ANSYS Learning Modules

What is ANSYS?

ANSYS is a finite-element analysis package used widely in industry to simulate the response of a physical system to structural loading, and thermal and electromagnetic effects. ANSYS uses the finite-element method to solve the underlying governing equations and the associated problem-specific boundary conditions.

About the ANSYS learning modules

This ANSYS short course consists of a set of learning modules on using ANSYS to solve problems in solid mechanics. The learning modules lead the user through the steps involved in solving a selected set of problems using ANSYS. We not only provide the solution steps but also the rationale behind them. It is worthwhile for the user to understand the underlying concepts as she goes through the learning modules in order to be able to correctly apply ANSYS to other problems. The user would be ill-served by clicking through the learning modules in zombie-mode. Each learning module is followed by problems which are geared towards strengthening and reinforcing the knowledge and understanding gained in the learning modules. Working through the problem sets is an intrinsic part of the learning process and shouldn't be skipped.

These learning modules have been developed by the Swanson Engineering Simulation Program in the Sibley School of Mechanical and Aerospace Engineering at Cornell University. The Swanson Engineering Simulation Program has been established with the goal of integrating computer-based simulations into the mechanical engineering curriculum. This program has been endowed by Dr. John Swanson, the founder of ANSYS Inc. and an alumnus of the Sibley School. The development of these learning modules is being supported by a Faculty Innovation in Teaching award from Cornell University.

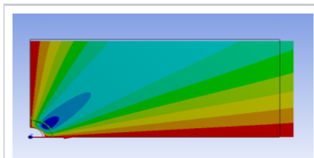
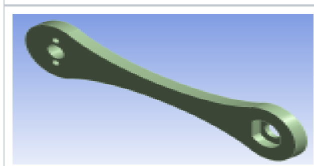
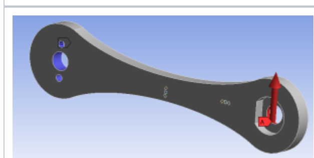
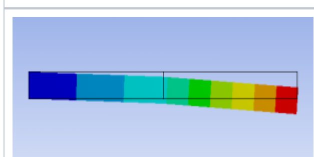
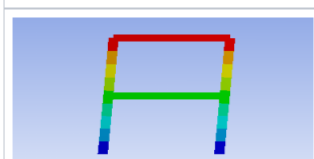
List of Learning Modules

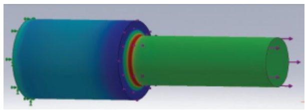
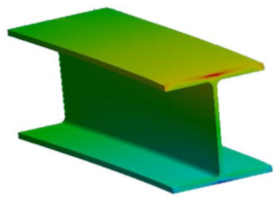
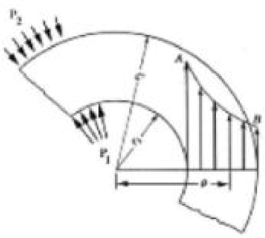
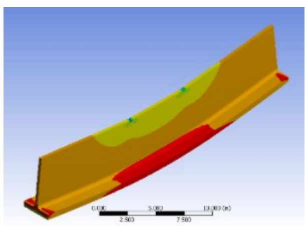
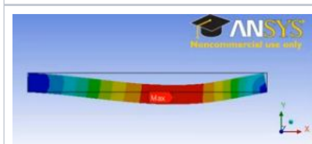
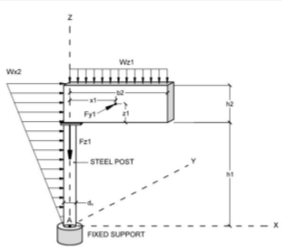
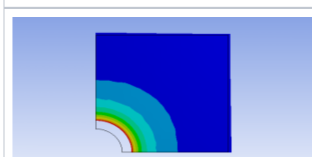
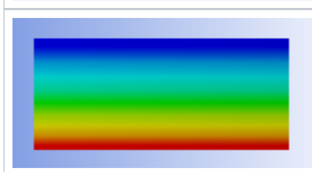
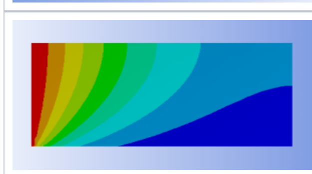
Each learning module below contains a step-by-step tutorial that shows details of how to solve a selected problem using ANSYS, a popular tool for finite-element analysis (FEA). The tutorial topics are drawn from Cornell University courses, the Prantil et al textbook, student/research projects etc. If 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.

