CFD in COMSOL Multiphysics

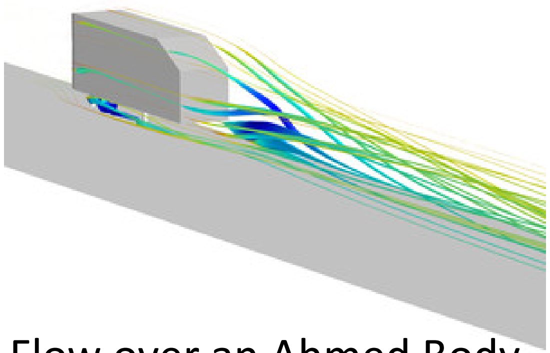
Mats Nigam

Worldwide Sales Offices

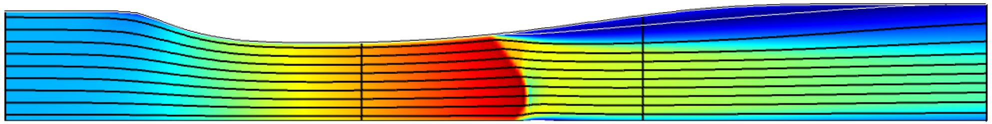
COMSOL

CFD – The Classical View

- Laminar
- Turbulent
  - RANS
  - LES
  - ....
- Incompressible
- Compressible
  - Mach number effects



Flow over an Ahmed Body

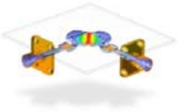


Flow in a Sajben diffuser

COMSOL

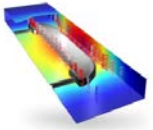
Traditional Approach to Modeling

Electromagnetic Fields



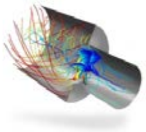
Acoustics

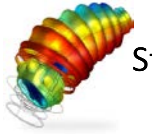
Chemical Reactions



Heat Transfer

Fluid Flow



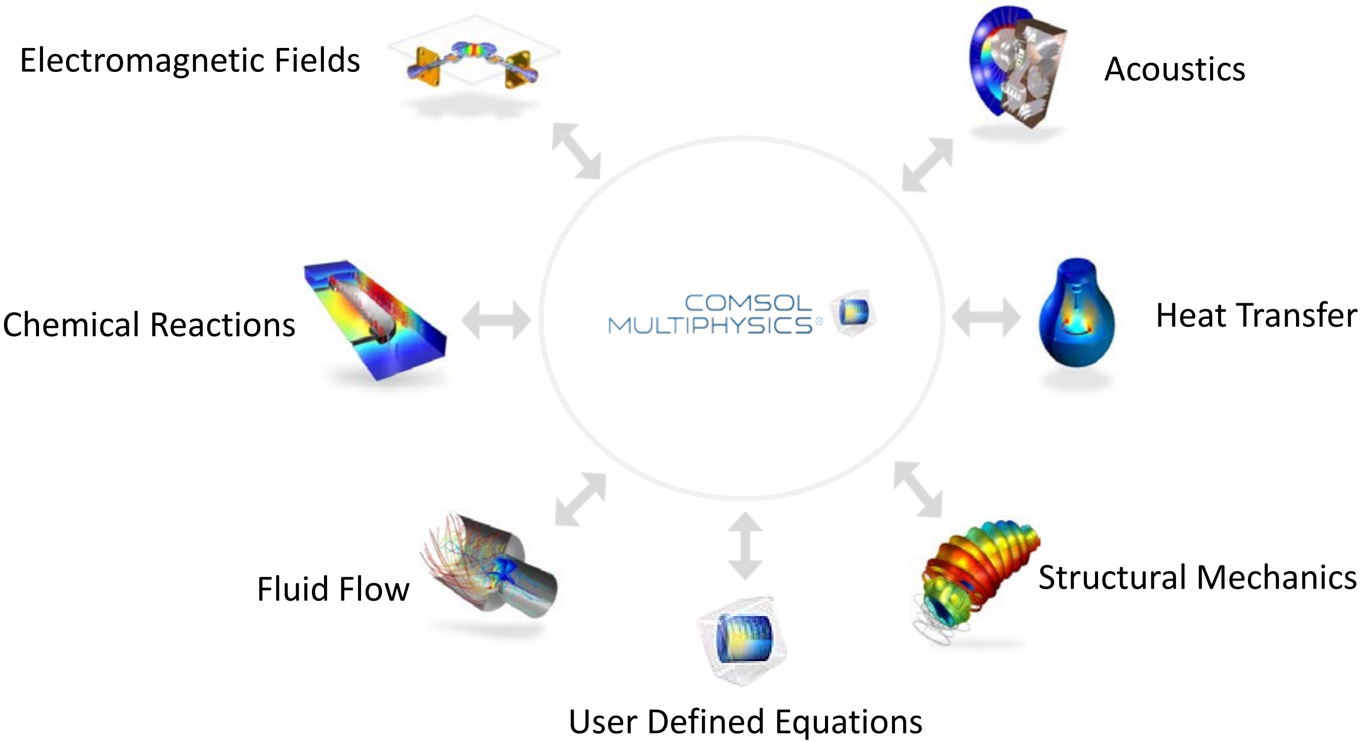


Structural Mechanics

User Defined Equations

COMSOL

COMSOL Approach to Modeling



COMSOL

Typical Multiphysics Couplings

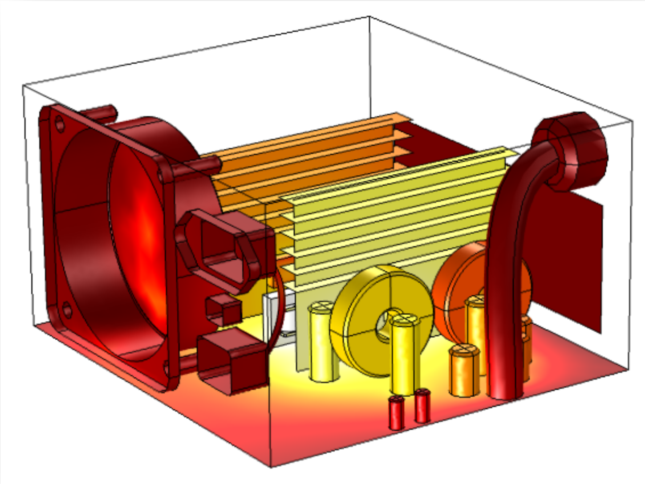
- Flow with heat transfer: Non-isothermal flow/Conjugate heat transfer
- Flow with mass transfer: Reacting flow
- Flow and structures: Fluid-Structure Interaction (FSI)
- Flow with particles: Particle tracing

Conjugate Heat Transfer - Example

- The model examines the air cooling of a power supply unit (PSU) with multiple electronics components acting as heat sources.

