Introductory Tutorials

ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

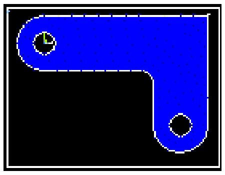
Release 19.2
August 2018

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Published in the U.S.A.

Table of Contents

Welcome to the Mechanical APDL Introductory Tutorials



The Mechanical APDL Introductory Tutorials provide an introduction to the extensive capabilities of the Mechanical APDL family of products. Each tutorial is a complete step-by-step analysis procedure. You can choose from several analysis disciplines. The tutorials are designed to be run interactively, on the same screen as the program. Included are full color graphics and animations that are exact replicas of what appear at several points within the steps of the tutorials. A glossary of terms is also included that you can view as a stand-alone document with an alphabetical listing of the terms, or you can view the definition of terms on demand by simply clicking on linked terms within the context of the tutorials.

Before you begin a tutorial, read Start Here (p. 1) for recommendations on preparing your screen for displaying the tutorial window on the same screen as the program, as well as descriptions of the formats and conventions used in the tutorials.

Chapter 1: Start Here

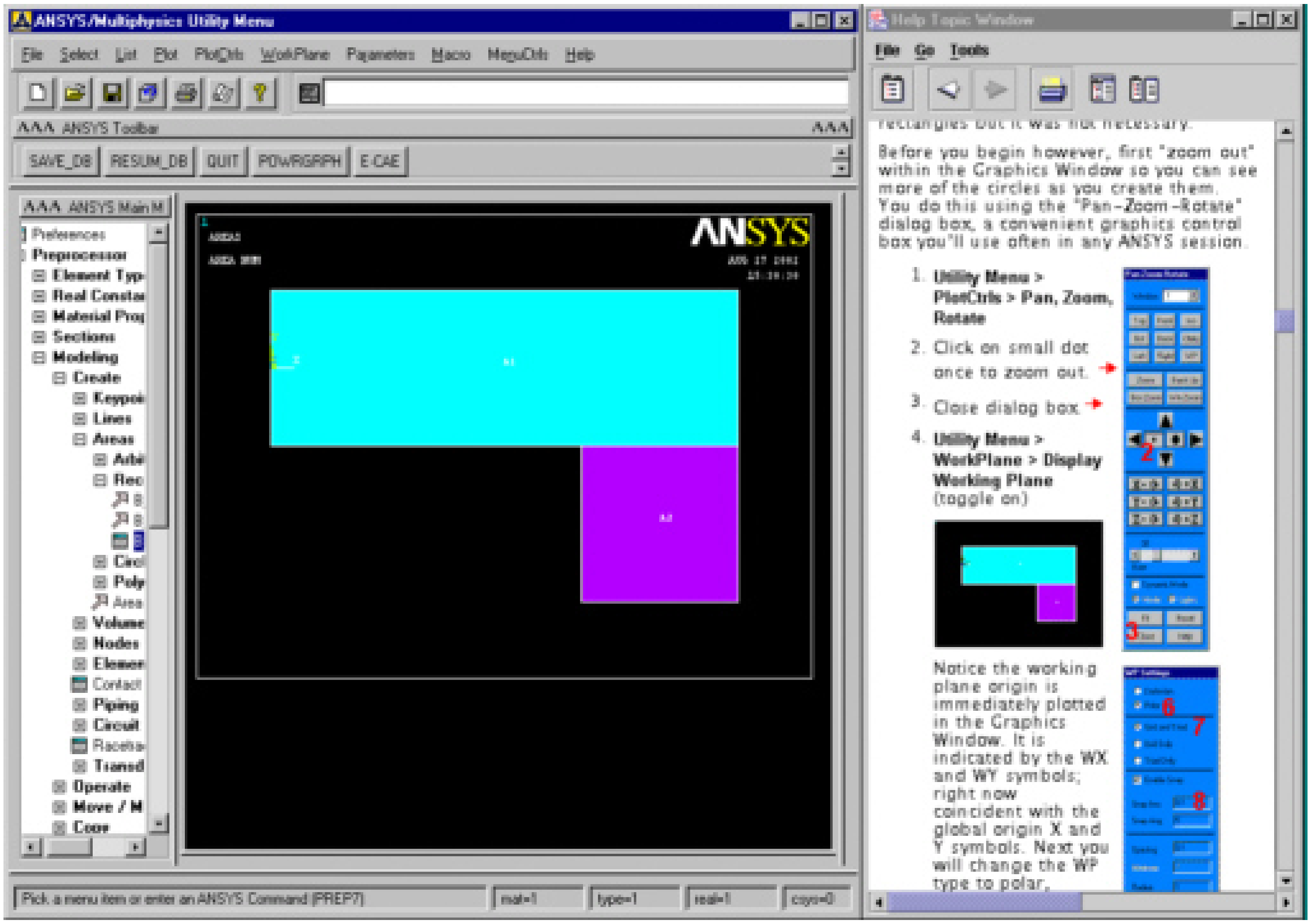
1.1. About These Tutorials

The purpose of these tutorials is to introduce you to the extensive analysis capabilities of Mechanical APDL.

1.1.1. Preparing Your Screen

Running the tutorials online *while* running Mechanical APDL requires that you make the best use of your monitor's display area. With just minor adjustments to the browser and the Mechanical APDL interface, you can read a tutorial's instructions on one side of your screen, and perform the instructions in Mechanical APDL on the other.

Following is an example layout captured on a 21-inch monitor. It is a typical representation of how a display appears while running a tutorial using the Windows version of Mechanical APDL.



The tutorial window containing the tabs was removed by clicking the **Undock** button (large button located furthest to the right), then minimizing the tabbed window.

The tutorial window is moved to the right side of the screen, and the Mechanical APDL window is reduced horizontally to accommodate the tutorial window. Use this layout as an example and adjust your own display, based on the size of your monitor.

If a Mechanical APDL dialog box appears on top of either a tutorial or the Mechanical APDL window, move it anywhere on the screen by dragging the window header.

1.1.2. Formats and Conventions Used

Each tutorial begins with a problem description that includes approaches and assumptions. A summary of the tasks required to complete the tutorial is provided.

1.1.2.1. Tasks

A task contain the specific *actions* needed to complete the task. Example tasks include "Add areas," "Define material properties," "Mesh the area," and "Plot the deformed shape." A typical tutorial requires approximately 20 tasks, with the number varying according to analysis complexity.

1.1.2.2. Actions

For each task, individually numbered actions guide you through the task.

A menu path is typically one of the first action within a task. An example of a menu path action is:

1. **Main Menu> Preprocessor> Modeling> Operate> Booleans> Add> Areas**

A menu path represents the complete location of a particular function in the Graphical User Interface (GUI). The first part of the path (Main Menu) determines where the function is found. It is usually either the Main Menu or the Utility Menu. Go to that region to perform the function. The remaining part of the path lists the menu topics that you select with the mouse.

Actions presented after a menu path either guide you through completing a dialog box, or show you how to pick graphical locations (p. 3).

For completing a dialog box, some actions are followed by a red arrow (→) indicating that a small picture of the dialog box is available if you scroll to the right. The picture includes large red numbers, each referencing the corresponding action. (The numbers are positioned in the dialog box at the locations where you are to perform the actions (button, box, drop-down list, etc.).)

Task actions in several of the tutorials adhere to the following conventions:

- Items that you need to fill in reproduce the wording in the dialog boxes and are in quotes, followed by an equal sign, then the value you should enter. Example:

  3. "Load VOLT Value" = 5

- Button labels are in brackets. Example:

  4. [Pick All]

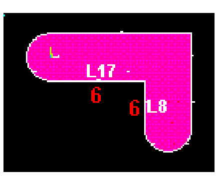
- Actions, locations, or any other items that may not be obvious are enclosed in parentheses before or after GUI wording in quotes. Examples:

  2. (double-click) "Structural"
  5. (drop down) "Action"
  7. "Scaler Tet 98" (right column)

1.1.2.3. Picking Graphics

Some task actions instruct you to pick specific entities on a graphic. An example of the convention is shown:

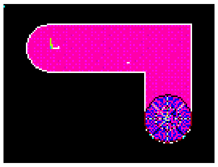
6. Pick lines 17 and 8



Here, red numbers are displayed on the picture at the locations where you are to pick. The red number is a cross-reference to the action.

1.1.2.4. Interim Result Graphics

Following the required actions, a task typically concludes with a small interim result graphic showing how the Mechanical APDL graphic should appear in the Graphics window at a given point in the tutorial. An example is shown:



1.1.3. Jobnames and Preferences

Specify a jobname (p. 6) for each tutorial analysis. The jobname helps to identify files generated by Mechanical APDL that are related to a specific analysis. When starting Mechanical APDL, you can specify the jobname in the launcher. While working in Mechanical APDL, you can change the jobname by selecting:

Utility Menu > File > Change Jobname

then typing the jobname, and selecting OK.

It is good practice to set preferences (p. 7) for each tutorial analysis. When setting a preference for a given engineering discipline, Mechanical APDL displays only those menu options that apply to the specified discipline. (If you do not set preferences, menu options for *all* disciplines appear, but non-applicable options are dimmed based on the element types used in the model.)

Set preferences at or near the beginning of an analysis. (Most of the tutorials include this step before the model is meshed.) You can set a preference by selecting:

Main Menu > Preferences

then checking the box associated with the desired engineering discipline, and selecting OK.

1.1.4. Selecting a Tutorial

Try running Structural Tutorial (p. 9) first, even if you typically run analyses in other engineering disciplines. The Structural tutorial is documented extensively, includes graphics of all dialog boxes used, and introduces you to Mechanical APDL terminology used in other tutorials.

After successfully performing the Structural tutorial, feel free to try any of the others in any order. You can select a problem that demonstrates the Mechanical APDL features in your discipline, but *all* introductory tutorials demonstrate universal Mechanical APDL analysis techniques in some way. You can learn something from every tutorial.

You can access a tutorial via the Table of Contents or by clicking on the name of the tutorial in the following list:

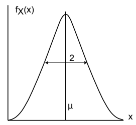
- Structural Tutorial (p. 9)
- Thermal Tutorial (p. 29)
- Electromagnetics Tutorial (p. 45)
- Micro-Electromechanical System (MEMS) Tutorial (p. 57)
- Contact Tutorial (p. 71)
- Modal Tutorial (p. 87)

- ANIMATE Program (p. 97)

1.2. Glossary

Analysis Options	Typical analysis options are the method of solution, stress stiffening on or off, and Newton-Raphson options for nonlinearities.
Analysis Type	Any of seven analysis types offered in Mechanical APDL: static, modal, harmonic, transient, spectrum, eigenvalue buckling, and substructuring. Whether the problem is linear or nonlinear will be identified here.
Applicable Products	Indicates which Mechanical APDL licenses can be used to run the example problem. Applicable products are determined by the discipline and complexity of the problem. Possibilities include: ANSYS Multiphysics, ANSYS Mechanical, ANSYS Professional, ANSYS Structural, ANSYS Emag, ANSYS PrepPost.
Boolean Operations	Boolean Operations (based on Boolean algebra) provide a means of combining sets of data using such logical operators as add, subtract, intersect, etc. There are Boolean operations available for volume, area, and line solid model entities.
Direct Element Generation	Defining an element by defining nodes directly.
Discipline	Any of five physical (engineering) disciplines may be solved by the Mechanical APDL program: structural, thermal, electric, magnetic, and fluid.
Element Options	Many element types also have additional *element options* to specify such things as element behavior and assumptions, element results printout options, etc.
Element Types Used	Indicates the element types used in the problem; over 100 element types are available in Mechanical APDL. You select an element type which characterizes, among other things, the degree-of-freedom set (displacements and/or rotations, temperatures, etc.) the characteristic shape of the element (line, quadrilateral, brick, etc.), whether the element lies in 2-D space or 3-D space, the response of your system, and the accuracy level you're interested in.
Features Demonstrated	Lists the noteworthy Mechanical APDL features demonstrated in the problem.
Gaussian Distribution	The Gaussian or normal distribution is a very fundamental and commonly used distribution for statistical matters. It is typically used to describe the scatter of the measurement data of many physical phenomena. Strictly speaking, every random variable follows a normal distribution if it is generated by a linear combination of a very large number of other random effects, regardless which distribution these random effects originally follow. The Gaussian distribution is also valid if the random variable is a linear combination of two or more other effects if those effects also follow a Gaussian distribution.

You provide values for the mean value μ and the standard deviation σ of the random variable *x*.



Help Resources

Information in the Mechanical APDL help system that is relevant to the overall topics covered in a particular tutorial.

Higher-Order Elements

Higher-order, or midside-node elements, have a quadratic shape function (instead of linear) to map degree-of-freedom values within the element.

Interactive Time Required

This is an approximate range, in minutes, for you to complete the interactive step-by-step solution. Of course the amount of time it takes you to perform the problem depends on the computer system you use, the amount of network "traffic" on it, the working pace that is comfortable for you, and so on.

Jobname

The file name prefix used for all files generated in an analysis. All files are named *Jobname.ext*, where *ext* is a unique Mechanical APDL extension that identifies the contents of the file. The jobname specified in the launcher when you start Mechanical APDL is called the *initial* jobname. You can always change the jobname within a Mechanical APDL session.

Latin Hypercube Sampling

The Latin Hypercube Sampling (LHS) technique is a Monte Carlo Simulation method that is more advanced and efficient than the Direct Monte Carlo Sampling technique. LHS has a sample "memory," meaning it avoids repeating samples that have been evaluated before (it avoids clustering samples). It also forces the tails of a distribution to participate in the sampling process.

Level of Difficulty

Three levels are offered: easy, moderate, and advanced. Although the "advanced" problems are still easy to follow using the interactive step-by-step solution, they include features that are typically thought of as advanced Mechanical APDL capabilities, such as nonlinearities, macros, or advanced postprocessing.