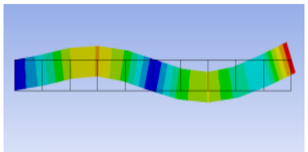
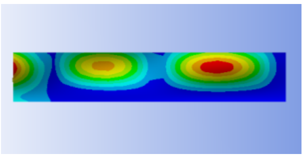
Finite Element Analysis Using ANSYS Mechanical

The following ANSYS tutorials show you how to obtain an FEA solution from scratch using *ANSYS Mechanical*.

Introductory Tutorials

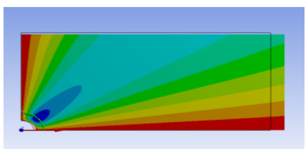
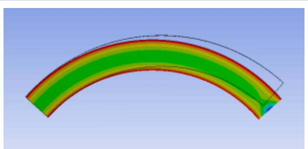
	Plate With a Hole	MAE 3250/ MAE 4700--5700	Static Structural
	Bike Crank	MAE 3250/MAE 3272	Static Structural
	Bike Crank: Part 2	MAE 3272	Static Structural
	Cantilever Beam	MAE 4700-5700	Static Structural
	Plane Frame	MAE 4700-5700	Static Structural

	A stepped shaft in axial tension	Prantil et al textbook	Static Structural
	A non-slender cantilever beam under point tip loading	Prantil et al textbook	Static Structural
	Hoop and axial stresses in thick-walled pressure vessels	Prantil et al textbook	Static Structural
	A four-point bend test on a T-beam	Prantil et al textbook	Static Structural
	Planar approximations for a two-dimensional beam analysis	Prantil et al textbook	Static Structural
	Three-dimensional analysis of combined loading in a signpost	Prantil et al textbook	Static Structural
	Plate With a Hole: Optimization	MAE 3250/MAE 4700-5700	Optimization
	Heat Conduction in a Cylinder	MAE 4700-5700	Heat Transfer
	2D Steady Conduction in a Rectangular Domain	MAE 3240/ MAE 6510	Heat Transfer

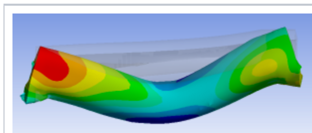
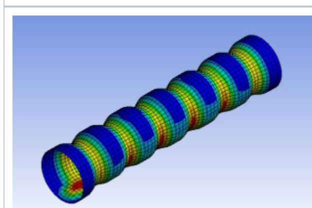
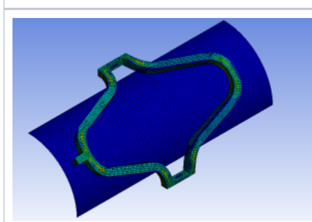
	Cantilever Beam Modal Analysis	MAE 4700-5700	Dynamics
	Modal Analysis of a Wing		Dynamics

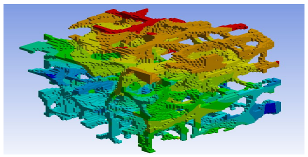
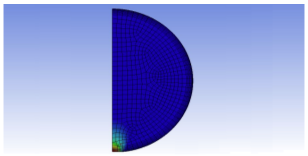
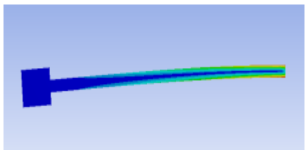
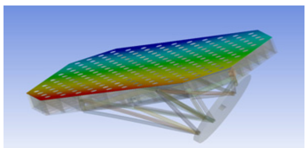
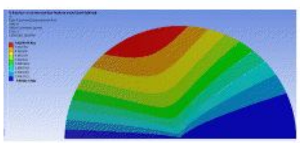
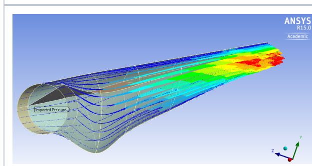
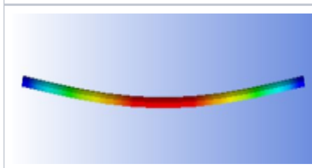
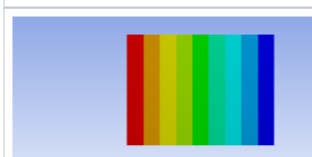
Finite Element Analysis Using ANSYS Mechanical: Results-Interpretation