- Avoid damaging components by excessively high temperatures

- Extracting fan and a perforated grille cause an air flow in the enclosure. Fins are used to improve cooling efficiency.



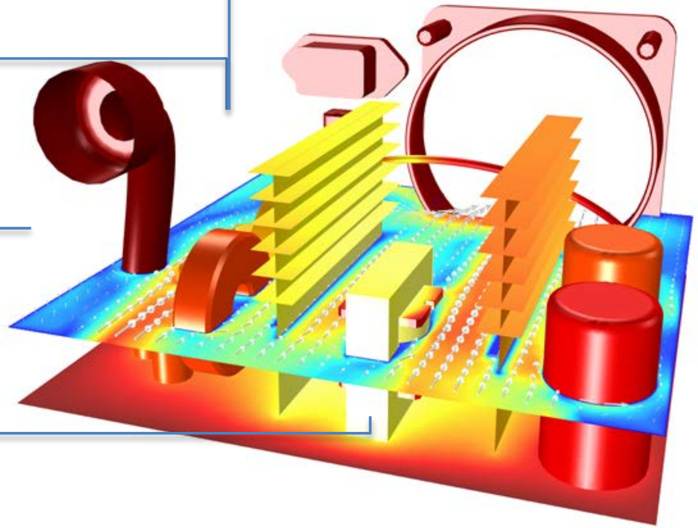
Cooling of a Power Supply Unit (PSU)

Conjugate Heat Transfer - Example

Fluid flow described by Navier-Stokes in air in the compartment

Heat transfer by conduction and advection in air

Continuity in heat flux and temperature at solid-air interfaces



Heat transfer by conduction in the solid parts

COMSOL

Reacting Flow - Example

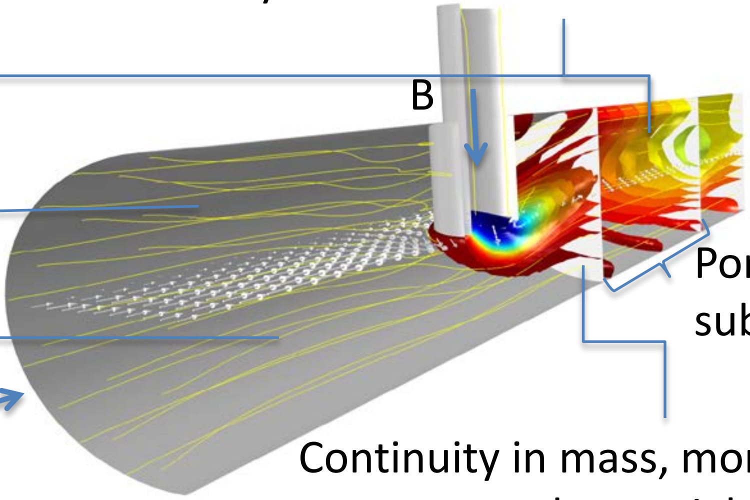
Chemical species transport and reactions in porous media

Flow in porous media described by Brinkman's extension of Darcy

B

C

Chemical species transport



Porous subdomain

Fluid flow described by Navier-Stokes

A

Continuity in mass, momentum, pressure, and material across porous media interface

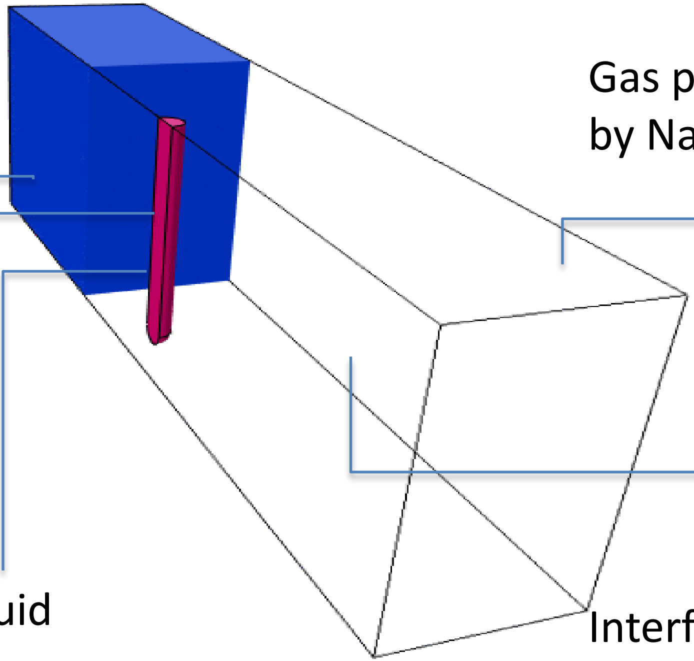
COMSOL

Fluid-Structure Interaction - Example

Liquid phase described by Navier-Stokes

Gas phase described by Navier-Stokes

Solid mechanics in the obstacle with moving mesh



Interface between solid and fluid described by moving mesh using the ALE method

Interface between the two fluids modeled with phase fields

Fluid-Structure Interaction - ALE

$$\rho\left(\frac{\partial \mathbf{u}}{\partial T}+\left(\mathbf{u}-\frac{\partial \mathbf{x}}{\partial T}\right)\cdot \nabla_x \mathbf{u}\right)=\nabla_x\cdot\boldsymbol{\sigma}$$

$$\rho\frac{\partial^2 \mathbf{u}_{\text{solid}}}{\partial t^2}=\nabla_X\cdot\boldsymbol{\sigma}_{\text{solid}}$$

- At fluid-solid interface:

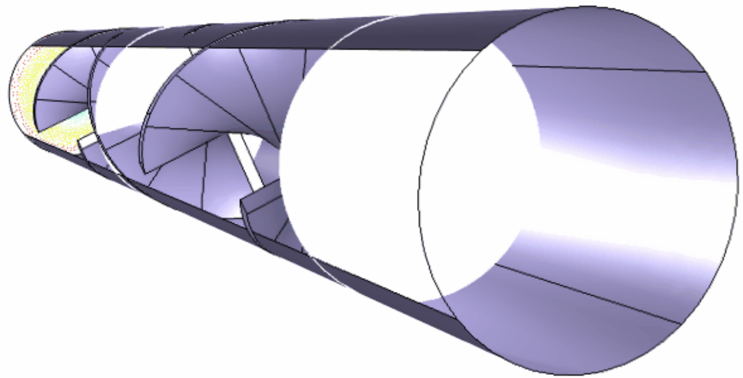
$$\mathbf{u}=\frac{\partial \mathbf{u}_{\text{solid}}}{\partial t},\qquad \boldsymbol{\sigma}_{\text{solid}}\mathbf{n}=\boldsymbol{A}\boldsymbol{\sigma}\mathbf{n},\qquad \boldsymbol{A}\colon \mathbf{x}\rightarrow\mathbf{X},\qquad \mathbf{X}+\mathbf{u}_{\text{solid}}=\mathbf{x}_{\text{fs}-\text{interface}}$$

- Use smoothing for interior mesh points

COMSOL

Particle Tracing - Example

- Animation using comet tail plots and Poincare maps



COMSOL

The Finite Element Method

- General PDE: $L(u) - f = 0$
- Assume that $u \approx \tilde{u} = \sum_i u_i \phi_i$ (1)
  Where ϕ_i is a set of trial functions.

