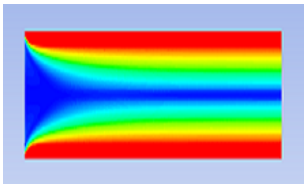
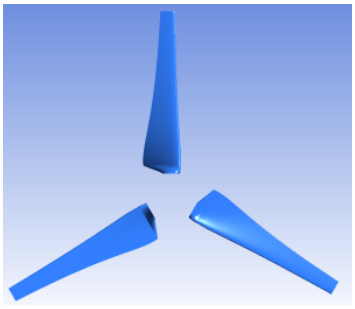
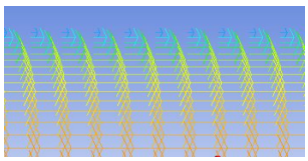
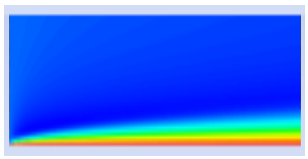
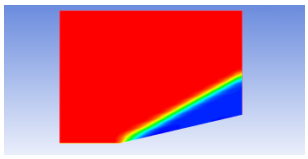
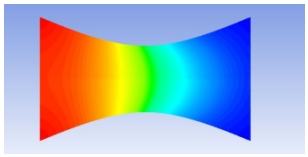
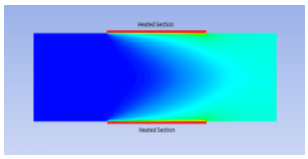
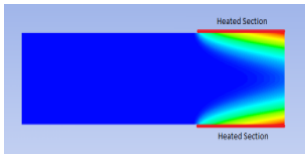
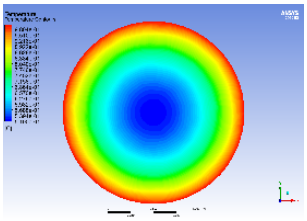
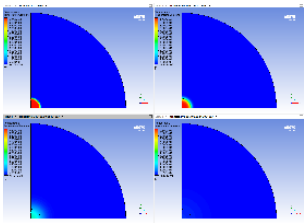
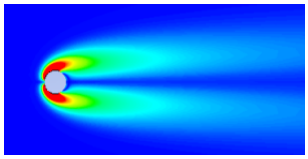
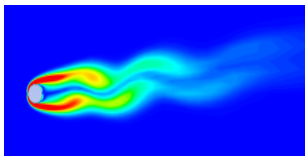
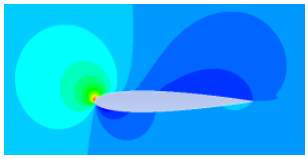
FLUENT Learning Modules

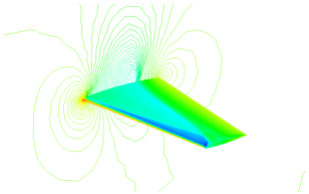
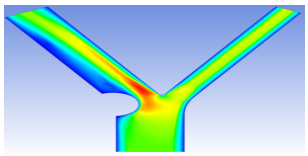
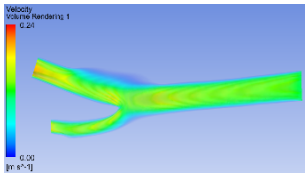
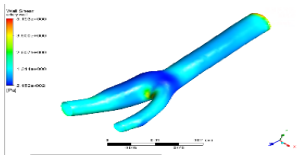
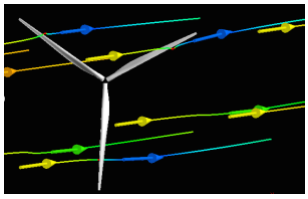
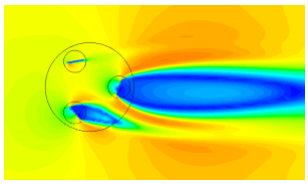
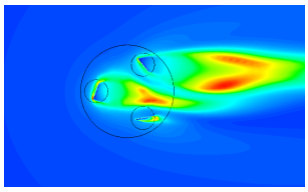
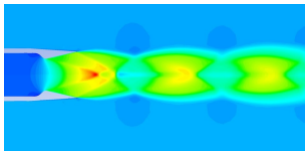
List of learning modules

The following tutorials show how to solve selected fluid flow problems using *ANSYS Fluent*. The tutorial topics are drawn from Cornell University courses, the Prantil et al textbook, student/research projects etc. If a tutorial is from a course, the relevant course number is indicated below. All tutorials have a common structure and use the same high-level steps starting with *Pre-Analysis* and ending with *Verification and Validation*. *Pre-Analysis* includes hand calculations to predict expected results while *Verification and Validation* can be thought of as a formal process for checking computer results. Both these steps are extremely important in practice though often overlooked. The pedagogical philosophy behind these modules is discussed in this article from the *ANSYS Advantage* magazine.