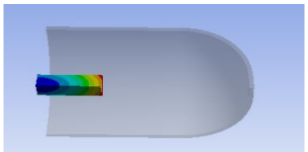
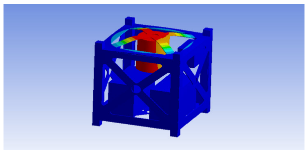
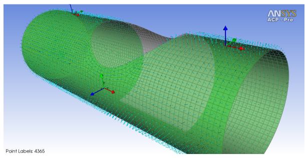
The following ANSYS tutorials focus on the *interpretation and verification* of FEA results (rather than on obtaining an FEA solution from scratch). The ANSYS solution files are provided as a download. We read the solution into *ANSYS Mechanical* and then move directly to reviewing the results critically. We are particularly interested in the comparison of FEA results with hand calculations.

	Tensile Bar	MAE 3250	Static Structural
	Plate With a Hole	MAE 3250	Static Structural
	Bending of a Curved Beam	MAE 3250	Static Structural

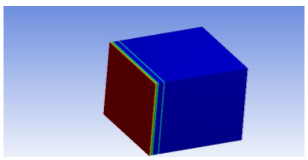
Advanced Tutorials

	Rat Femur	BME 4490	Static Structural
	Trachea Analysis	BME 2000	Static Structural
Under Construction	Bone Compression		Static Structural
	Cardiovascular Stent		Static Structural

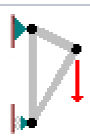
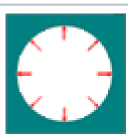
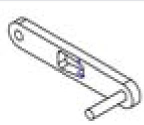
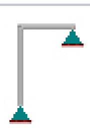
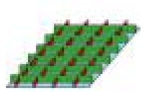
	High Resolution FE Model of Bone	MAE 6640	Static Structural
	Hertz Contact Mechanics	Undergrad Project	Static Structural
	Stress due to Gravity		Static Structural
	Advanced FEA for Large Telescope Truss	CCAT Telescope Project	Static Structural
	Crack Between Neo-Hookean Material and Rigid Body	MAE 5700	Static Structural
	Wind Turbine Blade FSI (Part 2)	MAE 4020-5020	Static Structural, FSI
	Linear Column Buckling		Structural
	Thermal Stresses in a Bar		Coupled Static Structural and Heat Transfer
	Transient 2D Conduction		Heat Transfer
	3D Conduction		Heat Transfer

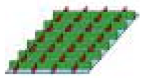
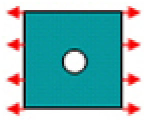
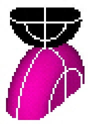
	Radiation Between Surfaces		Heat Transfer
	Modal Analysis of a Satellite	Cornell CubeSat Team	Dynamics
	Modal Analysis of a Composite Monocoque	Cornell Formula SAE team	

Tips and tricks

	Tips and Tricks

Finite Element Analysis Using ANSYS APDL (These tutorials are no longer being updated)

	Two-Dimensional Static Truss	ANSYS 11.0 12.0 APDL	Basic
	Plate with a hole	ANSYS 11.0 12.0 APDL	Basic
	Three-dimensional bicycle crank	ANSYS 12.0 APDL	Intermediate
	Three-dimensional curved beam	ANSYS 11.0 APDL	Intermediate
	Vibration analysis of a frame	ANSYS 7.0	Intermediate
	Semi-monocoque shell	ANSYS 10.0 APDL	Intermediate

	Semi-monocoque shell, Part 2: Parametric study	ANSYS 10.0 APDL	Intermediate
	Orthotropic plate with a hole	ANSYS 11.0 12.0 APDL	Intermediate
	Disks in point contact	ANSYS 7.1 Classic	Intermediate

	Frequently Asked Questions