Lognormal Distribution

The lognormal distribution is a basic and commonly used distribution. It is typically used to describe the scatter of the measurement data of physical phenomena, where the logarithm of the data would follow a normal distribution. The lognormal distribution is very suitable for phenomena that arise from the multiplication of a large number of error effects. It is also correct to use the lognormal distribution for a random variable that is the result of multiplying two or more random effects (if the effects that get multiplied are also lognormally distributed).

Material Properties

Physical properties of a material such as modulus of elasticity or density that are independent of geometry. Although they are not necessarily tied to the element type, the material properties required to solve the element matrices are listed for each element type for your convenience. Depending on the application, material properties may be linear, nonlinear, and/or anisotropic. As with element types and real constants, you may have multiple material property sets (to correspond with multiple materials) within one analysis. Each set is given a reference number.

Monte Carlo
: The Monte Carlo Simulation method is the most common and traditional method for a probabilistic analysis. This method lets you simulate how virtual components behave the way they are built. One simulation loop represents one manufactured component that is subjected to a particular set of loads and boundary conditions.

Plane Stress
: A state of stress in which the normal stress and the shear stresses directed perpendicular to the plane are assumed to be zero.

Postprocessing
: Mechanical APDL analysis phase where you review the results of the analysis through graphics displays and tabular listings. The general postprocessor (POST1) is used to review results at one substep (time step) over the entire model. The time-history postprocessor (POST26) is used to review results at specific points in the model over all time steps.

Preferences
: The "Preferences" dialog box allows you to select the desired engineering discipline for context filtering of menu choices. By default, menu choices for all disciplines are shown, with non-applicable choices "dimmed" based on a set of element types in your model. If you prefer not to see the dimmed choices at all, you should turn on filtering. For example, turning on structural filtering completely suppresses all thermal, electromagnetic, and fluid menu topics.

Preprocessing
: Mechanical APDL analysis phase where you provide data such as the geometry, materials, and element types to the program.

Primitives
: Simple predefined geometric shapes that Mechanical APDL provides. A rectangle primitive, for example defines the following solid model entities in one step: one area, four lines, and four keypoints.

Probabilistic Analysis File
: A probabilistic analysis file is a Mechanical APDL input file that contains a complete analysis sequence (preprocessing, solution, postprocessing). It must contain a parametrically defined model using parameters to represent all inputs and outputs which will be used as random input variables (RVs) and random output parameters (RPs). From this file, a probabilistic design loop file (Jobname.LOOP) is automatically created and used by the probabilistic design system to perform analysis loops.

Probabilistic Design
: Probabilistic Design is a technique you can use to assess the effect of uncertain input parameters and assumptions on your analysis model. Using a probabilistic analysis you can find out how much the results of a finite element analysis are affected by uncertainties in the model.

Probabilistic Simulation
: A simulation is the collection of all samples that are required or that you request for a certain probabilistic analysis. A simulation contains the information used to determine how the component would behave under real-life conditions (with all the existing uncertainties and scatter), and all samples therefore represent the simulation of this behavior.

Random Input Variables

Random Input Variables (RVs) are quantities that influence the result of an analysis. In probabilistic literature, these random input variables are also called the "drivers" because they drive the result of an analysis.

Random Output Parameters

Random Output Parameters (RPs) are the results of a finite element analysis. The RPs are typically a function of the random input variables (RVs); that is, changing the values of the random input variables should change the value of the random output parameters.

Real Constants

Provide additional geometry information for element types whose geometry is not fully defined by its node locations. Typical real constants include shell thicknesses for shell elements and cross-sectional properties for beam elements. All properties required as input for a particular element type are entered as one *set* of real constants.

Solution

Mechanical APDL analysis phase where you define analysis type and options, apply loads and load options, and initiate the finite element solution. A new, static analysis is the default.

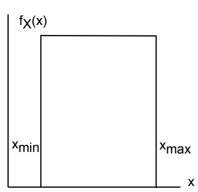
Standard Deviation

The standard deviation is a measure of variability (dispersion or spread) about the arithmetic mean value; this is often used to describe the width of the scatter of a random output parameter or of a statistical distribution function. The larger the standard deviation the wider the scatter and the more likely it is that there are data values further apart from the mean value.

Uniform Distribution

The uniform distribution is fundamental for cases where no other information apart from a lower and an upper limit exists. It is useful to describe geometric tolerances. It can also be used in cases where no evidence exists that any value of the random variable is more likely than any other within a certain interval.

You provide the lower and the upper limit x_{min} and x_{max} of the random variable x.

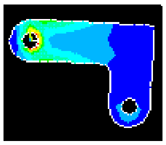


Working Plane (WP)

An imaginary plane with an origin, a 2-D coordinate system (either Cartesian or Polar), a snap increment, and a display grid. It is used to locate solid model entities. By default, the working plane is a Cartesian plane located at the global origin.

Chapter 2: Structural Tutorial

Static Analysis of a Corner Bracket



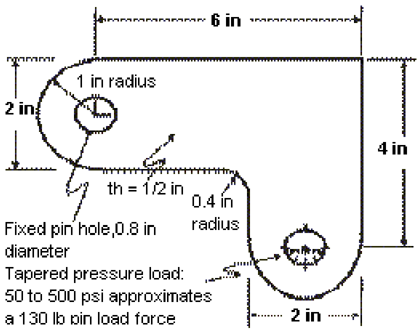
2.1. Static Analysis of a Corner Bracket

2.1.1. Problem Specification

Applicable Products: (p. 5)	ANSYS Multiphysics, ANSYS Mechanical, ANSYS Structural
Level of Difficulty: (p. 6)	Easy
Interactive Time Required: (p. 6)	60 to 90 minutes
Discipline: (p. 5)	Structural
Analysis Type: (p. 5)	Linear static
Element Types Used: (p. 5)	PLANE183
Features Demonstrated: (p. 5)	Solid modeling including primitives, Boolean operations, and fillets; tapered pressure load; deformed shape and stress displays; listing of reaction forces; examination of structural energy error
Help Resources: (p. 6)	Structural Static Analysis and PLANE183

2.1.2. Problem Description

This is a simple, single-load-step, structural static analysis of a corner angle bracket. The upper-left pin hole is constrained (welded) around its entire circumference, and a tapered pressure load is applied to the bottom of the lower-right pin hole. The US Customary system of units is used.



Objective: Demonstrate a typical Mechanical APDL analysis procedure.

2.1.2.1. Given

The dimensions of the corner bracket are shown in the accompanying figure. The bracket is made of A36 steel with a Young's modulus of 30E6 psi and Poisson's ratio of .27.

2.1.2.2. Approach and Assumptions

Because the bracket is thin in the *z* direction (1/2 inch thickness) compared to its *x* and *y* dimensions, and because the pressure load acts only in the *x*-*y* plane, assume plane stress (p. 7) for the analysis.

Your approach is to use solid modeling to generate the 2-D model and automatically mesh it with nodes and elements. (An alternative approach is to create the nodes and elements directly.)

2.1.2.3. Summary of Steps

Use the information in the problem description and the steps below as a guideline in solving the problem on your own. Or, use the detailed interactive step-by-step solution by selecting the link for step 1.

Build Geometry

1. Define rectangles. (p. 11)

2. Change plot controls and replot. (p. 12)

3. Change working plane to polar and create first circle. (p. 13)

4. Move working plane and create second circle. (p. 15)

5. Add areas. (p. 16)

6. Create line fillet. (p. 16)

7. Create fillet area. (p. 17)

8. Add areas together. (p. 18)

Define Materials

Generate Mesh

Apply Loads

Obtain Solution

Review Results

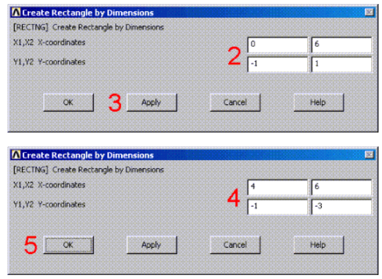
2.1.3. Build the Geometry

This is the beginning of preprocessing (p. 7).

2.1.3.1. Step 1: Define rectangles.

There are several ways to create the model geometry within Mechanical APDL, some more convenient than others. The first step is to recognize that you can construct the bracket easily with combinations of rectangles and circle Primitives (p. 7).

Select an arbitrary *global origin* location, then define the rectangle and circle primitives relative to that origin. For this analysis, use the center of the upper-left hole. Begin by defining a rectangle relative to that location.



1. **Main Menu> Preprocessor> Modeling> Create> Areas> Rectangle> By Dimensions**
2. Enter the following:

   X1 = 0

   X2 = 6

   Y1 = -1

   Y2 = 1

3. Apply to create the first rectangle.
4. Enter the following:

   X1 = 4

   X2 = 6

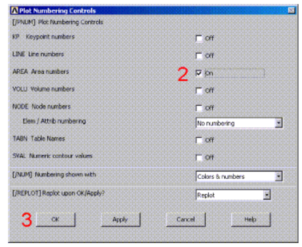
   Y1 = -1

   Y2 = -3

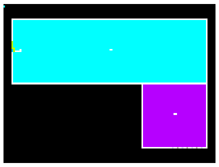
5. OK to create the second rectangle and close the dialog box.

2.1.3.2. Step 2: Change plot controls and replot.

The area plot shows both rectangles, which are *areas*, in the same color. To more clearly distinguish between areas, turn on area numbers and colors. The "Plot Numbering Controls" dialog box on the Utility Menu controls how items are displayed in the Graphics Window. By default, the program performs a replot upon execution of the dialog box. The replot operation repeats the last plotting operation that occurred (in this case, an area plot).



1. **Utility Menu> Plot Ctrls> Numbering**
2. Turn on area numbers.
3. OK to change controls, close the dialog box, and replot.



Save the work you have done thus far. The program stores any input data in memory to the *Mechanical APDL database*. To save the database to a file, use the SAVE operation, available as a tool on the Toolbar.

Mechanical APDL names the database file using the format *jobname.db*. If you started Mechanical APDL using the product launcher, you can specify a jobname (p. 6) at that point. (The default jobname is *file*.)

Check the current jobname at any time via **Utility Menu> List> Status> Global Status**. It is good practice to save the database periodically, especially at specific milestone points in the analysis (for example, after the model is complete, or after meshing the model) via **Utility Menu> File> Save As**, specifying jobnames containing helpful words such as *model*, *mesh*, etc.

If you make a mistake, you can restore the model from the last saved state via RESUME on the Toolbar. (You can also find SAVE and RESUME on the Utility Menu, under File.)

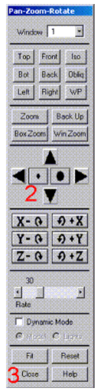
4. Toolbar: **SAVE_DB**.

2.1.3.3. Step 3: Change working plane to polar and create first circle.

Create the half circle at each end of the bracket. You will actually create a full circle on each end and then combine the circles and rectangles with a Boolean (p. 5) add operation (discussed in step 5). To create the circles, you will use and display the working plane (p. 8) (WP).

Before you begin, use the Pan-Zoom-Rotate graphics-control tool to zoom out within the Graphics Window to see more of the circles as you create them.

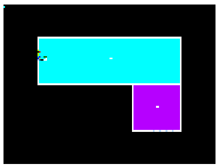
1. **Utility Menu> PlotCtrls> Pan, Zoom, Rotate**



2. Click on small dot once to zoom out.

3. Close dialog box.

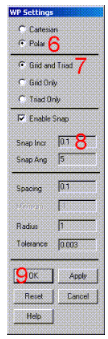
4. **Utility Menu> WorkPlane> Display Working Plane** (toggle on)



Notice how the working plane origin is immediately plotted in the Graphics Window. It is indicated by the WX and WY symbols, now coincident with the global origin X and Y symbols.

Change the WP type to polar, change the snap increment, and display the grid.

5. **Utility Menu> WorkPlane> WP Settings**

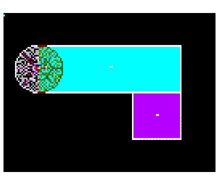


6. Click on Polar.

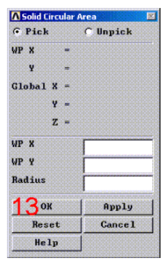
7. Click on Grid and Triad.

8. Enter .1 for snap increment.

9. OK to define settings and close the dialog box.



10. **Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle**



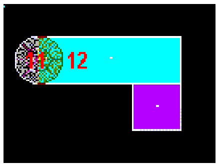
Read the prompt before picking.

11. Pick center point at:

    WP X = 0 (in Graphics Window shown below)

    WP Y = 0

12. Move mouse to radius of 1 and click left button to create circle.



13. OK to close picking menu.

14. Toolbar: **SAVE_DB**.

While positioning the cursor for picking, the dynamic WP X and Y values are displayed in the Solid Circular Area dialog.

As an alternative to picking, you can type the values along with the radius into the appropriate fields.

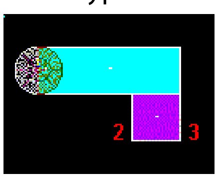
2.1.3.4. Step 4: Move working plane and create second circle.

To create the circle at the other end of the bracket in the same way, first move the working plane to the origin of the circle. The simplest way to do so without entering number offsets is to move the WP to an average keypoint location by picking the keypoints at the bottom corners of the lower-right rectangle.

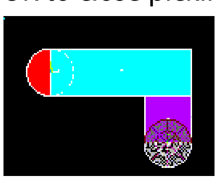
1. **Utility Menu> WorkPlane> Offset WP to> Keypoints**

2. Pick keypoint at lower *left* corner of rectangle.

3. Pick keypoint at lower *right* of rectangle.



4. OK to close picking menu.



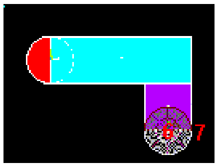
5. **Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle**

6. Pick center point at:

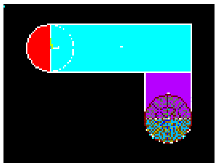
   WP X = 0

   WP Y = 0

7. Move mouse to radius of 1 and click left button to create circle.



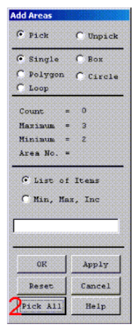
8. OK to close picking menu.



