Femap Flow

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

fact sheet

Siemens PLM Software

www.siemens.com/plm/femap

Summary

Femap® Flow software 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. Femap Flow can also be explicitly coupled 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, power generation and process.

Benefits

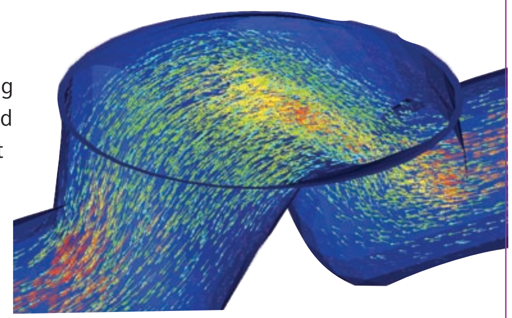
Allows for investigation of multiple "what-if" scenarios involving complex assemblies

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, etc.

By default, all 2D and 3D solids will 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



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.

The Femap Flow technology allows a user to model complex fluid flow problems. The 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 (k-ε mixing length), 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
- Low-speed incompressible and high-speed compressible flows modeling
- Motion modeling for rotating and translating surfaces
- Humidity modeling
- General scalars modeling for contaminants tracking, etc.
- Multiple rotating frames of reference
- Model symmetry and slip conditions
- Fluid buoyancy
- Multiple enclosures
- Multiple fluids

Features

Automatic connection between disjoint fluid meshes within an assembly

Leverages the Femap geometry modeling, abstraction and meshing toolset

All solid surfaces obstructing the fluid can automatically transfer heat to the fluid they adjoin

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

- Internal or 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)
- 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 is linear with model size
- Calculation points for momentum, mass and energy are co-located
- Momentum and mass equations are solved simultaneously, not separately
- Turbulence models include: mixing length, k-ε and fixed turbulent viscosity
- Near wall effects and convection are handled by enhanced three log-law wall functions
- First- or second-order advection schemes are available
- Solver monitor with dynamic plotting of solution convergence and attributes
- Intermediate results display and recovery directly from solver progress monitor

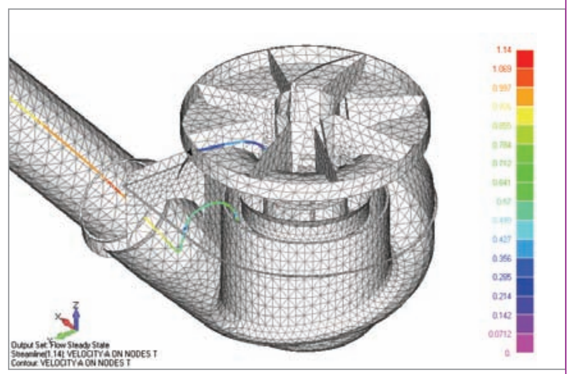
Simulation results

Simulation results can be displayed with graphical plots, charts and reports. The Femap post-processing 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 post-processing:

- Fluid velocity and streamlines
- Fluid and solid temperatures
- Mass flux at the different fluid boundaries
- Heat flux
- Fluid pressure
- Heat transfer coefficients
- K- ε turbulence data
- Fluid density
- Surface shear stresses
- HVAC-specific (PPD, PMV, etc.)
- Mach number

Features

Automatic connection between disjoint fluid meshes. The Femap Flow solver can automatically join 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 automatically 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 re-meshing 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 interactions problems. Whenever the Femap Flow and Femap Thermal products are combined, the thermo-fluid iterative solver is automatically and seamlessly turned 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 thermally coupling the disjoint solid and fluid meshes provides a complete solution for simulation of conjugate heat transfer problems.

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. Thus, all the model solid surfaces will 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 will 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.

For more information, contact your local Velocity Series™ portfolio representative:

Contact

Siemens PLM Software – www.siemens.com/plm/femap

Americas 800 807 2200
Europe 44 (0) 1202 243455
Asia-Pacific 852 2230 3308

SIEMENS