(1) is a Fourier expansion

ϕ_i is a polynomial in each mesh cell

ϕ_i is a constant value for each cell/node

Spectral methods

Finite elements

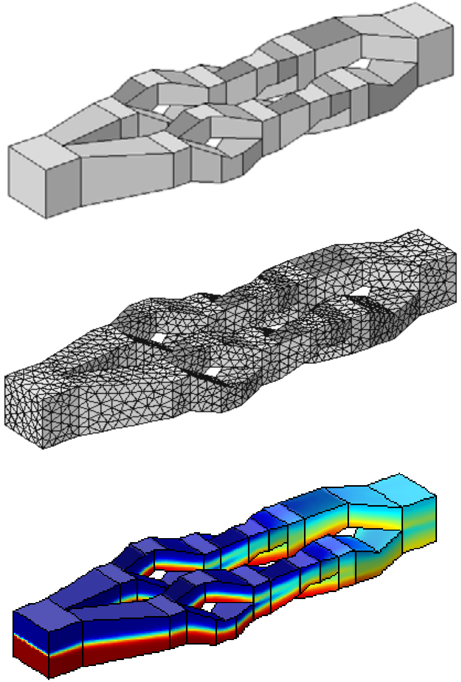
Finite volumes

ϕ_i is piecewise constant

COMSOL

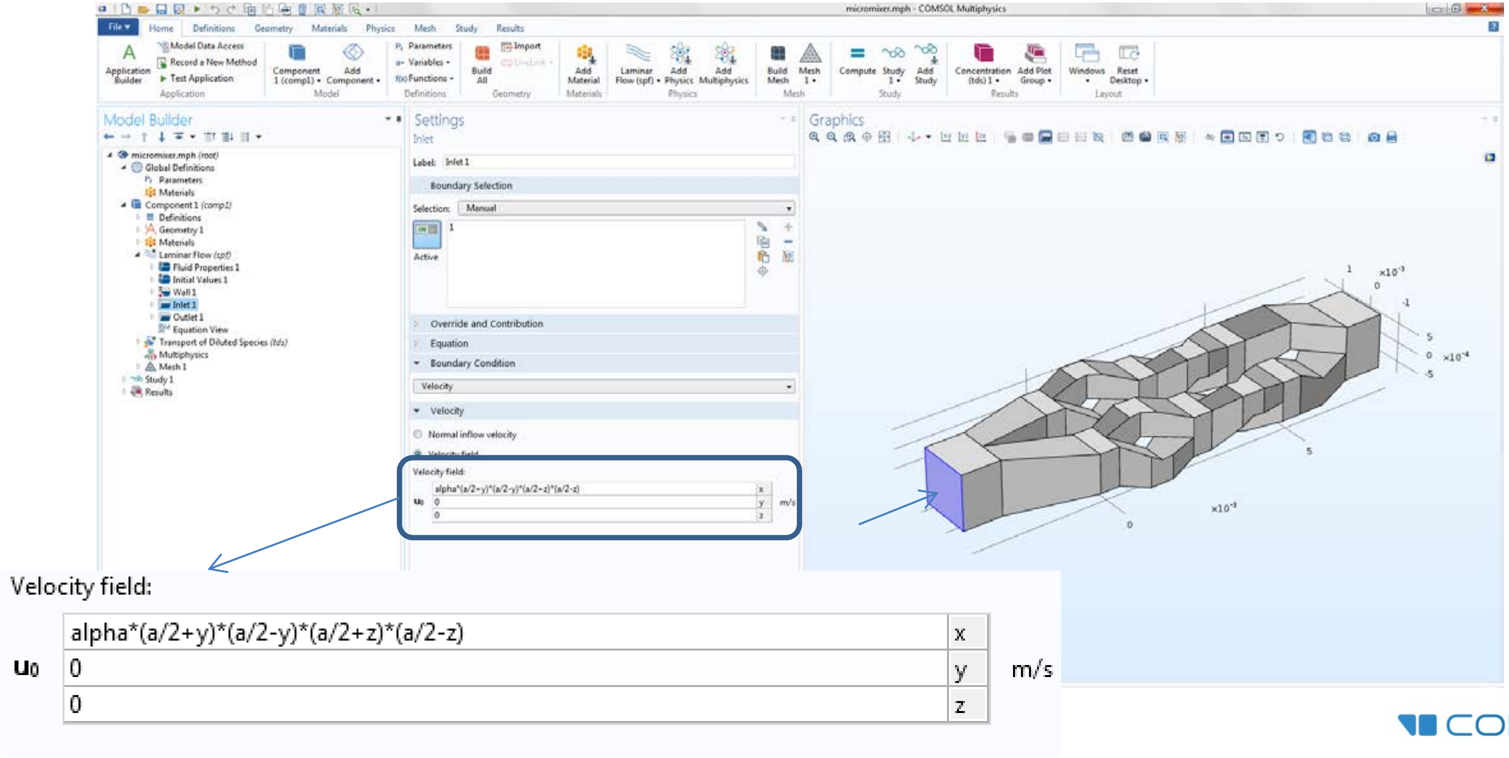
COMSOL Multiphysics Workflow

Micromixer



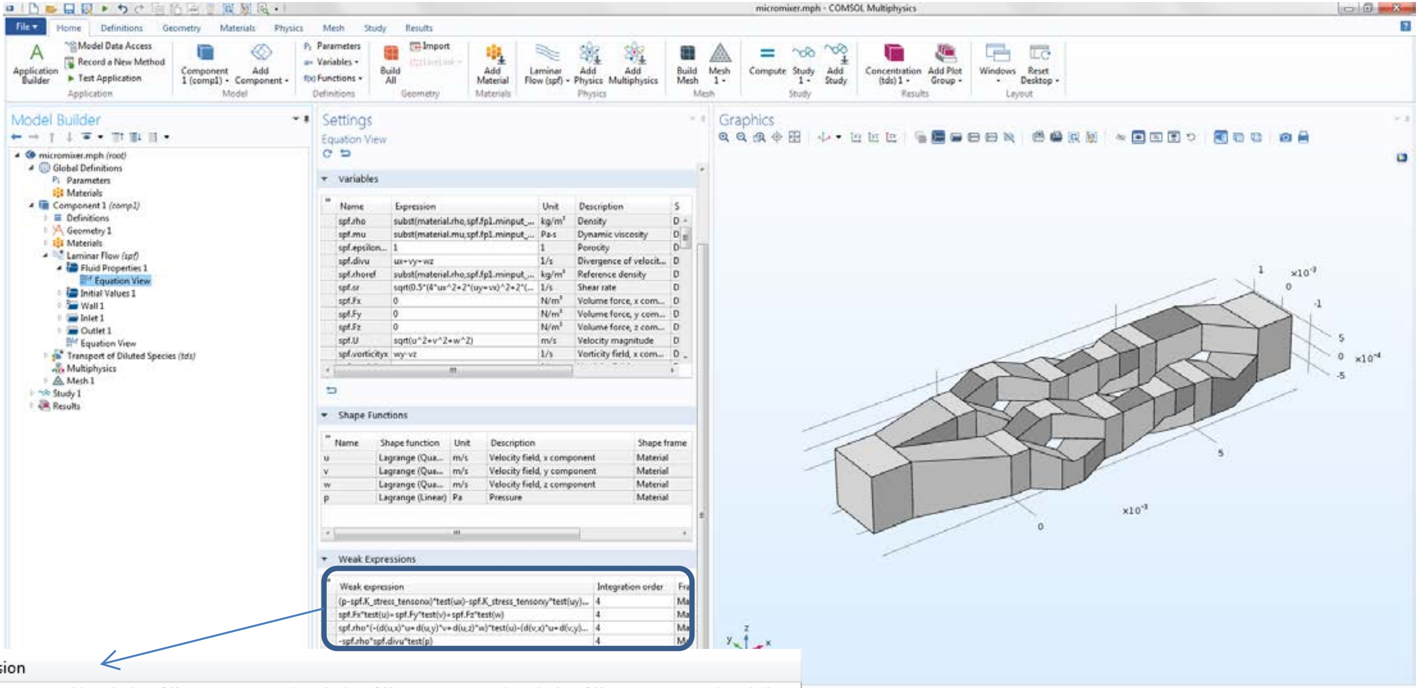
- Model
- Definitions
- Geometry
- Materials
- Physics
- Mesh
- Study
- Results

User Friendly Interface



COMSOL

Everything is Equation Based



Weak expression
(p-spf.K_stress_tensorxx)*test(ux)-spf.K_stress_tensorxy*test(uy)-spf.K_stress_tensorxz*test(uz)-spf.K_stress_tensoryx*test(vx)+...
spf.Fx*test(u)+spf.Fy*test(v)+spf.Fz*test(w)
spf.rho*(-(d(u,x)*u+d(u,y)*v+d(u,z)*w)*test(u)-(d(v,x)*u+d(v,y)*v+d(v,z)*w)*test(v)-(d(w,x)*u+d(w,y)*v+d(w,z)*w)*test(w))
-spf.rho*spf.divu*test(p)

Add Your Own Equations to COMSOL's