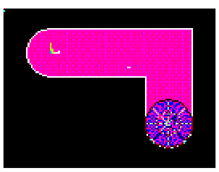
9. Toolbar: **SAVE_DB**.

2.1.3.5. Step 5: Add areas.

Now that the appropriate pieces of the model (rectangles and circles) are defined, add them together so the model becomes one continuous piece. Use the Boolean (p. 5) add operation for areas.

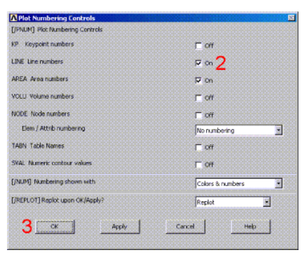


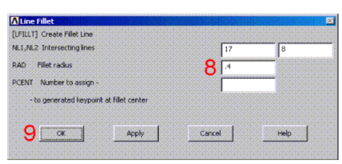
1. **Main Menu> Preprocessor> Modeling> Operate> Booleans> Add> Areas**
2. Pick All for all areas to be added.



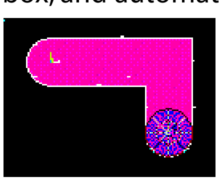
3. Toolbar: **SAVE_DB**.

2.1.3.6. Step 6: Create line fillet.

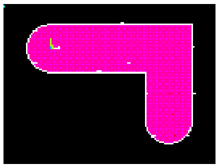




1. **Utility Menu> PlotCtrls> Numbering**
2. Activate line numbering.
3. OK to change controls, close the dialog box, and automatically replot.

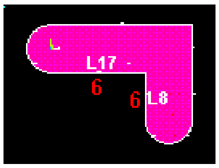


4. **Utility Menu> WorkPlane> Display Working Plane** (toggle off)



5. **Main Menu> Preprocessor> Modeling> Create> Lines> Line Fillet**

6. Pick lines 17 and 8.

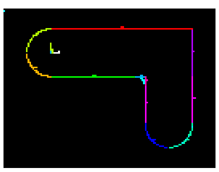


7. OK to finish picking lines (in picking menu).

8. Enter .4 as the radius.

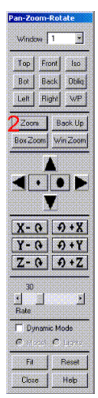
9. OK to create line fillet and close the dialog box.

10. **Utility Menu> Plot> Lines**



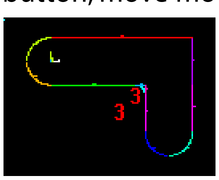
2.1.3.7. Step 7: Create fillet area.

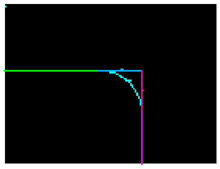
1. **Utility Menu> PlotCtrls> Pan, Zoom, Rotate**

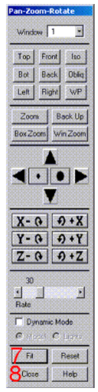


2. Click on Zoom button.

3. Move mouse to fillet region, click left button, move mouse out and click again.

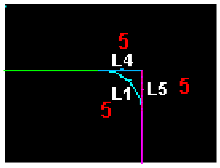






4. **Main Menu> Preprocessor> Modeling> Create> Areas> Arbitrary> By Lines**

5. Pick lines 4, 5, and 1.

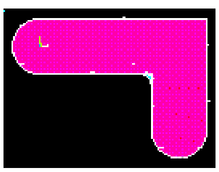


6. OK to create area and close the picking menu.

7. Click on Fit button.

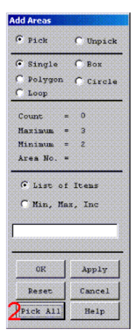
8. Close the Pan, Zoom, Rotate dialog box.

9. **Utility Menu> Plot> Areas**



10. Toolbar: **SAVE_DB**.

2.1.3.8. Step 8: Add areas together.



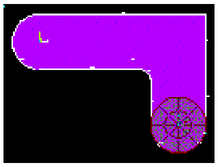
1. **Main Menu> Preprocessor> Modeling> Operate> Booleans> Add> Areas**

2. Pick All for all areas to be added.

3. Toolbar: **SAVE_DB**.

2.1.3.9. Step 9: Create first pin hole.

1. **Utility Menu> WorkPlane> Display Working Plane** (toggle on)



2. **Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle**

3. Pick center point at:

   WP X = 0 (in Graphics Window)

   WP Y = 0

4. Move mouse to radius of .4 (shown in the picking menu) and click left button to create circle.

5. OK to close picking menu.

2.1.3.10. Step 10: Move working plane and create second pin hole.

1. **Utility Menu> WorkPlane> Offset WP to> Global Origin**

2. **Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle**

3. Pick center point at:

   WP X = 0 (in Graphics Window)

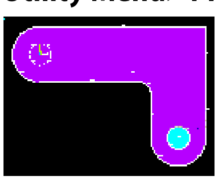
   WP Y = 0

4. Move mouse to radius of .4 (shown in the picking menu) and click left mouse button to create circle.

5. OK to close picking menu.

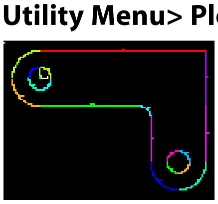
6. **Utility Menu> WorkPlane> Display Working Plane** (toggle off)

7. **Utility Menu> Plot> Replot**

   It appears that one of the pin-hole areas is missing; however, it is there (indicated by the presence of its lines). You cannot see it in the final display because the bracket area is drawn on top of it. An easy way to see all areas is to plot the lines instead.

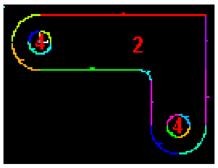
8. **Utility Menu> Plot> Lines**

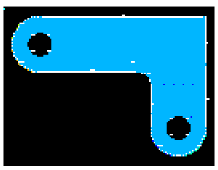
9. Toolbar: **SAVE_DB**.

2.1.3.11. Step 11: Subtract pin holes from bracket.

1. **Main Menu> Preprocessor> Modeling> Operate> Booleans> Subtract> Areas**
2. Pick bracket as base area from which to subtract.
3. Apply (in picking menu).
4. Pick both pin holes as areas to be subtracted.



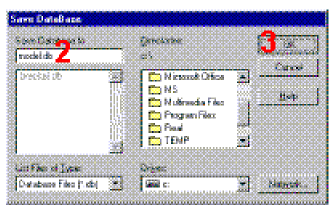
5. OK to subtract holes and close picking menu.



2.1.3.12. Step 12: Save the database as model.db.

Save the database to a file named to represent the model before meshing (in this case, *model.db*). Use that file to resume the analysis if you decide to go back and remesh.

1. **Utility Menu> File> Save As**
2. Enter *model.db* for the database file name.
3. OK to save and close dialog box.

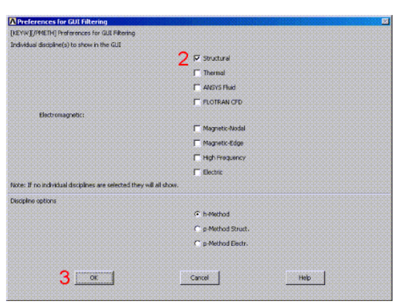


2.1.4. Define the Materials

2.1.4.1. Step 13: Set preferences.

In preparation for defining materials, set preferences (p. 7) so that only materials pertaining to a structural analysis are available.

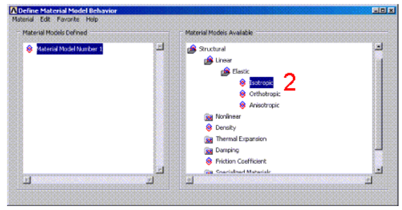
1. **Main Menu> Preferences**
2. Turn on structural filtering. The options may differ from what is shown here since they depend on the product you are using.
3. OK to apply filtering and close the dialog box.

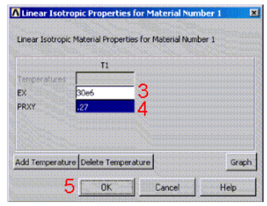


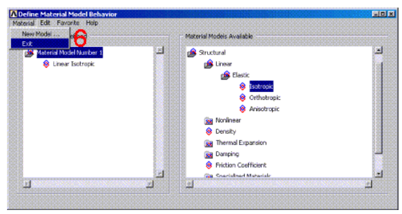
2.1.4.2. Step 14: Define material properties.

To define material properties (p. 6), there is only one material for the bracket, A36 Steel, with given values for Young's modulus of elasticity and Poisson's ratio.

1. **Main Menu> Preprocessor> Material Props> Material Models**
2. Double-click on Structural, Linear, Elastic, Isotropic.
3. Enter 30e6 for EX.
4. Enter .27 for PRXY.
5. OK to define material property set and close the dialog box.
6. **Material> Exit**







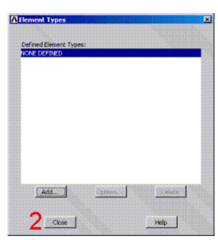
2.1.4.3. Step 15: Define element types and options.

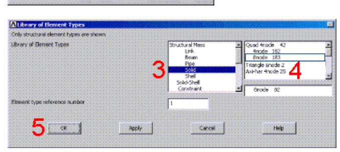
In any analysis, you select elements from a library of element types (p. 5) and define the appropriate ones for the analysis. In this case, only one element type is used: PLANE183, a 2-D, quadratic, structural, higher-order element (p. 6).

A higher-order element enables you to have a coarser mesh than with lower-order elements while still maintaining solution accuracy. Also, Mechanical APDL generates some triangle-shaped elements in the mesh that would otherwise be inaccurate when using used lower-order elements.

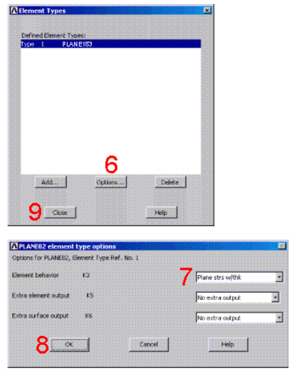
Specify plane stress with thickness as an option for PLANE183. (Thickness is defined as a real constant in Step 16: Define real constants. (p. 22).)

1. **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**
2. Add an element type.
3. Structural solid family of elements.
4. Select the 8-node quad (PLANE183).
5. OK to apply the element type and close the dialog box.





6. Options (p. 5) for PLANE183 are to be defined.
7. Select plane stress with thickness option for element behavior.
8. OK to specify options and close the options dialog box.
9. Close the element type dialog box.



2.1.4.4. Step 16: Define real constants.

Assuming plane stress with thickness, enter the thickness as a real constant (p. 8) for PLANE183:

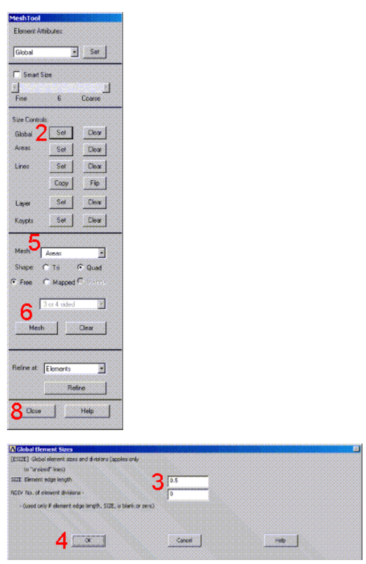
1. **Main Menu> Preprocessor> Real Constants> Add/Edit/Delete**
2. Add a real constant set.
3. OK for PLANE183.
4. Enter .5 for THK.
5. OK to define the real constant and close the dialog box.
6. Close the real constant dialog box.

2.1.5. Generate the Mesh

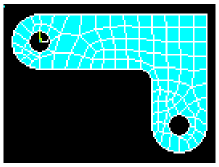
2.1.5.1. Step 17: Mesh the area.

You can mesh the model without specifying mesh-size controls. If you are unsure of how to determine mesh density, you can allow Mechanical APDL to apply a default mesh. For this model, however, you will specify a global element size to control overall mesh density.

1. **Main Menu> Preprocessor> Meshing> Mesh Tool**
2. Set Global Size control.
3. Type in 0.5.
4. OK.
5. Select Area Meshing.
6. Click on Mesh.
7. Pick All for the area to be meshed (in picking menu). Close any warning messages that appear.



8. Close the Mesh Tool.

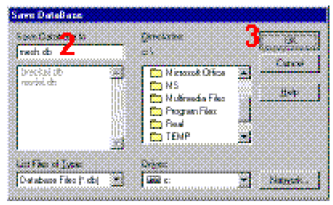


Your mesh may vary slightly from the mesh shown. You may see slightly different results during post-processing. For a discussion of results accuracy, see Planning Your Approach.

2.1.5.2. Step 18: Save the database as mesh.db.

As before, save the database to a named file, this time *mesh.db*.

1. **Utility Menu> File> Save as**
2. Enter *mesh.db* for database file name.
3. OK to save file and close dialog box.



2.1.6. Apply Loading

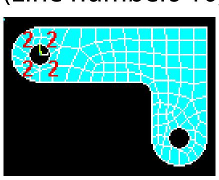
This step represents the beginning of the solution (p. 8) phase.

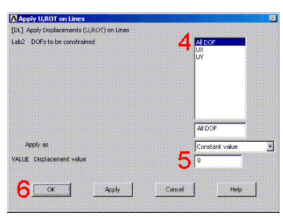
A new, static analysis is the default, so specifying an analysis type for this problem is unnecessary. Also, no analysis options (p. 5) exist for this problem.