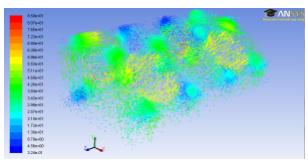
Fluent tutorials

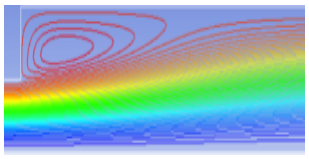
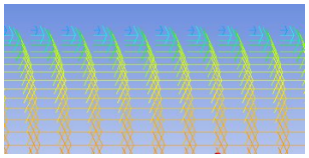
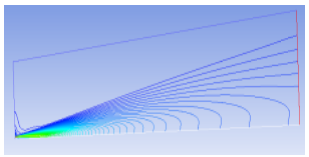
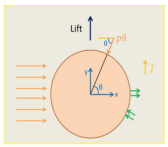
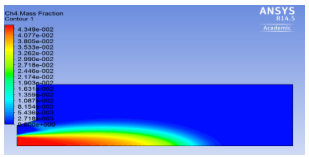
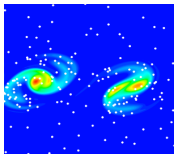
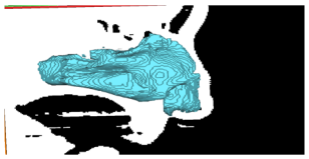
	What's Under the CFD Blackbox?	ENGR2000X A Hands-on Introduction to Engineering Simulations
$\rho(\vec{V}\cdot\nabla)\vec{V} = -\nabla p + \mu\nabla^2\vec{V}$ $\nabla\cdot\vec{V}=0$	Big Ideas: Fluid Dynamics	ENGR2000X A Hands-on Introduction to Engineering Simulations
$\dot{V}_{1-2} = u_{1-2}\,\Delta y\,(1) + \text{Error}$ $u_{1-2} = \frac{u_1+u_2}{2} + \text{Error}$ $\dot{V}_{1-2} = \left(\frac{u_1+u_2}{2}\right)\Delta y + 2\,\text{Errors}$	Computational Fluid Dynamics	ENGR2000X A Hands-on Introduction to Engineering Simulations
	Laminar Pipe Flow	ENGR2000X A Hands-on Introduction to Engineering Simulations
	Wind Turbine Blade	MAE 4020/4021/5020
	Turbulent Pipe Flow	MAE 4230/MAE 5230
	Flat Plate Boundary Layer	MAE 4230/MAE 5230/MAE 6510

	Supersonic Flow Over a Wedge	MAE 4230/MAE 5230
	Compressible Flow in a Nozzle	MAE 4230/MAE 5230
	Turbulent Forced Convection	MAE 4272
	Laminar Convection: Nusselt Problem	MAE 3240 (Summer)
	3D Steady Diffusion	BME 2000
	2D Transient Diffusion	BME 2000
	Steady Flow Past a Cylinder	MAE 5230
	Unsteady Flow Past a Cylinder	MAE 5230
	Flow over an Airfoil	MAE 5230

	3D Transonic Flow over a Wing	MAE 5230
	2D Bifurcating Artery	MAE 6690
	3D Bifurcating Artery (steady)	BME 2000
	3D Bifurcating Artery (unsteady)	MAE 6690
	Wind Turbine Blade FSI (Part 1) (Advanced)	MAE 5230/MAE 5020
	Vertical Axis Wind Turbine (Part 1) (Advanced)	
	Vertical Axis Wind Turbine (Part 2) (Advanced)	
	Over-expanded Nozzle (Advanced) [Under-Construction]	MAE 4230

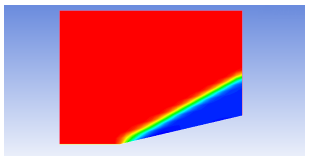
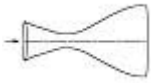
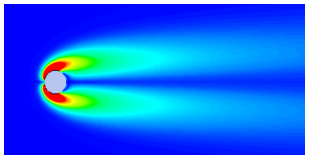
Some other Fluent tutorials

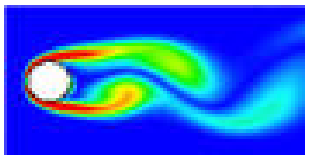
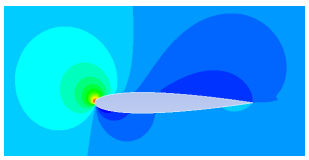
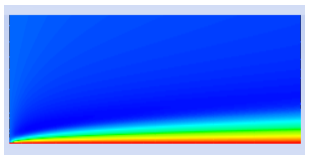
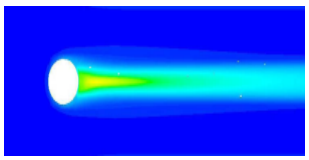
	3D Convection through an Electronics Box	Mars Rover Project

	Sudden Expansion in Laminar Pipe Flow	University of California, Irvine
	Turbulent Pipe Flow (LES)	Stephen B. Pope Research Group
	Turbulent Jet (k-epsilon model)	MAE 7340
	User Defined Functions	MAE 4230//MAE 5230
	Partially Premixed Combustion	
	Particles in a Periodic Double Shear Layer	Donald L Koch Research Group
	Nasal Airway Model	Cornell M.Eng Project

Older Tutorials

These tutorials are for older version of FLUENT and are no longer being updated. They use GAMBIT for meshing.

	Supersonic Flow Over a Wedge	ANSYS FLUENT 6.3
	Compressible Flow in a Nozzle	ANSYS FLUENT 6.3
	Steady Flow Past a Cylinder	ANSYS FLUENT 6.3

	Unsteady Flow Past a Cylinder	ANSYS FLUENT 6.3
	Flow over an Airfoil	ANSYS FLUENT 6.3
	Forced Convection over a Flat Plate	ANSYS FLUENT 6.3
	Flow past a Sphere	ANSYS FLUENT 6.3

Computational Fluid Dynamics