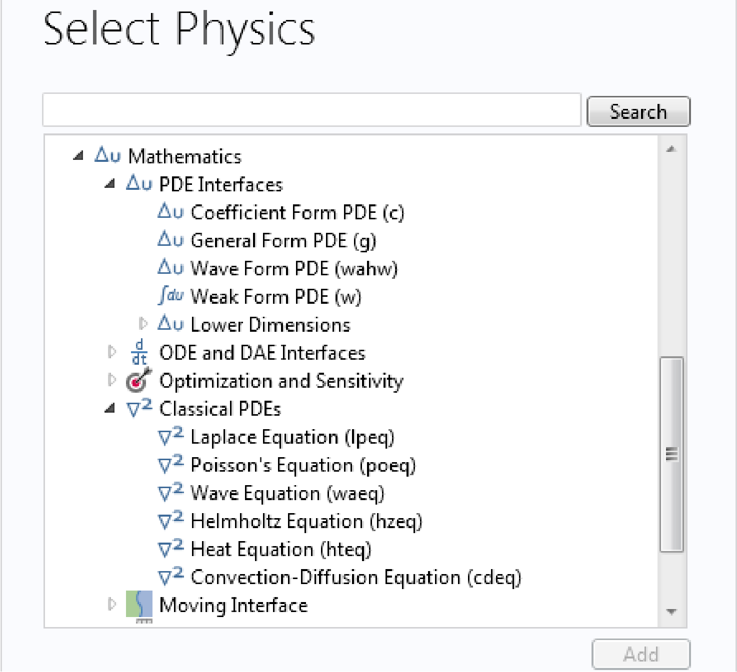
Don't see what you need?

Add your own equation

- ODE's
- PDE's
- Classical PDE's

Just type them in

- No Recompiling
- No Programming



Product Suite – COMSOL® 5.2

COMSOL Multiphysics®

COMSOL Server™

ELECTRICAL
- AC/DC Module
- RF Module
- Wave Optics Module
- Ray Optics Module
- MEMS Module
- Plasma Module
- Semiconductor Module

MECHANICAL
- Heat Transfer Module
- Structural Mechanics Module
- Nonlinear Structural Materials Module
- Geomechanics Module
- Fatigue Module
- Multibody Dynamics Module
- Acoustics Module

FLUID
- CFD Module
- Mixer Module
- Microfluidics Module
- Subsurface Flow Module
- Pipe Flow Module
- Molecular Flow Module

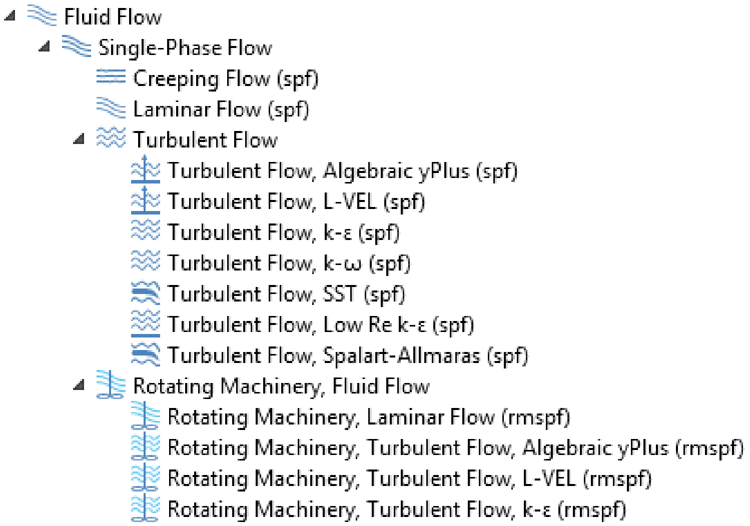
CHEMICAL
- Chemical Reaction Engineering Module
- Batteries & Fuel Cells Module
- Electrodeposition Module
- Corrosion Module
- Electrochemistry Module