2.1.6.1. Step 19: Apply displacement constraints.

You can apply displacement constraints directly to lines.

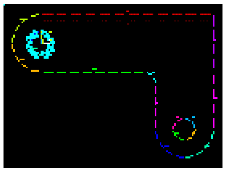
1. **Main Menu> Solution> Define Loads> Apply> Structural> Displacement> On Lines**
2. Pick the four lines around left-hand hole (Line numbers 10, 9, 11, 12).





3. OK (in picking menu).
4. Click on All DOF.
5. Enter 0 for zero displacement.
6. OK to apply constraints and close dialog box.

7. **Utility Menu> Plot Lines**



8. Toolbar: **SAVE_DB**.

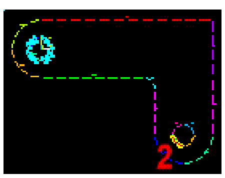
2.1.6.2. Step 20: Apply pressure load.

Apply the tapered pressure load to the bottom-right pin hole. (*Tapered* here means varying linearly.)

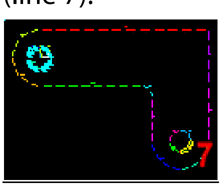
When a circle is created in Mechanical APDL, four lines define the perimeter; therefore, apply the pressure to *two* lines making up the lower half of the circle. Because the pressure tapers from a maximum value (500 psi) at the bottom of the circle to a minimum value (50 psi) at the sides, apply pressure in two separate steps, with reverse tapering values for each line.

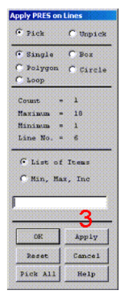
The Mechanical APDL convention for pressure loading is that a *positive* load value represents pressure *into* the surface (compressive).

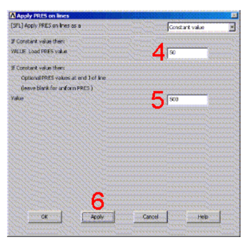
1. **Main Menu> Solution> Define Loads> Apply> Structural> Pressure> On Lines**
2. Pick line defining bottom *left* part of the circle (line 6).

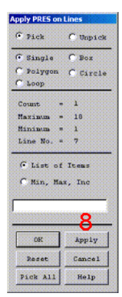


3. Apply.
4. Enter 50 for VALUE.
5. Enter 500 for optional value.
6. Apply.
7. Pick line defining bottom *right* part of circle (line 7).

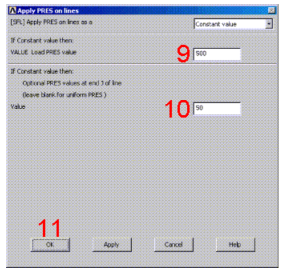


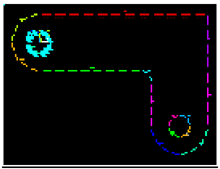






8. Apply.
9. Enter 500 for VALUE.
10. Enter 50 for optional value.
11. OK.



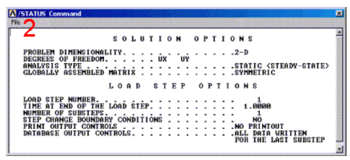


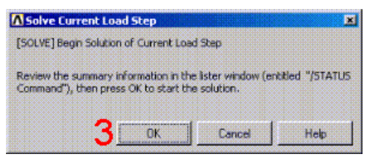
12. Toolbar: **SAVE_DB**.

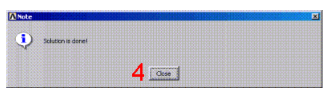
2.1.7. Obtain the Solution

2.1.7.1. Step 21: Solve.

1. **Main Menu> Solution> Solve> Current LS**
2. Review the information in the status window, then select **File> Close** (Windows), or **Close** (Linux), to close the window.
3. OK to begin the solution. Select Yes to any Verify messages that appear.
4. Close the information window when solution is done.







Mechanical APDL stores the results of this single-load-step problem in the database and in the results file, *Jobname*.RST (or *Jobname*.RTH for thermal, *Jobname*.RMG for magnetic). The database can contain only one set of results at any given time, so in a multiple-load-step or multiple-substep analysis, Mechanical APDL stores only the *final* solution in the database.

Mechanical APDL stores *all* solutions in the results file.

2.1.8. Review the Results

This step represents the beginning of the postprocessing (p. 7) phase.

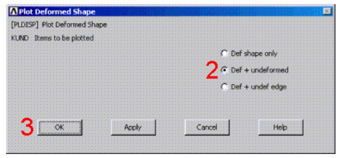
The results you see may vary slightly from what is shown due to variations in the mesh.

2.1.8.1. Step 22: Enter the general postprocessor and read in the results.

1. **Main Menu> General Postproc> Read Results> First Set**

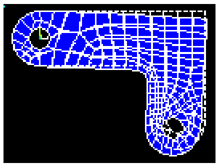
2.1.8.2. Step 23: Plot the deformed shape.

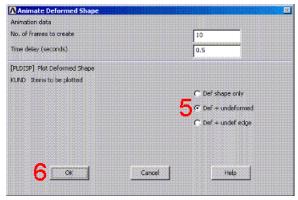
1. **Main Menu> General Postproc> Plot Results> Deformed Shape**



2. Select Def + undeformed.

3. OK.



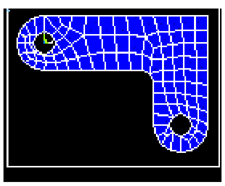


You can also generate an animated version of the deformed shape:

4. **Utility Menu> Plot Ctrls> Animate> Deformed Shape**

5. Select Def + undeformed.

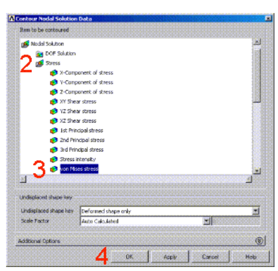
6. OK.



7. Make selections in the Animation Controller (not shown), if necessary, then select Close.

2.1.8.3. Step 24: Plot the von Mises equivalent stress.

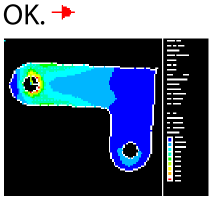
1. **Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu**

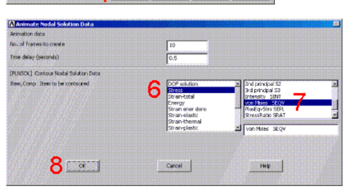


2. Select Stress to be contoured.

3. Scroll down and select von Mises (SEQV).

4. OK.





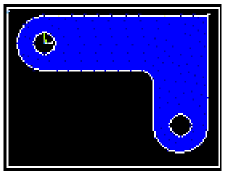
You can also produce an animated version of these results:

5. **Utility Menu> Plot Ctrls> Animate> Deformed Results**

6. Select Stress to be contoured.

7. Scroll down and select von Mises (SEQV).

8. OK.



9. Make selections in the Animation Controller (not shown), if necessary, then select Close.

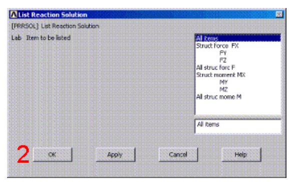
2.1.8.4. Step 25: List reaction solution.

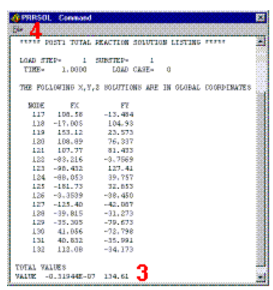
1. **Main Menu> General Postproc> List Results> Reaction Solu**

2. OK to list all items and close the dialog box.

3. Scroll down and find the total vertical force, FY.

4. **File> Close** (Windows), or **Close** (Linux), to close the window.





The value of 134.61 is comparable to the total pin load force.

The values shown are representative and may vary from the values that you obtain.

Many other options are available for reviewing results in the general postprocessor. You will see some other options in other tutorials.

You have finished the analysis. Exit the program in the next step.

2.1.8.5. Step 26: Exit the Mechanical APDL program.

Exit the Mechanical APDL program. Upon exiting, you have the following options:

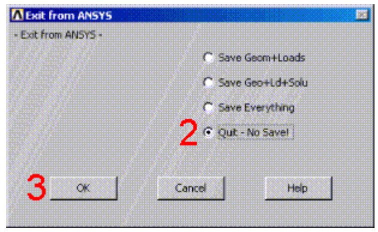
- Save the geometry and loads portions of the database (default).

- Save geometry, loads, and solution data (one set of results only).

- Save geometry, loads, solution data, and postprocessing data (save everything).
- Save nothing.

You can save nothing here, but use one of the other save options if you want to keep your data files.

1. Toolbar: **Quit**.
2. Select Quit - No Save!
3. OK.

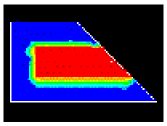


Although you have exited the Mechanical APDL program, you can still view animations via the ANIMATE program (p. 97). The program runs only on Windows computers and is useful for:

- Viewing Mechanical APDL animations on a Windows computer regardless of whether the files were created in Windows (AVI files) or Linux (ANIM files).
- Converting ANIM files to AVI files.
- Sending animations via email or sharing via the Web.

Chapter 3: Thermal Tutorial

Solidification of a Casting



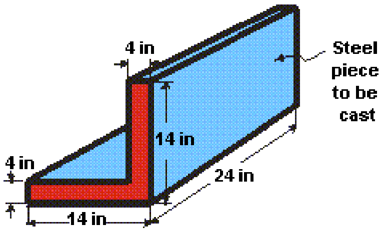
3.1. Solidification of a Casting

3.1.1. Problem Specification

Applicable Products: (p. 5)	ANSYS Multiphysics, ANSYS Mechanical
Level of Difficulty: (p. 6)	Moderate
Interactive Time Required: (p. 6)	60 to 90 minutes
Discipline: (p. 5)	Thermal
Analysis Type: (p. 5)	Nonlinear transient
Element Types Used: (p. 5)	PLANE55
Features Demonstrated: (p. 5)	Conduction, convection, phase change, selecting, solution control, time-history postprocessing, use of a "get function"
Help Resources: (p. 6)	Transient Thermal Analysis and PLANE55

3.1.2. Problem Description

This is a transient heat-transfer analysis of a casting process. The solidification process occurs over a duration of four hours. The casting is made in an L-shaped sand mold with four-inch-thick walls. Convection occurs between the sand mold and the ambient air.



Objective: Track the temperature distribution in the steel casting and mold during solidification.

3.1.2.1. Given

Material Properties for Sand	
Conductivity (KXX)	0.025 Btu/(hr-in-^{o}F)
Density (DENS)	0.054 lb/in^{3}
Specific heat (C)	0.28 Btu/(lb-^{o}F)

Conductivity (KXX) for Steel	
at 0^{o}F	1.44 Btu/(hr-in-^{o}F)
at 2643^{o}F	1.54
at 2750^{o}F	1.22
at 2875^{o}F	1.22

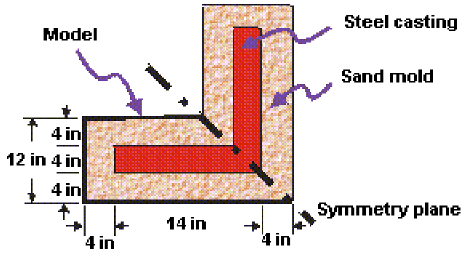
Enthalpy (ENTH) for Steel	
at 0^{o}F	0.0 Btu/in^{3}
at 2643^{o}F	128.1
at 2750^{o}F	163.8
at 2875^{o}F	174.2

Initial Conditions	
Temperature of steel	2875 ^{o}F
Temperature of sand	80 ^{o}F

Convection Properties	
Film coefficient	0.014 Btu/(hr-in^{2}-^{o}F)
Ambient temperature	80 ^{o}F

3.1.2.2. Approach and Assumptions

You will perform a 2-D analysis of a one unit thick slice. Half-symmetry is used to reduce the size of the model. The lower half is the portion to be modeled.



The mold material (sand) has constant material properties. The casting (steel) has temperature-dependent thermal conductivity and enthalpy; both are input in a table of values versus temperature. The enthalpy property table captures the latent heat capacity of the metal as it solidifies. Radiation effects are ignored.

Solution control is used to establish several nonlinear options, including automatic time stepping. Automatic time stepping determines the proper time step increments needed to converge the phase change nonlinearity. This means that smaller time step sizes will be used during the transition from molten metal to solid state.

3.1.2.3. Summary of Steps

Use the information in the problem description and the steps below as a guideline in solving the problem on your own. Or, use the detailed interactive step-by-step solution by selecting the link for step 1.

Prepare for a Thermal Analysis

1. Set preferences. (p. 32)

Input Geometry

2. Read in the geometry of the casting. (p. 32)

Define Materials

3. Define material properties. (p. 32)

4. Plot material properties vs. temperature. (p. 34)

5. Define element type. (p. 34)

Generate Mesh

6. Mesh the model. (p. 35)

Apply Loads

7. Apply convection loads on the exposed boundary lines. (p. 37)

Obtain Solution

8. Define analysis type. (p. 37)

9. Examine solution control. (p. 38)

10. Specify initial conditions for the transient. (p. 38)

Review Results

3.1.3. Prepare for a Thermal Analysis

3.1.3.1. Step 1: Set preferences.

To Set Preferences (p. 7):

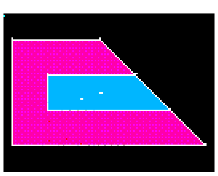
1. **Main Menu> Preferences**
2. (check) "Individual discipline(s) to show in the GUI" = Thermal
3. [OK]

3.1.4. Input Geometry

3.1.4.1. Step 2: Read in the geometry of the casting.

Reading in the file containing the casting model.

