that are meshed having thermal solid material properties (2D or 3D) are treated as flow surfaces by the Femap Flow solver. As a result, all the model solid surfaces automatically transfer heat to the fluid elements they adjoin. Similarly, all volumes that are meshed with nonfluid 3D meshes and not already defined as flow blockages automatically transfer heat from their surfaces as well.

Product availability

Femap Flow is a module in the suite of advanced simulation applications available within the Femap product configuration. When used in combination with Femap Thermal and/or Femap Advanced Thermal, Femap Flow provides a coupled multi-physics solution for complex fluid flow/thermal applications.

Siemens Digital Industries Software
www.sw.siemens.com

Americas +1 314 264 8499
Europe +44 (0) 1276 413200
Asia-Pacific +852 2230 3333

© 2018 Siemens. A list of relevant Siemens trademarks can be found here. Other trademarks belong to their respective owners.

10848-C7 1/18 C