MULTIPURPOSE
- Optimization Module
- Material Library
- Particle Tracing Module

INTERFACING
- LiveLink™ *for* MATLAB®
- LiveLink™ *for* Excel®
- CAD Import Module
- Design Module
- ECAD Import Module
- LiveLink™ *for* SOLIDWORKS®
- LiveLink™ *for* Inventor®
- LiveLink™ *for* AutoCAD®
- LiveLink™ *for* Revit®
- LiveLink™ *for* PTC® Creo® Parametric™
- LiveLink™ *for* PTC® Pro/ENGINEER®
- LiveLink™ *for* Solid Edge®
- File Import *for* CATIA® V5

Single-Phase Flow

- Creeping flow/Stokes flow
- Laminar flow
  - Newtonian and
  - Non-Newtonian flow
- Turbulent flow
  - Algebraic yPlus model
  - L-VEL model
  - k-ε model
  - k-ω model
  - SST model
  - Low Re k-ε model
  - Spalart-Allmaras model
- Rotating machinery
  - Laminar and turbulent flow



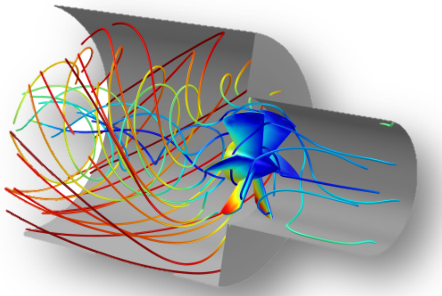
The Single-Phase Flow user interfaces as displayed in the Physics list in the CFD Module.

COMSOL

Single-Phase Flow

General functionality for both laminar and turbulent flow

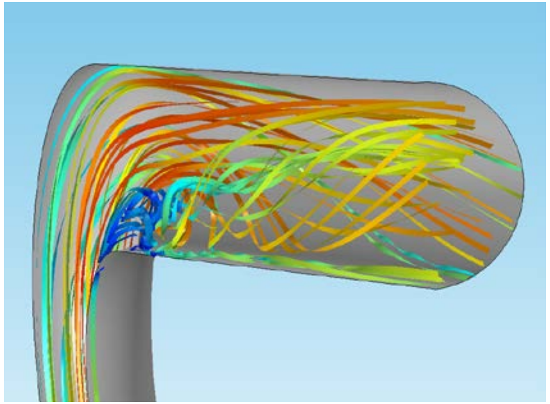
- Swirl flow
  - Includes the out-of-plane velocity component for axisymmetric flows
- Specific boundary conditions
  - Fully developed laminar inflow and outflow for simulating long inlet and outlet channels
  - Assembly boundaries for geometries consisting of several parts
  - Wall conditions on internal shells for simulating thin immersed structures
  - Screen conditions for simulating thin perforated plates and wire gauzes



Streamlines in an HVAC duct

Turbulent Flow with Wall Functions

- Models with wall functions
  - k-ε model
    - The standard k-ε model with realizability constraints
    - The basic industrial modeling tool
  - k-ω model
    - The revised Wilcox k-ω model (1998) with realizability constraints
- Versatile and easy to use models
- Wall functions for smooth and rough walls



Flow in a pipe elbow simulated with the k-ω model.

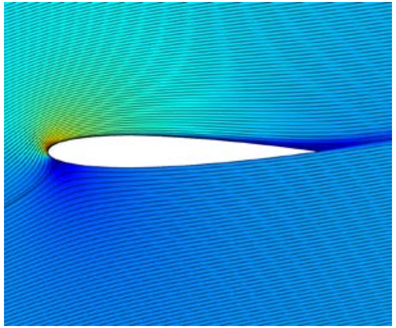
COMSOL

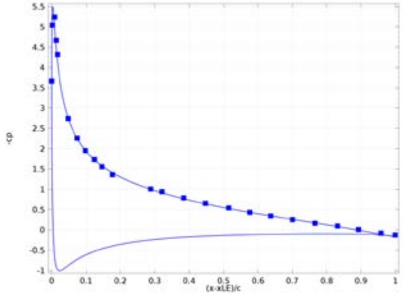
Wall Resolved Turbulent Flow

- Algebraic turbulence models
  - Algebraic yPlus model
  - L-VEL model

  Turbulent viscosity is defined from local flow speed and wall distance – no additional boundary conditions

- Transport-equation models
  - SST model with realizability constraints
  - Low Re k-ε turbulence model with realizability constraints
  - Spalart-Allmaras model with rotational correction

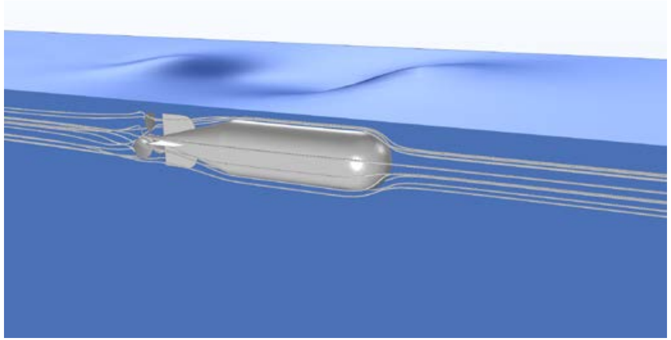




COMSOL

Rotating Machinery

- Laminar and turbulent
- Sliding mesh
  - Accurate time-dependent simulations
- Frozen rotor
  - Fast, stationary approximations
  - Can provide starting conditions for a sliding mesh simulation
  - Stationary free surface post-processing feature
- Interior wall conditions
  - Simulate infinitely thin blades and baffles