1. **Utility Menu> File> Read Input from ...**
2. File name: `casting.inp` (Click the link to download the file in .zip format.)
3. [OK]



3.1.5. Define Materials

3.1.5.1. Step 3: Define material properties.

Define the sand mold material properties (p. 6) as material number 1. These are *not* functions of temperature.

1. **Main Menu> Preprocessor> Material Props> Material Models**
2. (double-click) “Thermal”, then “Conductivity”, then “Isotropic”
3. “KXX” = 0.025
4. [OK]
5. (double-click) “Specific Heat”
6. “C” = 0.28
7. [OK]
8. (double-click) “Density”
9. “DENS” = 0.54
10. [OK]

    The metal casting is defined as material number 2. These properties change significantly as the metal cools down from the liquid phase to the solid phase. They are therefore entered in a table of properties versus temperature.

    First define the temperature-dependent thermal conductivity.

11. **Material> New Model**
12. “Define Material ID” = 2
13. [OK]
14. (double-click) “Isotropic”
15. [Add Temperature] three times to create fields for the four temperatures.
16. “T1” = 0
17. “T2” = 2643
18. “T3” = 2750
19. “T4” = 2875
20. “KXX” at “T1” = 1.44
21. “KXX” at “T2” = 1.54
22. “KXX” at “T3” = 1.22
23. “KXX” at “T4” = 1.22

    Copy the four temperatures so that you can paste them into the Enthalpy dialog box.

24. Select the temperatures by holding the left mouse button and dragging across the temperature row so that the row is highlighted.

25. **Ctrl**+**C** to copy the temperatures.

26. [OK]

    Define the temperature dependent enthalpy.

27. (double-click) "Enthalpy"

28. [Add Temperature] three times to create fields for the four temperatures.

29. Paste the temperatures into the dialog box by highlighting the T1 temperature field, and pressing **Ctrl**+**V**.

30. "ENTH" at "T1" = 0

31. "ENTH" at "T2" = 128.1

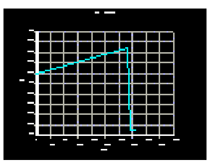
32. "ENTH" at "T3" = 163.8

33. "ENTH" at "T4" = 174.2

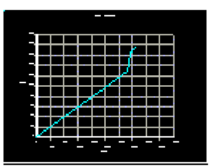
34. [OK]

3.1.5.2. Step 4: Plot material properties vs. temperature.

1. (double-click) "Thermal conduct. (iso)" under **Material Model Number 2**.
2. [Graph]
3. [OK]



4. (double-click) "Enthalpy" under the right or left window.
5. [Graph]
6. [OK]



7. **Material> Exit**
8. Toolbar: **SAVE_DB**

3.1.5.3. Step 5: Define element type.

Define the element type (p. 5) as PLANE55.

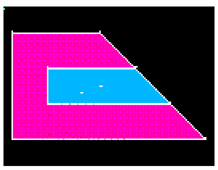
1. **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**

2. [Add ...]
3. “Thermal Solid” (left column)
4. “Quad 4node 55” (right column)
5. [OK]
6. [Close]
7. Toolbar: **SAVE_DB**

3.1.6. Generate Mesh

3.1.6.1. Step 6: Mesh the model.

1. **Utility Menu> Plot> Areas**

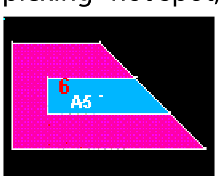
   

   Specify a SmartSize of 4. This will allow a slightly finer mesh than the default.

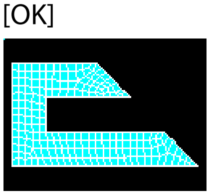
2. **Main Menu> Preprocessor> Meshing> MeshTool**
3. (check) “Smart Size”
4. (slide) “Fine Course” = 4
5. [Mesh]

   Mesh the mold area first. Note that the material attribute reference number defaults to 1 and there is no need to set attributes before meshing the area.

6. Pick the mold area A5. (Hint: Place the mouse cursor on top of the A5 label when you pick—this is the picking "hot spot," based on the centroid of the area.)

7. [OK]

   Before meshing the casting area, set the material attribute to that of steel (material 2).

8. (drop down in MeshTool) “Element Attributes” = Global, then [Set]

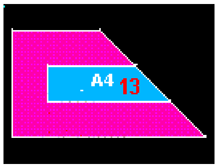
9. (drop down) “Material number” = 2

10. [OK]

11. **Utility Menu> Plot> Areas**

12. [Mesh] in MeshTool

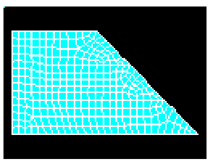
13. Pick area A4



14. [OK]

15. [Close] in MeshTool

16. **Utility Menu> Plot> Elements**



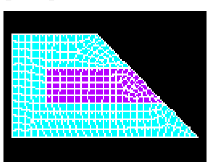
Note: The mesh you obtain may vary slightly from the mesh shown here. As a result of this, you may see slightly different results during postprocessing. For a discussion of results accuracy, see Planning Your Approach in the *Modeling and Meshing Guide*.

To verify that the elements have the right materials, plot them with different colors for different materials.

17. **Utility Menu> PlotCtrls> Numbering**

18. (drop down) “Elem / Attrib numbering” = Material numbers

19. [OK]

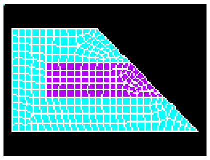


Note: the elements of material 1 form the sand mold. The elements of material 2 form the steel casting. You can also plot the elements showing materials in different colors without displaying the associated material numbers.

20. **Utility Menu> PlotCtrls> Numbering**

21. (drop down) “Numbering shown with” = Colors only

22. [OK]



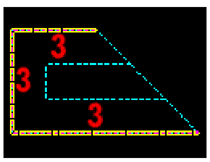
23. Toolbar: **SAVE_DB**

3.1.7. Apply Loads

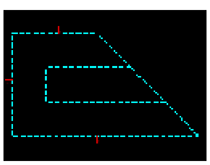
3.1.7.1. Step 7: Apply convection loads on the exposed boundary lines.

Apply the convection to the lines of the solid model. Loads applied to solid modeling entities are automatically transferred to the finite element model during solution.

1. **Utility Menu> Plot> Lines**
2. **Main Menu> Preprocessor> Loads> Define Loads> Apply> Thermal> Convection> On Lines**
3. Pick the three lines that are exposed to ambient air.



4. [OK]
5. “Film coefficient” = 0.014
6. “Bulk temperature” = 80
7. [OK]



8. Toolbar: **SAVE_DB**

3.1.8. Obtain Solution

3.1.8.1. Step 8: Define analysis type.

1. **Main Menu> Solution> Analysis Type> New Analysis**
2. (check) “Type of analysis” = Transient
3. [OK]
4. (check) “Solution method” = Full
5. [OK]

3.1.8.2. Step 9: Examine solution control.

The Approach and Assumptions section of this tutorial mentioned that solution control is used to establish several nonlinear options. In this step, you will be directed to the online help for solution control so you can examine the details of this feature.

You will access this help topic by clicking on the Help button from within the Nonlinear Solution Control dialog box.

1. **Main Menu> Solution> Load Step Opts> Solution Ctrl**

   Note that solution control is on by default.

   Before clicking on the Help button in the next step, you should be aware that the help information may appear in the same window as this tutorial, *replacing* the contents of the tutorial. If this is the case, after reading the help information, you will need to click on the Back button to return to this tutorial. If the help information appears in a *separate*window from the tutorial, you can minimize or close the help window after you read the help information.

2. [Help] then read the details on Solution Control.

3. If the help information replaced the tutorial, click on the Back button to return to the tutorial. If the help information appears in a separate window, you can close or minimize that window.

4. [Cancel] to remove the dialog box.

3.1.8.3. Step 10: Specify initial conditions for the transient.

The mold is initially at an ambient temperature of 80°F and the molten metal is at 2875°F. Use select entities to obtain the correct set of nodes on which to apply the initial temperatures. First select the casting area, then select the nodes within that area and apply the initial molten temperature to those nodes. Next, invert the selected set of nodes and apply the ambient temperature to the mold nodes.

Start by plotting areas.

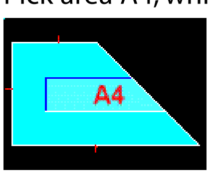
1. **Utility Menu> Plot> Areas**

2. **Utility Menu> Select> Entities**

3. (first drop down) “Areas”

4. [OK]

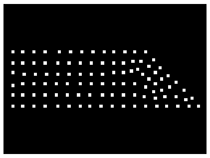
5. Pick area A4, which is the casting.

6. [OK]

7. **Utility Menu> Select> Everything Below> Selected Areas**

8. **Utility Menu> Plot> Nodes**



9. **Main Menu> Solution> Define Loads> Apply> Initial Condit'n> Define**

10. [Pick All] to use *selected* nodes.

11. (drop down) "DOF to be specified" = TEMP

12. "Initial value of DOF" = 2875

13. [OK]

14. **Utility Menu> Select> Entities**

15. (first drop down) "Nodes"

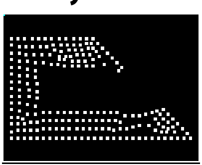
16. (second drop down) "Attached to"

17. (check) "Areas, all"

18. [Invert] This is an action command; the selected set of nodes is immediately inverted.

19. [Cancel] to close the dialog box.

20. **Utility Menu> Plot> Nodes**



21. **Main Menu> Solution> Define Loads> Apply> Initial Condit'n> Define**

22. [Pick All] to use all selected nodes.

23. "Initial value of DOF" = 80

24. [OK]

Always select Everything *again after you have finished selecting the nodes.*

25. **Utility Menu> Select> Everything**

26. Toolbar: **SAVE_DB**

3.1.8.4. Step 11: Set time, time step size, and related parameters.

Stepped boundary conditions simulate the sudden contact of molten metal at 2875 ^{O}F with the mold at ambient temperature. The program selects automatic time stepping, enabling the time-step size to be modified depending on the severity of nonlinearities in the system. (For example, the program uses smaller time steps during the phase change.) The maximum and minimum time-step sizes represent the limits for this automated procedure.

1. **Main Menu> Solution> Load Step Opts> Time/Frequenc> Time-Time Step**
2. "Time at end of load step" = 4

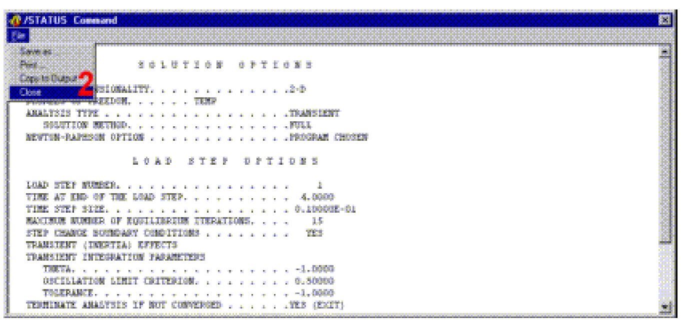
   This setting represents 4 *hours*.
3. "Time step size" = 0.01
4. (check) "Stepped or ramped b. c." = Stepped
5. "Minimum time step size" = 0.001
6. "Maximum time step size" = 0.25
7. [OK]

3.1.8.5. Step 12: Set output controls.

1. **Main Menu> Solution> Load Step Opts> Output Ctrls> DB/Results File**
2. (check) "File write frequency" = Every substep
3. [OK]
4. Toolbar: **SAVE_DB**

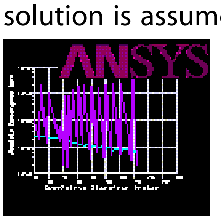
3.1.8.6. Step 13: Solve.

1. **Main Menu> Solution> Solve> Current LS**
2. Review the information in the status window, then select **File> Close** (Windows), or **Close** (Linux), to close the window.



3. [OK] to initiate the solution.
4. [Close] when the solution is done.

While Mechanical APDL solves the analysis, the Graphical Solution Tracking (GST) monitor plots the "Absolute Convergence Norm" as a function of the "Cumulative Iteration Number." Notice that the solution is assumed to have converged for values less than or equal to the convergence criteria.

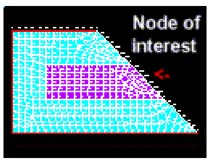


3.1.9. Review Results

3.1.9.1. Step 14: Enter the time-history postprocessor and define variables.

Use the time-history postprocessor to look at the variation of temperature with respect to time at one point on the casting (on the symmetry plane).

1. **Utility Menu> PlotCtrls> Numbering**
2. (check) "Node numbers" = On
3. (drop down) "Numbers shown with" = Colors & numbers
4. [OK]
5. **Utility Menu> Plot> Elements**

   The node at the center of the casting on the symmetry plane is the node of interest. Use a "get function" to define a variable equal to the value of the node number at the location of interest (16,6,0). By using a variable to identify the node at the center point, the analysis will be more flexible in that the center node will always be used even if the mesh, and therefore node numbers, change.

6. **Utility Menu> Parameters> Scalar Parameters**
7. "Selection"=cntr_pt = node (16,6,0)
8. [Accept}

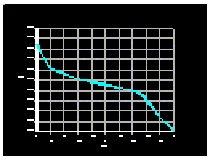
   Note the center point node number. This number can vary due to differences in the mesh.

9. [Close]
10. **Main Menu> TimeHist Postproc**
11. [+] to add data.
12. (double-click) "Nodal Solution", then "DOF Solution", then "Temperature"
13. "Variable Name" = center
14. [OK]
15. Type cntr_pt in the picker, then press Enter.
16. [OK] in the picker.
17. **File> Close**

3.1.9.2. Step 15: Plot temperature vs. time.

1. **Main Menu> TimeHist Postpro> Graph Variables**

2. “1st variable to graph” = 2
3. [OK] to plot the results at cntr_point as a function of time.



