

# 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.

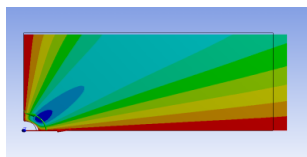
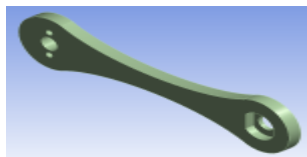
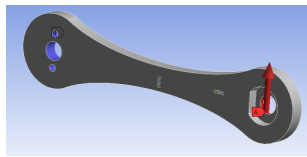
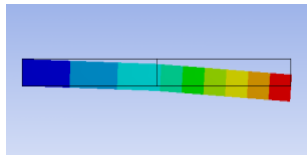
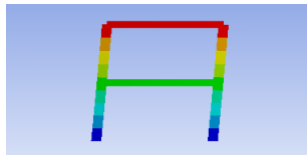
## List of Learning Modules

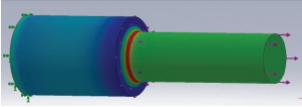
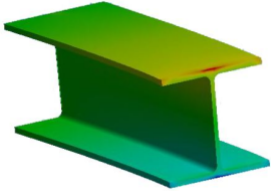
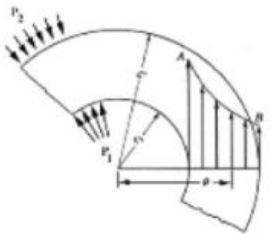
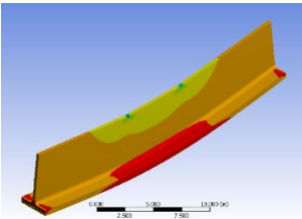
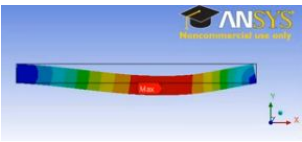
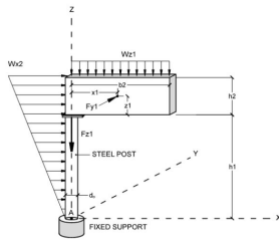
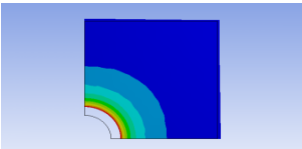
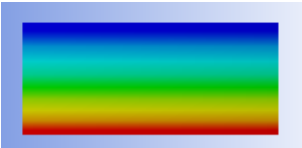
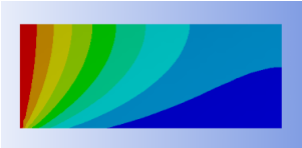
Each learning module below contains a step-by-step tutorial that shows details of how to solve a selected problem using ANSYS, a popular tool for finite-element analysis (FEA). The tutorial topics are drawn from Cornell University courses, the [Prantl et al](#) textbook, student/research projects etc.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

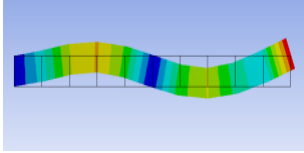
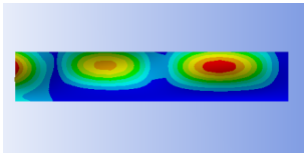
## Finite Element Analysis Using ANSYS Mechanical

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

### Introductory Tutorials

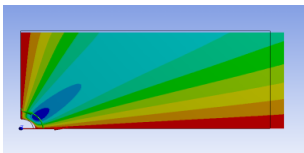
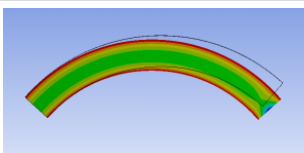
|                                                                                     |                                    |                                          |                   |
|-------------------------------------------------------------------------------------|------------------------------------|------------------------------------------|-------------------|
|  | <a href="#">Plate With a Hole</a>  | <a href="#">MAE 3250/ MAE 4700--5700</a> | Static Structural |
|  | <a href="#">Bike Crank</a>         | <a href="#">MAE 3250/MAE 3272</a>        | Static Structural |
|  | <a href="#">Bike Crank: Part 2</a> | <a href="#">MAE 3272</a>                 | Static Structural |
|  | <a href="#">Cantilever Beam</a>    | <a href="#">MAE 4700-5700</a>            | Static Structural |
|  | <a href="#">Plane Frame</a>        | <a href="#">MAE 4700-5700</a>            | 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     |