Flow around a torpedo

COMSOL

Rotating Machinery - ALE

- Sliding mesh:

$$\rho\left(\frac{\partial \mathbf{u}}{\partial T}+\left(\mathbf{u}-\frac{\partial \mathbf{x}}{\partial T}\right)\cdot\nabla\mathbf{u}\right)=\nabla\cdot\boldsymbol{\sigma}$$

$$\frac{\partial}{\partial T}=\frac{\partial}{\partial t}+\boldsymbol{\Omega}\times, \qquad \mathbf{u}=\mathbf{v}+\boldsymbol{\Omega}\times\mathbf{x}, \qquad \frac{\partial \mathbf{x}}{\partial T}=\boldsymbol{\Omega}\times\mathbf{x}\Longrightarrow$$

$$\rho\left(\left(\frac{\partial}{\partial t}+\boldsymbol{\Omega}\times\right)(\mathbf{v}+\boldsymbol{\Omega}\times\mathbf{x})+(\mathbf{v}+\boldsymbol{\Omega}\times\mathbf{x}-\boldsymbol{\Omega}\times\mathbf{x})\cdot\boldsymbol{\nabla}(\mathbf{v}+\boldsymbol{\Omega}\times\mathbf{x})\right)=\nabla\cdot\boldsymbol{\sigma}\Longrightarrow$$

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t}+\mathbf{v}\cdot\nabla\mathbf{v}+2\boldsymbol{\Omega}\times\mathbf{v}+\boldsymbol{\Omega}\times(\boldsymbol{\Omega}\times\mathbf{x})+\frac{\partial \boldsymbol{\Omega}}{\partial t}\times\mathbf{x}\right)=\nabla\cdot\boldsymbol{\sigma}$$

COMSOL

Rotating Machinery - ALE

- Frozen rotor:

$$\rho\left(\frac{\partial \mathbf{u}}{\partial T}+\left(\mathbf{u}-\frac{\partial \mathbf{x}}{\partial T}\right)\cdot\nabla\mathbf{u}\right)=\nabla\cdot\boldsymbol{\sigma}$$

$$\frac{\partial \mathbf{u}}{\partial T}\stackrel{\text{def}}{=}\boldsymbol{\Omega}\times\mathbf{u},\qquad \mathbf{u}=\mathbf{v}+\boldsymbol{\Omega}\times\mathbf{x},\qquad \frac{\partial \mathbf{x}}{\partial T}=\boldsymbol{\Omega}\times\mathbf{x}\Longrightarrow$$

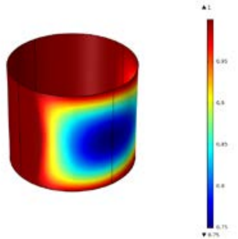
$$\rho\big(\boldsymbol{\Omega}\times(\mathbf{v}+\boldsymbol{\Omega}\times\mathbf{x})+(\mathbf{v}+\boldsymbol{\Omega}\times\mathbf{x}-\boldsymbol{\Omega}\times\mathbf{x})\cdot\boldsymbol{\nabla}(\mathbf{v}+\boldsymbol{\Omega}\times\mathbf{x})\big)=\nabla\cdot\boldsymbol{\sigma}\Longrightarrow$$

$$\rho\big(\mathbf{v}\cdot\nabla\mathbf{v}+2\boldsymbol{\Omega}\times\mathbf{v}+\boldsymbol{\Omega}\times(\boldsymbol{\Omega}\times\mathbf{x})\big)=\nabla\cdot\boldsymbol{\sigma}$$

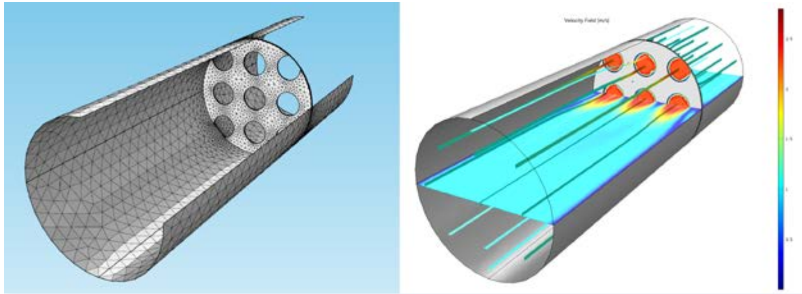
  - Set: $T\equiv 0$

Thin-Film and Porous Media Flow

- Thin-film flow
  - For lubrication and flow in narrow structures, which are modeled as 3D shells
  - Supports gaseous cavitation
- Porous media flow
  - Laminar or turbulent free-flow coupled to porous media flow including Forchheimer drag (high interstitial velocities)
  - Darcy's law and Brinkman equations with isotropic/anisotropic permeability tensor
  - Two-phase flow, Darcy's Law with capillary pressure models



Mass fraction for cavitating flow in a journal bearing modeled using the Thin-Film Flow, Shell interface.

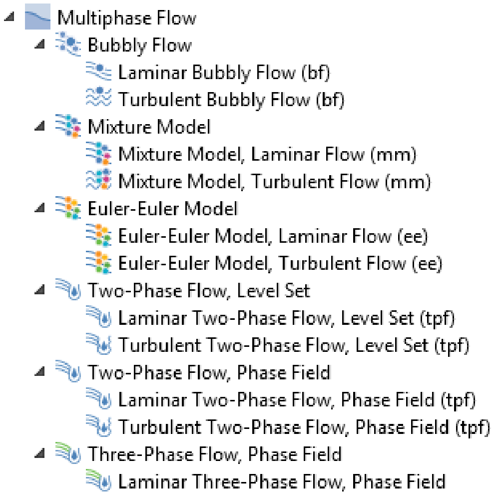


A porous filter which is supported by a perforated plate and immersed in turbulent pipe flow is modeled using the Free and Porous Media Flow interface.

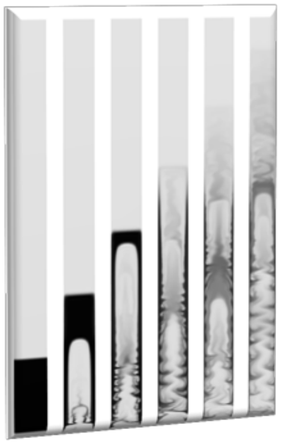
COMSOL

Multiphase Flow

- Disperse flows
  - Bubbly Flow
  - Mixture Model
  - Euler-Euler Model
- Separated flows
  - Two-Phase Flow, Level Set
  - Two-Phase Flow, Phase Field
  - Three-Phase Flow, Phase Field



The Multiphase Flow interfaces as displayed in the Physics list in the CFD Module

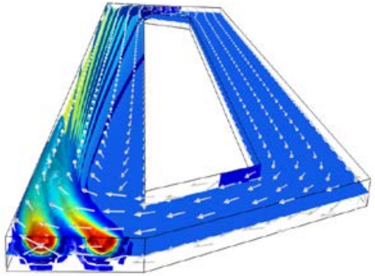


Startup of a fluidized bed modeled using the Euler-Euler Model interface

Multiphase Flow – Disperse Flows

The equations of motion are averaged over volumes which are small compared to the computational domain but large compared to the size of the dispersed particles/bubbles/droplets.