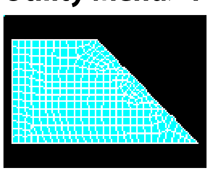
Notice that the solidification region is approximately 2643°F - 2750°F. Your graph may vary slightly.

3.1.9.3. Step 16: Set up to animate the results.

Animate the solidification of the molten metal. To better visualize the solidification process, specify three contours. One will represent the molten metal (T greater than 2750 °F), one will represent the solidified metal (T less than 2643 °F), and the third will represent everything in between.

To generate an animation, enter the General Postprocessor and read the first set of results.

1. **Main Menu> General Postproc> Read Results> First Set**
2. **Utility Menu> PlotCtrls> Numbering**
3. (check) “Node numbers” = Off
4. (drop down) “Elem / Attrib numbering” = No numbering
5. (drop down) “Replot upon OK/Apply?” = Do not replot
6. [OK]
7. **Utility Menu> Plot> Elements**



8. **Utility Menu> PlotCtrls> Style> Contours> Non_uniform Contours**

   As indicated in the brackets at the upper left corner of the dialog box, the command to specify non_uniform contours is **/CVAL**.

9. “V1” = 2643
10. “V2” = 2750
11. “V3” = 3000

    The three values represent the upper bounds of the first, second, and third contours, respectively.

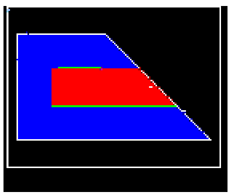
12. [OK]

3.1.9.4. Step 17: Animate the results.

Linux only: To capture the animation sequence in terminal segment memory, reduce the size of the Graphics Window or you will run out of terminal memory. For this analysis, the ratio of the original window size to the reduced window size should be about 2:1.

Procedure on all systems:

1. **Utility Menu> PlotCtrls> Animate> Over Time**
2. "Number of animation frames" = 30
3. (check) "Auto contour scaling" = Off
4. [OK]

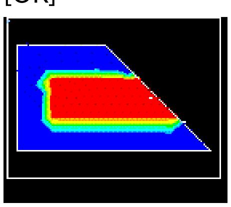


During the animation, notice the three separate colors - red for temperatures greater than 2750 ^{o}F (molten steel), green for temperatures between 2643 ^{o}F and 2750 ^{o}F (the "mushy" phase-change region), and blue for temperatures below 2643 ^{o}F (the solidified steel and the sand mold). As you would expect, the last region to solidify is the material at the center of the casting. (Remember that a symmetry model was used.)

5. Make selections in the Animation Controller (not shown), if necessary, then [Close].

To visualize the temperature distribution throughout the model over the 4 hour span, animate the temperature distribution with the default contour settings. To change the contour settings back to their default value, enter **/CVAL** in the Input Window. (Alternatively, you can go back to the Non_Uniform Contours window and set all values to zero.)

6. Type **/CVAL**, then press Enter.
7. **Utility Menu> PlotCtrls> Animate> Over Time**
8. [OK]



9. Make selections in the Animation Controller (not shown), if necessary, then [Close].

3.1.9.5. Step 18: Exit the program.

1. Toolbar: **QUIT**
2. (check) "Quit - No Save!"

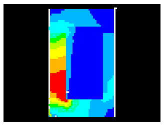
3. [OK]

Although you have exited the Mechanical APDL program, you can still view animations via the ANIMATE program (p. 97). The program runs only on Windows computers and is useful for:

- Viewing Mechanical APDL animations on a Windows computer regardless of whether the files were created in Windows (AVI files) or Linux (ANIM files).
- Converting ANIM files to AVI files.
- Sending animations via email or sharing via the Web.

Chapter 4: Electromagnetics Tutorial

Magnetic Analysis of a Solenoid Actuator



4.1. Magnetic Analysis of a Solenoid Actuator

4.1.1. Problem Specification

Applicable Products: (p. 5)	ANSYS Multiphysics, ANSYS Emag
Level of Difficulty: (p. 6)	Easy
Interactive Time Required: (p. 6)	60 to 75 minutes
Discipline: (p. 5)	Electromagnetics
Analysis Type: (p. 5)	Linear static
Element Types Used: (p. 5)	PLANE13
Features Demonstrated: (p. 5)	Axisymmetry, vector plots, element table operations, path operations
Help Resources: (p. 6)	Two-dimensional Static Magnetic Analysis, PLANE13.

4.1.2. Problem Description

A solenoid actuator is analyzed as a 2-D axisymmetric model.

4.1.2.1. Given

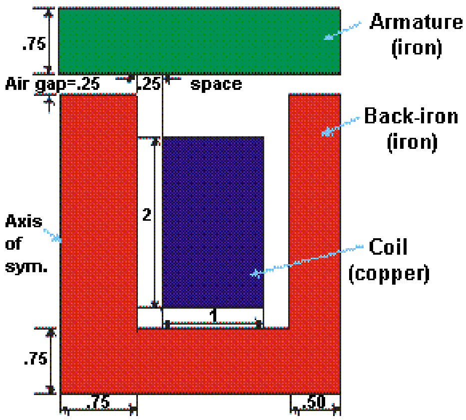
The dimensions of the solenoid actuator are in centimeters.

The *armature* is the moving component of the actuator.

The *back-iron* is the stationary iron component of the actuator that completes the magnetic circuit around the coil.

The stranded, wound *coil* of 650 windings with 1 amp/turn supplies the predefined current. The current per winding is 1 amp.

The *air-gap* is the thin rectangular region of air between the armature and the pole faces of the back-iron.



Objective: For the given current, determine the force on the armature.

4.1.2.2. Approach and Assumptions

The magnetic flux produced by the coil current is assumed to be so small that no saturation of the iron occurs, allowing a single-iteration linear analysis. The flux leakage out of the iron at the perimeter of the model is assumed to be negligible. This assumption is made simple to keep the model small. The model would normally be created with a layer of air surrounding the iron equal to or greater than the maximum radius of the iron.

The air gap is modeled so that a quadrilateral mesh is possible. A quadrilateral mesh allows for a uniform thickness of the air elements adjacent to the armature where the virtual work force calculation is performed. This is desirable for an accurate force calculation.

The program requires current to be input in the form of current density (current over the area of the coil).

The assumption of no leakage at the perimeter of the model means that the flux is acting parallel to this surface. The assumption is enforced by the flux parallel boundary condition placed around the model, used for models where the flux is contained in an iron circuit.

Forces for the virtual work calculation are stored in an element table and then summed. The force is also calculated by the Maxwell Stress Tensor method and the two values are found to be relatively close.

4.1.2.3. Summary of Steps

Use the information in the problem description and the steps below as a guideline in solving the problem on your own. Or, use the detailed interactive step-by-step solution by choosing the link for step 1.

Input Geometry

1. Read in geometry input file. (p. 48)

Define Materials

2. Set preferences. (p. 48)

3. Specify material properties. (p. 48)

Generate Mesh

4. Define element type and options. (p. 49)

5. Assign material property attributes. (p. 50)

6. Specify meshing-size controls on air gap. (p. 51)

7. Mesh the model using the MeshTool. (p. 51)

8. Scale model to meters for solution. (p. 52)

Apply Loads

9. Define the armature as a component. (p. 52)

10. Apply force boundary conditions to armature. (p. 53)

11. Apply the current density. (p. 53)

12. Obtain a flux parallel field solution. (p. 54)

Obtain Solution

13. Solve. (p. 54)

Review Results

14. Plot the flux lines in the model. (p. 54)

15. Summarize magnetic forces. (p. 55)

16. Plot the flux density as vectors. (p. 55)

17. Plot the magnitude of the flux density. (p. 55)

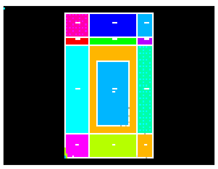
18. Exit the program. (p. 56)

4.1.3. Input Geometry

4.1.3.1. Step 1: Read in geometry input file.

Read in the file that includes the model geometry.

1. **Utility Menu> File> Read Input from ...**
2. File name: `solenoid.inp` (Click the link to download the file in .zip format.)
3. [OK]



4.1.4. Define Materials

4.1.4.1. Step 2: Set preferences.

Set preferences (p. 7) to filter quantities that pertain to this discipline only.

1. **Main Menu> Preferences**
2. (check) “Magnetic-Nodal”
3. [OK]

4.1.4.2. Step 3: Specify material properties.

Specify the material properties (p. 6) for the magnetic permeability of air, back-iron, coil, and armature. For simplicity, all material properties are assumed to be linear. (Typically, iron is input as a nonlinear B-H curve.)

Material 1 is used for the air elements, Material 2 for the back-iron elements, Material 3 for the coil elements, and Material 4 for the armature elements.

1. **Main Menu> Preprocessor> Material Props> Material Models**
2. (double-click) “Electromagnetics”, then "Relative Permeability", then “Constant”
3. “MURX” = 1
4. [OK]
5. **Edit> Copy**
6. [OK] to copy Material Model Number 1 to become Material Model Number 2.
7. (double-click) “Material Model Number 2”, then “Permeability (Constant)”
8. “MURX” = 1000

9. [OK]
10. **Edit> Copy**
11. “from Material Number” = 1
12. “to Material Number” = 3
13. [OK]
14. **Edit> Copy**
15. “from Material Number” = 2
16. “to Material Number” = 4
17. [OK]
18. (double-click) “Material Model Number 4”, then “Permeability (Constant)”
19. “MURX” = 2000
20. [OK]
21. **Material> Exit**
22. **Utility Menu> List> Properties> All Materials**
23. Review the list of materials, then:

    File> Close (Windows),

    or

    Close (Linux) to close the window.

4.1.5. Generate Mesh

4.1.5.1. Step 4: Define element types and options.

Define element types (p. 5) and specify options (p. 5) associated with those element types.

1. **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**
2. [Add...]
3. “Magnetic Vector” (left column)
4. “Vect Quad 4nod13 (PLANE13” (right column)
5. [OK]
6. [Options...]
7. (drop down) “Element behavior” = Axisymmetric

8. [OK]

9. [Close]

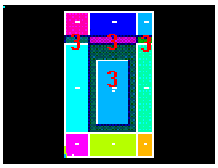
4.1.5.2. Step 5: Assign material properties.

Assign material properties (p. 6) to air gaps, iron, coil, and armature areas.

1. **Main Menu> Preprocessor> Meshing> MeshTool**

2. (drop down) "Element Attributes" = Areas; then [Set]

3. Pick four areas of air gaps, A13, A14, A17, and A18 (the picking "hot spot" is at the area number label).

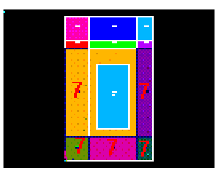


4. [OK]

5. (drop down) "Material number" = 1

6. [Apply]

7. Pick the five back-iron areas, A7, A8, A9, A11, A12.

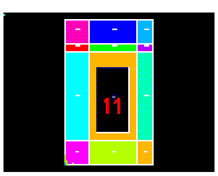


8. [OK]

9. (drop down) "Material number" = 2

10. [Apply]

11. Pick coil area, A4.

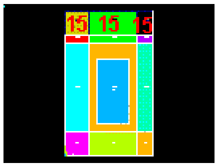


12. [OK]

13. (drop down) "Material number" = 3

14. [Apply}

15. Pick armature area, A10, A15, A16.



16. [OK]

17. (drop down) “Material number” = 4

18. [OK]

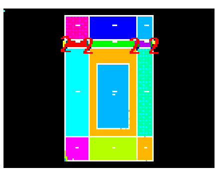
19. Toolbar: **SAVE_DB**

4.1.5.3. Step 6: Specify meshing-size controls on air gap.

Adjust meshing size controls to obtain two element divisions through the air gap.

1. **Main Menu> Preprocessor> Meshing> Size Cntrls> ManualSize> Lines> Picked Lines**

2. Pick four vertical lines through air gap.



3. [OK]

4. “No. of element divisions” = 2

5. [OK]

4.1.5.4. Step 7: Mesh the model using the MeshTool.

1. “Size Control Global” = [Set]

2. “Element edge length” = 0.25

3. [OK]

4. (drop down) “Mesh” = Areas

5. [Mesh]

6. [Pick All]

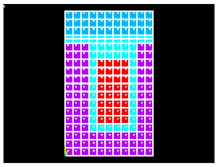
7. [Close]

   For a simplified model, select a coarse mesh. For production, however, use a finer mesh, especially in the air-gap region.

8. **Utility Menu> PlotCtrls> Numbering**

9. (drop down) “Elem / attrib numbering” = Material numbers

10. [OK]



Your mesh may vary slightly from the mesh shown here. As a result, you may see slightly different results during postprocessing (p. 7). For a discussion of results accuracy, see Planning Your Approach in the *Modeling and Meshing Guide*.

4.1.5.5. Step 8: Scale model to meters for solution.

For a magnetic analysis, a consistent set of units is necessary. This analysis uses MKS units; therefore, scale the model from centimeters to meters.

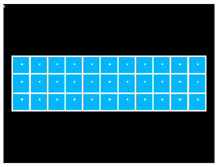
1. **Main Menu> Preprocessor> Modeling> Operate> Scale> Areas**
2. [Pick All]
3. "RX,RY,RZ Scale Factors" = 0.01, 0.01, 1
4. (drop down) "Existing areas will be" = Moved
5. [OK]
6. Toolbar: **SAVE_DB**

4.1.6. Apply Loads

4.1.6.1. Step 9: Define the armature as a component.

Define the armature as a component by selecting its elements.