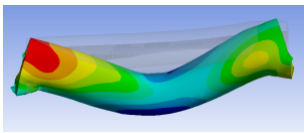
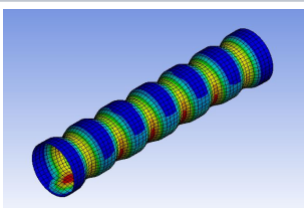
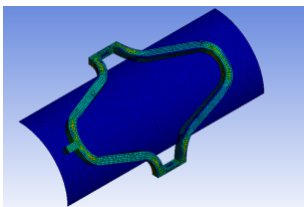
|                                                                                   |                                |               |          |
|-----------------------------------------------------------------------------------|--------------------------------|---------------|----------|
|  | Cantilever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

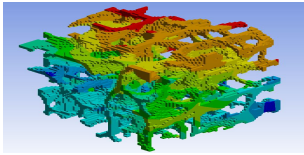
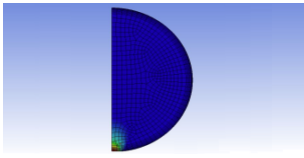
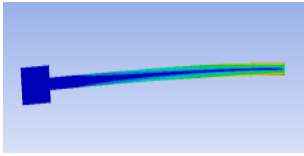
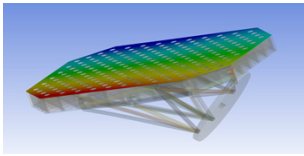
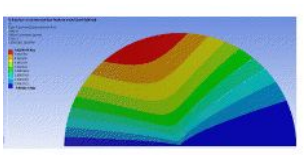
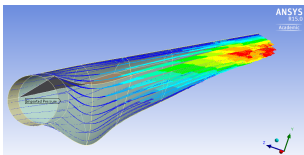
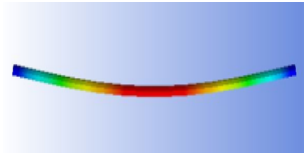
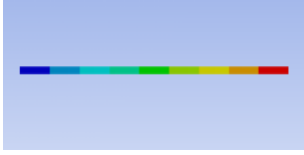
### Finite Element Analysis Using ANSYS Mechanical: Results-Interpretation

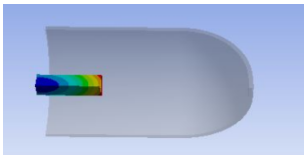
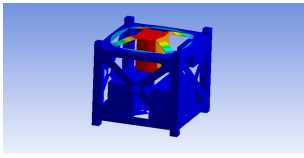
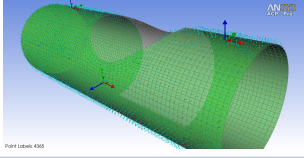
The following ANSYS tutorials focus on the *interpretation and verification* of FEA results (rather than on obtaining an FEA solution from scratch). The ANSYS solution files are provided as a download. We read the solution into *ANSYS Mechanical* and then move directly to reviewing the results critically.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

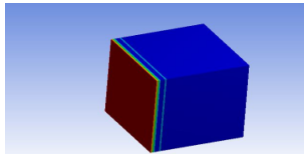
### Advanced Tutorials

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

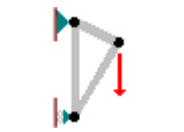
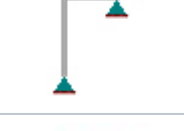
|                                                                                     |                                                   |                        |                                             |
|-------------------------------------------------------------------------------------|---------------------------------------------------|------------------------|---------------------------------------------|
|    | 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                               |

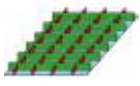
|                                                                                   |                                         |                          |               |
|-----------------------------------------------------------------------------------|-----------------------------------------|--------------------------|---------------|
|  | 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 |

|                                                                                   |                                                                    |                      |              |
|-----------------------------------------------------------------------------------|--------------------------------------------------------------------|----------------------|--------------|
|  | <a href="#">Semi-monocoque shell,<br/>Part 2: Parametric study</a> | ANSYS 10.0 APDL      | Intermediate |
|  | <a href="#">Orthotropic plate with a hole</a>                      | ANSYS 11.0 12.0 APDL | Intermediate |
|  | <a href="#">Disks in point contact</a>                             | ANSYS 7.1 Classic    | Intermediate |

|                                                                                   |                                            |
|-----------------------------------------------------------------------------------|--------------------------------------------|
|  | <a href="#">Frequently Asked Questions</a> |
|-----------------------------------------------------------------------------------|--------------------------------------------|