- Bubbly Flow & Mixture Model
  - Closures for the relative motion (slip) between the two phases assume that the particle relaxation time is small compared to the time scale of the mean flow.
  - For Bubbly flow, bubble concentration must be small (~ 0.1) unless coalescence is explicitly accounted for
  - Bubble induced turbulence in bubbly flow
  - Mass transfer between phases
  - Option to solve for interfacial area
  - Spherical and non-spherical particles
- Euler-Euler Flow
  - General two-phase flow
  - No restriction on particle relaxation time
  - Spherical and non-spherical particles
  - Mixture or phase-specific turbulence model

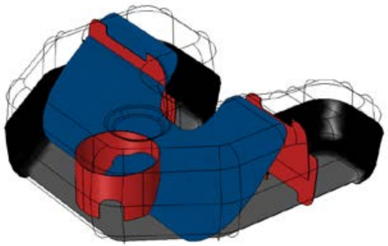


Bubble-induced turbulent flow in an airlift loop reactor

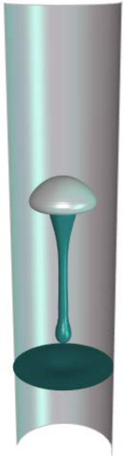
Multiphase Flow - Separated Flows

- Tracks the exact surface location using the Level-set or Phase-field models, or by using a Moving-mesh interface
- Accurate modeling of surface-tension effects
- Includes a surface-tension coefficient library
- Can be combined with the k-epsilon model for simulations of turbulent flow*

*Two-phase flow only



Sloshing in a fuel tank

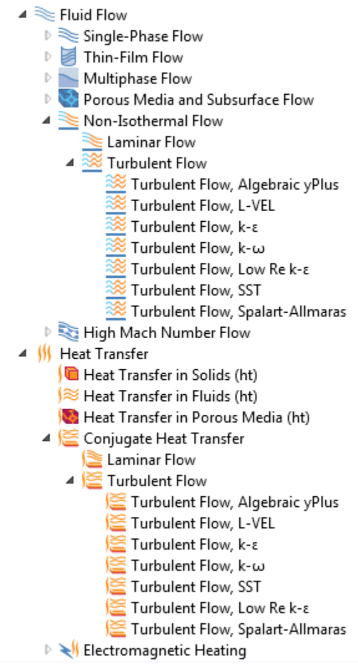


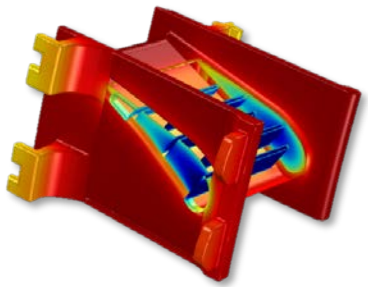
Gas bubble rising from a dense liquid up into a light liquid in a three-phase flow, phase field simulation

COMSOL

Non-Isothermal Flow and Conjugate Heat Transfer

- Heat transfer in fluids and solids
- Laminar and turbulent flow
- Compressible flow for $Ma < 0.3$
- Engineering correlations for convective heat transfer
- Porous media domains
- Thermal wall functions when using the k-epsilon or k-omega turbulence models
- Turbulent Prandtl number models

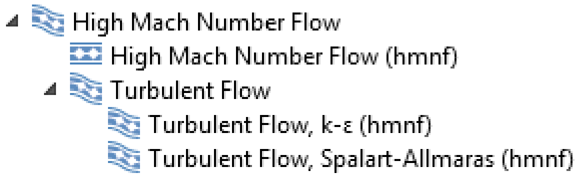




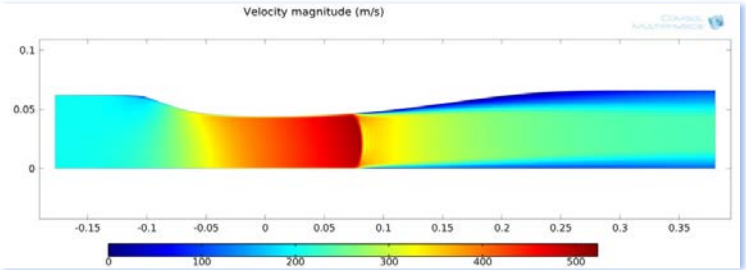
Flow and heat transfer in a turbine stator

High Mach Number Flow

- Laminar and turbulent flow
- k-ε turbulence model
- Spalart-Allmaras model
- Fully compressible flow for all Mach numbers
- Viscosity and conductivity can be determined from Sutherland’s law



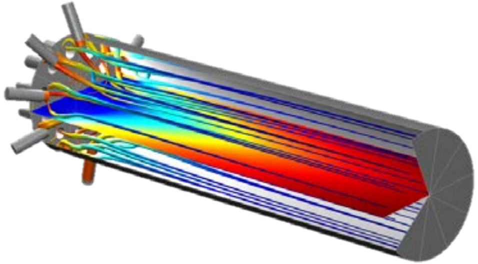
Turbulent compressible flow in a two-dimensional Sajben diffuser



COMSOL

Reacting Flow

- Multi-component transport and flow in diluted and concentrated solutions
  - Fickean and mixture-averaged formulations
  - Migration of charged species in electric fields
  - Mass transport in free and porous media flow
  - Turbulent mixing and reactions
  - Stefan velocities on boundaries with reactions
- Concentration-dependent density and viscosity in flow description



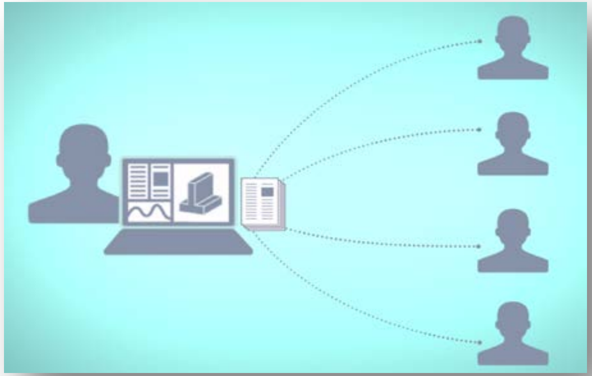
Turbulent reacting flow in a multi-jet reactor in a polymerization process.