1. **Utility Menu> Select> Entities**
2. (first drop down) "Elements"
3. (second drop down) "By Attributes"
4. "Min, Max, Inc" = 4
5. [OK]
6. **Utility Menu> Plot> Elements**



7. **Utility Menu> Select> Comp/Assembly> Create Component**

8. "Component name" = ARM

9. (drop down) "Component is made of" = Elements

10. [OK]

4.1.6.2. Step 10: Apply force boundary conditions to armature.

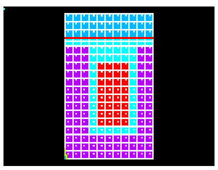
1. **Main Menu> Preprocessor> Loads> Define Loads> Apply> Magnetic> Flag> Comp. Force/Torq**

2. (highlight) "Component name" = ARM

3. [OK]

4. **Utility Menu> Select> Everything**

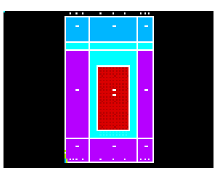
5. **Utility Menu> Plot> Elements**



4.1.6.3. Step 11: Apply the current density.

The current density is defined as the number of coil windings times the current, divided by the coil area. This equals (650)(1)/2, or 325. To account for scaling from centimeters to meters, the calculated value needs to be divided by .01**2.

1. **Utility Menu> Plot> Areas**



2. **Main Menu> Preprocessor> Loads> Define Loads> Apply> Magnetic> Excitation> Curr Density> On Areas**

3. Pick the coil area, which is the area in the center.

4. [OK]

5. "Curr density value" = 325/.01**2

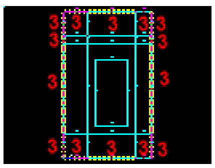
6. [OK]

   Close any warning messages that appear.

4.1.6.4. Step 12: Obtain a flux parallel field solution.

Apply a perimeter boundary condition to obtain a “flux parallel” field solution. This boundary condition assumes that the flux does not leak out of the iron at the perimeter of the model. Of course at the centerline this is true due to axisymmetry.

1. **Utility Menu> Plot> Lines**
2. **Main Menu> Preprocessor> Loads> Define Loads> Apply> Magnetic> Boundary> Vector Poten> Flux Par'l> On Lines**
3. Pick all lines around perimeter of model (14 lines).



4. [OK]
5. Toolbar: **SAVE_DB**

4.1.7. Obtain Solution

4.1.7.1. Step 13: Solve.

1. **Main Menu> Solution> Solve> Electromagnet> Static Analysis> Opt & Solve**
2. [OK] to initiate the solution.
3. [Close] the information window when solution is done.

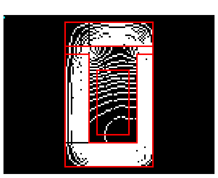
4.1.8. Review Results

4.1.8.1. Step 14: Plot the flux lines in the model.

Note that a certain amount of undesirable flux leakage occurs out of the back-iron.

1. **Main Menu> General Postproc> Plot Results> Contour Plot> 2D Flux Lines**
2. [OK]

   Close any Notes or Warnings.



Your results may vary slightly from what is shown here due to variations in the mesh.

4.1.8.2. Step 15: Summarize magnetic forces.

1. **Main Menu> General Postproc> Elec & Mag Calc> Component Based> Force**
2. (highlight) “Component name(s)” = ARM
3. [OK]
4. Review the information, then choose:

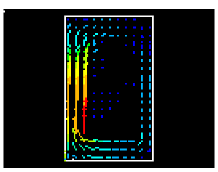
   File> Close (Windows),

   or

   Close (Linux), to close the window.

4.1.8.3. Step 16: Plot the flux density as vectors.

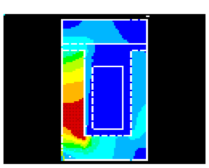
1. **Main Menu> General Postproc> Plot Results> Vector Plot> Predefined**
2. “Flux & gradient” (left column)
3. “Mag flux dens B” (right column)
4. [OK]



4.1.8.4. Step 17: Plot the magnitude of the flux density.

Plot the magnitude of the flux density without averaging the results across material discontinuities.

1. **Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu**
2. Select "Magnetic Flux Density," then "Magnetic flux density vector sum."
3. [OK]

   Next, you will see how the flux density is distributed throughout the entire actuator.

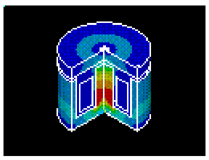
   Up to this point, the analysis and all associated plots have used the 2-D axisymmetric model, with the axis of symmetry aligned with the left vertical portion of the device.

   Mechanical APDL continues the analysis on the 2-D finite element model, but enables you to generate a three-quarter expanded-plot representation of the flux density throughout the device, based on the defined axisymmetry. The function is purely graphical, and no changes to the database occur.

4. **Utility Menu> PlotCtrls> Style> Symmetry Expansion> 2D Axi-Symmetric**
5. (check) “3/4 expansion”
6. [OK]

   Obtain an isometric view for a more meaningful representation.

7. **Utility Menu> PlotCtrls> Pan,Zoom,Rotate**
8. [Iso]
9. [Close]



4.1.8.5. Step 18: Exit the program.

1. Toolbar: **QUIT**
2. (check) “Quit - No Save!”
3. [OK]

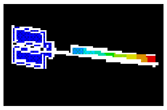
   You have completed this tutorial.

Although you have exited the Mechanical APDL program, you can still view animations via the ANIMATE program (p. 97). The program runs only on Windows computers and is useful for:

- Viewing Mechanical APDL animations on a Windows computer regardless of whether the files were created in Windows (AVI files) or Linux (ANIM files).
- Converting ANIM files to AVI files.
- Sending animations via email or sharing via the Web.

Chapter 5: Micro-Electromechanical System (MEMS) Tutorial

Multiphysics Analysis of a Thermal Actuator



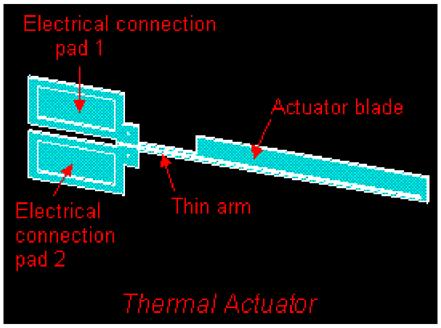
5.1. Multiphysics Analysis of a Thermal Actuator

5.1.1. Problem Specification

Applicable Products: (p. 5)	ANSYS Multiphysics
Level of Difficulty: (p. 6)	Easy
Interactive Time Required: (p. 6)	Approximately 30 minutes
Discipline: (p. 5)	Structural, thermal, electric
Analysis Type: (p. 5)	Static, nonlinear
Element Types Used: (p. 5)	Coupled-Field SOLID227
Features Demonstrated: (p. 5)	Importing an IGES model, SmartSizing, selecting entities, applying voltage, temperature, and displacement boundary conditions, plotting voltage, temperature, and displacement results, animating displacement results, listing heat flow and current
Help Resources: (p. 6)	Coupled-Field Analysis and Direct Coupled-Field Analysis, and SOLID227

5.1.2. Problem Description

This tutorial demonstrates how to analyze an electrical-thermal actuator used in a micro-electromechanical system (MEMS). The thermal actuator is fabricated from polysilicon and is shown below.



The thermal actuator works on the basis of a differential thermal expansion between the thin arm and blade.

The required analysis is a coupled-field multiphysics analysis that accounts for the interaction (coupling) between thermal, electric, and structural fields.

A potential difference applied across the electrical connection pads induces a current to flow through the arm and blade. The current flow and the resistivity of the polysilicon produce Joule heating (I^2R) in the arm blade. The Joule heating causes the arm and the blade to heat up. Temperatures in the range of 700 -1300 ^oK are generated. These temperatures produce thermal strain and thermally induced deflections.

The resistance in the thin arm is greater than the resistance in the blade. Therefore, the thin arm heats up more than the blade, which causes the actuator to bend towards the blade. The maximum deformation occurs at the actuator tip. The amount of tip deflection (or force applied if the tip is restrained) is a direct function of the applied potential difference. Therefore, the amount of tip deflection (or applied force) can be accurately calibrated as a function of applied voltage.

These thermal actuators are used to move micro devices, such as ratchets and gear trains. Arrays of thermal actuators can be connected together at their blade tips to multiply the effective force.

Primary Objective:

- Determine the blade-tip deflection for an applied potential difference across the electrical connection pads.

Secondary Objectives:

- Obtain temperature, voltage, and displacement plots.
- Animate displacement results.
- Determine total current and heat flow.

5.1.2.1. Given

You are provided with a solid model in an IGES file. Dimensions are in micrometers. The thermal actuator has an overall length of approximately 250 micrometers, and a thickness of 2 micrometers. The given potential difference across the electrical connection pads is 5 volts.

Material Properties for Polysilicon	
Young's modulus	169 GPa
Poisson's ratio	0.22
Resistivity	2.3e-5 ohm-m
Coefficient of thermal expansion	2.9e-6/^{o}K
Thermal conductivity	150 W/m^{o}K

5.1.2.2. Approach and Assumptions

Coupled-field problems can be solved using the direct method or the sequential method. The direct method performs the coupled-field analysis in one step using coupled-field elements. The sequential method performs the coupled-field analysis in multiple steps, where the results from one step are used as input to the next step. Coupled field elements are not required for the sequential method.

This tutorial uses the direct method to evaluate the actuator. The direct approach is the most efficient method for this problem. If it were necessary to include the effects of temperature-dependent material properties and/or thermal radiation, however, a more efficient option would likely be the sequential method. The nonlinear thermal-electric problem could be solved using SOLID227 elements with only the TEMP and VOLT degrees of freedom active, and the mechanical problem could be solved using SOLID187 elements. The temperatures calculated in the thermal analysis could be applied as loading to the mechanical model using the **LDREAD** command.

For this problem, you first import the file **actuator.iges**. Next, you define the element type as SOLID227 using the structural thermoelectric degrees of freedom (KEYOPT(1) = 111): UX, UY, UZ, TEMP, VOLT. The element simulates the coupled thermal-electric-structural response.

To define material properties for this analysis, you must convert the given units for Young's modulus, resistivity, and thermal conductivity to μMKSV units. The units have been converted to μMKSV for you.

Material Properties for Polysilicon (μMKSV units)	
Young's modulus	169e3 MPa
Poisson's ratio	0.22
Resistivity	2.3e-11 ohm-μm
Coefficient of thermal expansion	2.9e-6/^{o}K
Thermal conductivity	150e6 pW/μm^{o}K

Next, you mesh the model with the coupled field elements. You then apply voltages to the electrical connection pads and set their temperature to an assumed 30 ^{o}C. You then mechanically fix the electrical connection pads in the X, Y, and Z directions.

Finally, you obtain the solution and post process the results to achieve the analysis objectives.

5.1.2.3. Summary of Steps

Use the information in the problem description and the steps below as a guideline in solving the problem on your own. Or, use the detailed interactive step-by-step solution by choosing the link for step 1.

Import Geometry

1. Import IGES file. (p. 60)

Define Materials

2. Define element type. (p. 61)

3. Define material properties. (p. 61)

Generate Mesh

4. Mesh the model. (p. 62)

Apply Loads

5. Plot areas. (p. 62)

6. Apply boundary conditions to electrical connection pad 1. (p. 63)

7. Apply boundary conditions to electrical connection pad 2. (p. 64)

Obtain Solution

8. Solve. (p. 66)

Review Results

9. Plot temperature results. (p. 67)

10. Plot voltage results. (p. 67)

11. Plot displacement results and animate. (p. 67)

12. List total heat flow and current. (p. 68)

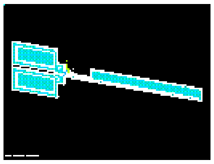
13. Exit the program. (p. 69)

5.1.3. Import Geometry

5.1.3.1. Step 1: Import IGES file.

Read in the actuator model. It is provided for you in the form of an IGES file.

1. **Utility Menu> File> Import> IGES**
2. “No defeaturing”
3. [OK]
4. File name: `actuator.iges` (Click the link to download the file in .zip format.)
5. [OK]



5.1.4. Define Materials

5.1.4.1. Step 2: Define element type.

Specify the element type (p. 5) as coupled-field element SOLID227, and the default degrees of freedom (KEYOPT(1) = 111): UX, UY, UZ, TEMP, VOLT. This problem uses all degrees of freedom.

1. **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**
2. [Add ...]
3. “Coupled Field” (left column)
4. “Tet 10node 227” (right column)
5. [OK]
6. [Options]
7. “Analysis Type K1” choose Structural-thermoelectric
8. [OK]
9. [Close]

5.1.4.2. Step 3: Define material properties.

Enter the material property (p. 6) values for polysilicon. The values are for Young's modulus, Poisson's ratio, thermal expansion coefficient, thermal conductivity, and resistivity. The units for the values are μMKSV.

1. **Main Menu> Preprocessor> Material Props> Material Models**
2. Select “Structural,” then “Linear,” then “Elastic,” then “Isotropic”
3. “EX” = 169e3
4. “PRXY” = 0.22
5. [OK]
6. Select “Thermal Expansion,” then “Secant Coefficient,” then “Isotropic”
7. “ALPX” = 2.9e-6
8. [OK]
9. Select “Thermal,” then “Conductivity,” then “Isotropic”
10. “KXX” = 150e6

11. [OK]

12. Select "Electromagnetics," then "Resistivity," then "Constant"

13. "RSVX" = 2.3e-11

14. [OK]

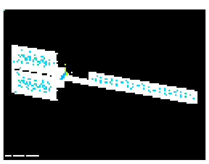
15. **Material> Exit**

5.1.5. Generate Mesh

5.1.5.1. Step 4: Mesh the model.

Obtain a course volume mesh using tetrahedral shapes. Setting the SmartSizing control to 10 (maximum) achieves the course mesh.