- Chemical Species Transport
  - Transport of Diluted Species (tds)
  - Transport of Concentrated Species (tcs)
  - Reacting Flow
    - Laminar Flow (rspf)
    - Turbulent Flow
      - Turbulent Flow, k-ε (rspf)
      - Turbulent Flow, k-ω (rspf)
      - Turbulent Flow, SST (rspf)
      - Turbulent Flow, Low Re k-ε (rspf)
  - Reacting Flow in Porous Media
    - Transport of Diluted Species (rfds)
    - Transport of Concentrated Species (rfcs)

From Model to App

- Simulations today:
  - Mostly used by dedicated simulation engineers and scientists – just like you!
  - Require some degree of training to get started
- Simulations tomorrow:

R&D



Engineering

Manufacturing

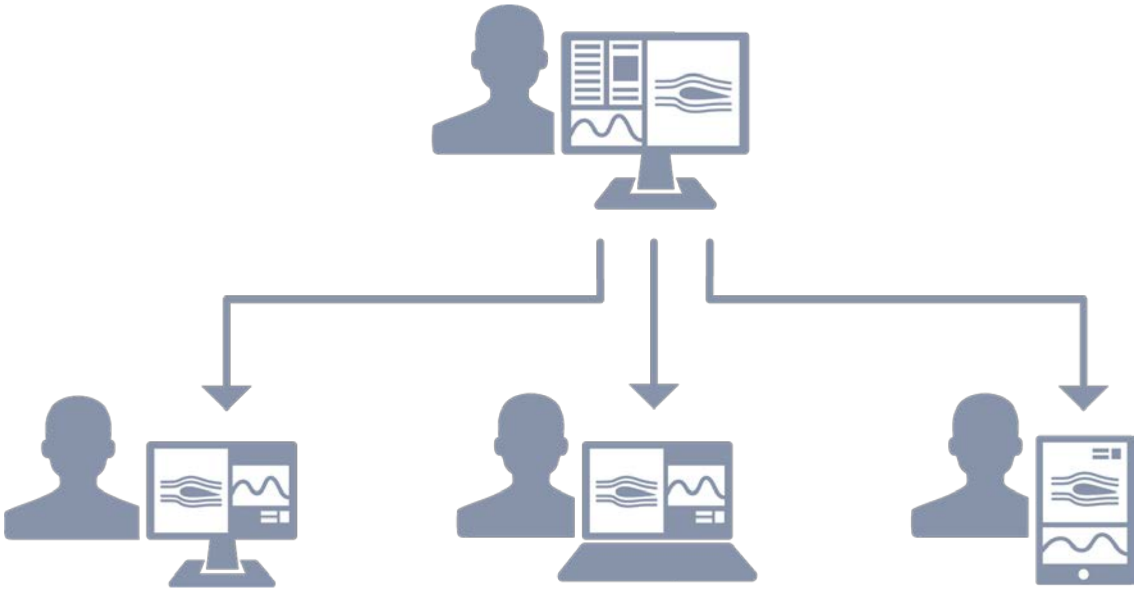
Installation

Sales

COMSOL Users Turn App Designers

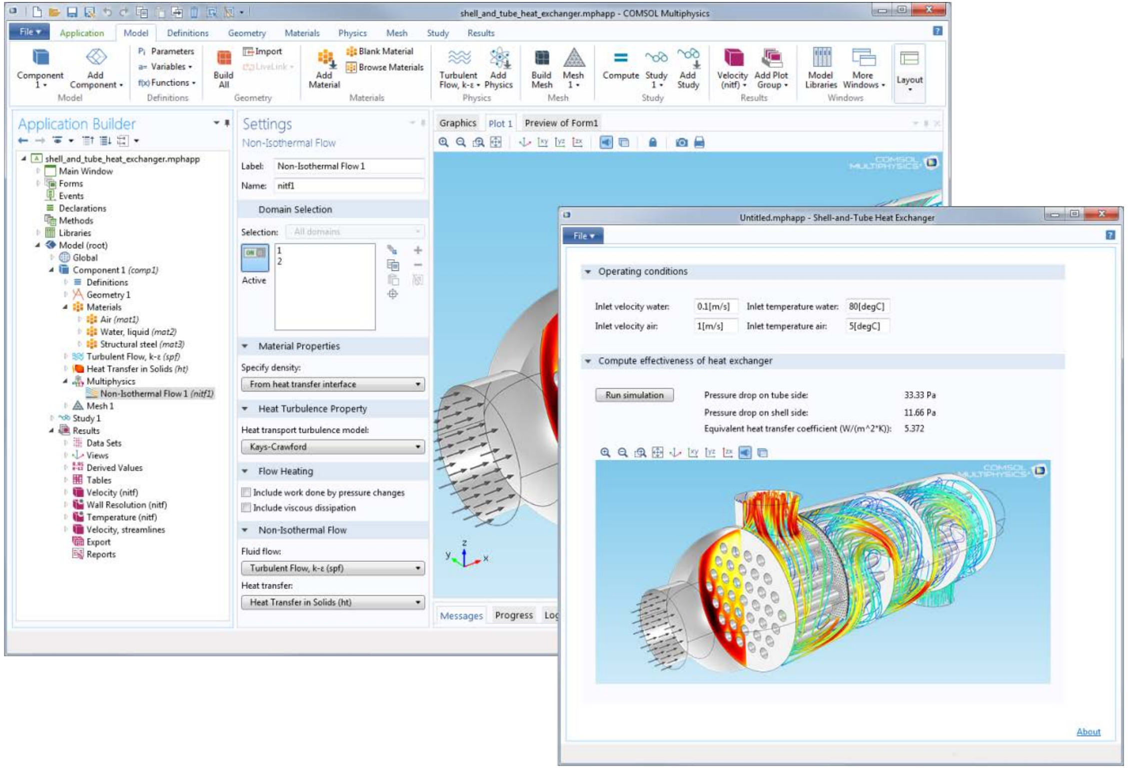
Building Apps

Running Apps

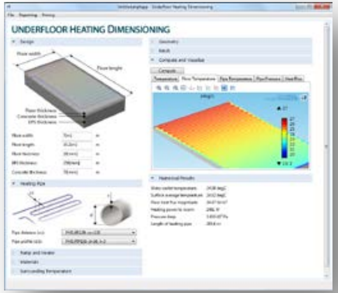


COMSOL

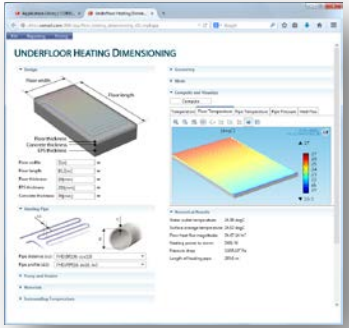
From Model to App



Running Apps



COMSOL Client



Web Browser

iPad

iPhone

COMSOL