1. **Main Menu> Preprocessor> Meshing> MeshTool**
2. (check) "SmartSize"
3. (slide) "Course" = "10"
4. [Mesh]
5. [Pick All]



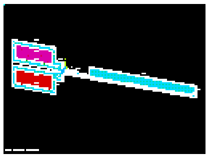
6. [Close] MeshTool

5.1.6. Apply Loads

5.1.6.1. Step 5: Plot areas.

Plot individual areas in preparation for applying the boundary conditions to the electrical connection pads (performed in subsequent steps). First set Mechanical APDL to display each of the areas in a distinguishing color and number:

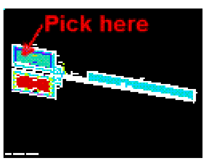
1. **Utility Menu> PlotCtrls> Numbering**
2. (check) "Area numbers" to "On"
3. [OK]
4. **Utility Menu> Plot> Areas**



5.1.6.2. Step 6: Apply boundary conditions to electrical connection pad 1.

Apply the voltage, temperature, and displacement boundary conditions to electrical connection pad 1. First select this pad area so that you have to perform the picking function only once there, even though you are applying three different types of boundary conditions.

1. **Utility Menu> Select> Entities**
2. (first drop down) "Areas"
3. (second drop down) "By Num/Pick"
4. [OK]
5. Pick electrical connection pad 1 (the upper pad). Ensure that you have picked the correct area by holding the mouse button down and dragging the mouse until ONLY the pad area highlights, then release the button.

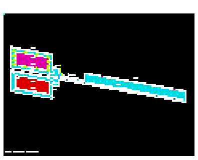
6. [OK]

   Now apply the voltage boundary condition to pad 1.

7. **Main Menu> Preprocessor> Loads> Define Loads> Apply> Electric> Boundary> Voltage> On Areas**
8. [Pick All]

   By choosing Pick All, you pick only the area representing pad 1 because that is the only entity you currently have selected.

9. "Load VOLT Value" = 5
10. [OK]

    Notice the voltage boundary condition symbols added to pad 1.

    Apply the temperature boundary condition to pad 1.

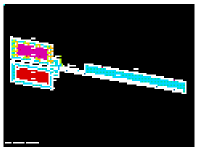
11. **Main Menu> Preprocessor> Loads> Define Loads> Apply> Thermal> Temperature> On Areas**

12. [Pick All]

13. “DOF to be constrained” = TEMP

14. “Load TEMP value” = 30

15. [OK]



Notice the temperature boundary condition symbols added to pad 1.

Apply the displacement boundary conditions to pad 1.

16. **Main Menu> Preprocessor> Loads> Define Loads> Apply> Structural> Displacement> On Areas**

17. [Pick All]

18. “DOFs to be constrained” = UX

19. “Displacement value” = 0

20. [Apply]

21. [Pick All]

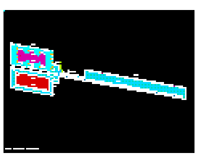
22. “DOFs to be constrained” = UY

23. [Apply]

24. [Pick All]

25. “DOFs to be constrained” = UZ

26. [OK]



Notice that Mechanical APDL has cumulatively added the displacement boundary condition symbols to the pad after you applied them (i.e., X constraint, then Y constraint, then Z constraint).

5.1.6.3. Step 7: Apply boundary conditions to electrical connection pad 2.

Apply the voltage, temperature, and displacement boundary conditions to electrical connection pad 2. The procedure is identical to the one you just performed to add boundary conditions to pad 1.

1. **Utility Menu> Select> Entities**

2. (first drop down) “Areas”

3. (second drop down) "By Num/Pick"

4. [OK]

5. Pick electrical connection pad 2 (the lower pad). Ensure that you have picked the correct area by holding the mouse button down and dragging the mouse until *only* the pad area highlights, then release the button.

6. [OK]

   Apply the voltage boundary condition to pad 2.

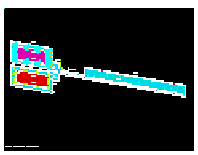
7. **Main Menu> Preprocessor> Loads> Define Loads> Apply> Electric> Boundary> Voltage> On Areas**

8. [Pick All]

   By selecting Pick All, you pick only the area representing pad 2 because that is the only entity you currently have selected.

9. "Load VOLT Value" = 0

10. [OK]



   Notice the voltage boundary condition symbols added to pad 2.

   Apply the temperature boundary condition to pad 2.

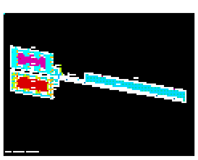
11. **Main Menu> Preprocessor> Loads> Define Loads> Apply> Thermal> Temperature> On Areas**

12. [Pick All]

13. "DOF to be constrained" = TEMP

14. "Load TEMP Value" = 30

15. [OK]



   Notice the temperature boundary condition symbols added to pad 2.

   Apply the displacement boundary conditions to pad 2.

16. **Main Menu> Preprocessor> Loads> Define Loads> Apply> Structural> Displacement> On Areas**

17. [Pick All]

18. "DOFs to be constrained" = UX

19. "Displacement value" = 0

20. [Apply]

21. [Pick All]

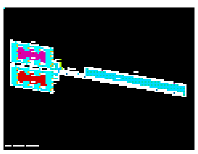
22. "DOFs to be constrained" = UY

23. [Apply]

24. [Pick All]

25. "DOFs to be constrained" = UZ

26. [OK]

    Notice how Mechanical APDL has cumulatively added the displacement boundary condition symbols to the pad after you applied them (X constraint, then Y constraint, then Z constraint).

    Before solving the problem, select the entire finite element model.

27. **Utility Menu> Select> Everything**

5.1.7. Obtain Solution

5.1.7.1. Step 8: Solve.

Initiate the Mechanical APDL solution:

1. **Main Menu> Solution> Solve> Current LS**

2. Review the information in the status window, then select:

   File> Close (Windows platforms),

   or

   Close (Linux platforms), after reviewing the information in the status window.

3. [OK]

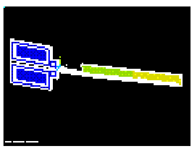
4. [Close] the information window when the solution is done.

5.1.8. Review Results

5.1.8.1. Step 9: Plot temperature results.

Plot the temperature results. This is one objective of this analysis.

1. **Main Menu> General Postproc> Read Results> Last Set**
2. **Main Menu> General Postproc> PlotResults> Contour Plot> Nodal Solu**
3. "DOF solution"
4. "Nodal Temperature"
5. [OK]



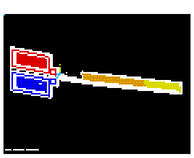
Refer to the legend beneath your plot for a numerical interpretation of the colors in the plot.

The electrical connection pads are the same color, reflecting the constant temperature boundary condition. There is a change in color in the blade, as viewed from the pads-end to the blade-tip end, indicating that the voltage difference across the pads causes a temperature difference across the blade. The thin arm is at higher temperatures than the blade.

5.1.8.2. Step 10: Plot voltage results.

Plot the voltage results. This is one objective of this analysis.

1. **Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu**
2. "DOF solution"
3. "Electric potential"
4. [OK]



Refer to the legend beneath your plot for a numerical interpretation of the colors in the plot. Note that the electrical connection pads are distinctly two different colors, reflecting the voltage difference across the pads. The change in color of the blade, as viewed from the pads-end to the blade-tip end, indicates that the voltage drop from pad 1 to pad 2 is distributed along the electrical conduction path of the actuator.

5.1.8.3. Step 11: Plot displacement results and animate.

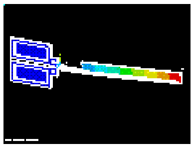
Now plot and animate the displacement results. These are two objectives of this analysis.

1. **Main Menu> General Postproc> PlotResults> Contour Plot> Nodal Solu**

2. “DOF solution”

3. “Y-Component of displacement”

4. [OK]

   Refer to the legend beneath your plot for a numerical interpretation of the colors in the plot. The electrical connection pads are the same color, indicating that the pads are constrained in all directions. Notice the gradual change in color in the blade and thin arm, as viewed from the pads end to the blade-tip end. This display, along with the animation you will generate next, clearly show the bending of the thermal actuator.

   The color of the blade tip indicates a deflection of approximately 3.07 micrometers, resulting from the 5 volts applied across the pads. You have achieved the primary objective of this analysis.

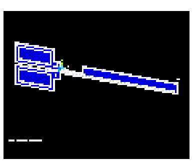
   Generate the corresponding animation.

5. **Utility Menu> PlotCtrls> Animate> Deformed Results**

6. (left column) “DOF solution”

7. (right column) “Translation UY”

8. [OK]

9. [Close] after making desired choices in the Animation Controller.

5.1.8.4. Step 12: List total heat flow and current.

Obtain a listing of results including the total heat flow and current. These are two objectives for this analysis.

1. **Utility Menu> Plot> Areas**

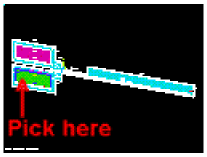
2. **Utility Menu> Select> Entities**

3. (first drop down) “Areas”

4. (second drop down) “By Num/Pick”

5. [OK]

6. Pick the electrical connection pad on the thin side of the actuator (lower pad as shown below).



7. [OK]

8. **Utility Menu> Select> Entities**

9. (first drop down) "Nodes"

10. (second drop down) "Attached to"

11. "Areas, all"

12. [OK]

13. **Utility Menu> List> Results> Reaction Solution**

14. "1st 10 items"

15. [OK]

    Scroll to the bottom of the list and note that the total heat flow is approximately 8.07e9 pW and the total current is approximately 3.23e9 pA.

16. When you are done viewing the listing, choose:

    File> Close (Windows platforms),

    or

    Close (Linux platforms)

17. **Utility Menu> Select> Everything**

5.1.8.5. Step 13: Exit the program.

1. Toolbar: **Quit**

2. "Quit - No Save!"

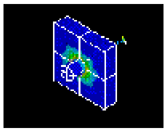
3. [OK]

Although you have exited the Mechanical APDL program, you can still view animations via the ANIMATE program (p. 97). The program runs only on Windows computers and is useful for:

- Viewing Mechanical APDL animations on a Windows computer regardless of whether the files were created in Windows (AVI files) or Linux (ANIM files).
- Converting ANIM files to AVI files.
- Sending animations via email or sharing via the Web.

Chapter 6: Contact Tutorial

Interference Fit and Pin Pull-Out Contact Analysis



6.1. Interference Fit and Pin Pull-Out Contact Analysis

6.1.1. Problem Specification

Applicable Products: (p. 5)	ANSYS Multiphysics, ANSYS Mechanical, ANSYS Structural
Level of Difficulty: (p. 6)	Moderate
Interactive Time Required: (p. 6)	45 to 60 minutes (includes 15 to 20 minutes for solution)
Discipline: (p. 5)	Structural
Analysis Type: (p. 5)	Nonlinear quasi-static
Element Types Used: (p. 5)	SOLID185, TARGE170, CONTA174
Features Demonstrated: (p. 5)	Symmetry boundary conditions, flexible-to-flexible surface contact, contact wizard, automatic time stepping, multiple load steps, symmetry expansion, animation, time history postprocessing, Solution Controls dialog box

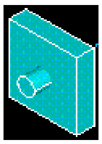
Help Resources: (p. 6)	The *Contact Technology Guide*,SOLID185 , TARGE170, and CONTA174

6.1.2. Problem Description

This is a 3-D analysis of a steel pin contacting a smooth pinhole in a block. Due to the model's inherent symmetry, the analysis occurs on a quarter-symmetry model. Two different load steps are defined.

Objectives: First load step – observe the interference fit stresses of the pin which is geometrically thicker than its pinhole. Second load step – observe the stresses, contact pressures and reaction forces due to the motion of the pin being pulled out of the block.

6.1.2.1. Given



The dimensions of the model are as follows:

- PIN radius = 0.5 units, length = 2.5 units
- BLOCK width = 4 units, length = 4 units, depth = 1 unit
- INHOLE radius = 0.49 units, depth = 1 unit

Both solids are made of structural steel (stiffness = 36e6, Poisson's ratio = 0.3) and are assumed to be flexible.

6.1.2.2. Approach and Assumptions

A quarter-symmetry model is appropriate for simulating the contact phenomena. Two load steps are used to set up the analysis:

- **Load Step 1:** Interference Fit -- solve the problem with no additional displacement constraints. The pin is constrained within the pinhole due to its geometry. Stresses are generated due to the general misfit between the target (pinhole) and the contact (pin) surfaces.
- **Load Step 2:** Pull-out -- move the pin by 1.7 units out of the block using degree-of-freedom displacement conditions on coupled nodes. Explicitly invoke Automatic Time Stepping to guarantee solution convergence. Read results for every 10th substep during solution.

6.1.2.3. Summary of Steps

Use the information in the problem description and the steps below as a guideline in solving the problem on your own. Or, use the detailed interactive step-by-step solution by choosing the link for step 1.

To run this tutorial, a minimum workspace memory of 64 MB is necessary (and 100-200 MB is preferable). Before starting the tutorial, check your workspace memory:

1. **Utility Menu> List> Status> Configuration**

2. Scroll down to the MEMORY STATISTICS heading and read the number of MB for Requested Initial Work Space.

3. If the memory amount is acceptable, proceed with the tutorial. If the number is too low, quit Mechanical APDL without saving changes, restart Mechanical APDL and, in the Interactive dialog, enter the appropriate number in the Memory requested for Total Workspace field before selecting Run.

Input Geometry

1. Read in the model of the pin and the block. (p. 74)

Define Material Property and Element Type

2. Define material. (p. 74)

3. Define element type for solid volume. (p. 74)

Generate Mesh

4. Mesh solid volume. (p. 75)

5. Smooth element edges for graphics display. (p. 76)

6. Create contact pair using Contact Wizard. (p. 76)

Specify Solution Criteria

7. Apply symmetry constraints on (quartered) volume. (p. 77)

8. Define boundary constraints on block. (p. 78)

9. Specify a large displacement static analysis. (p. 78)

Load Step 1

10. Define interference fit analysis options. (p. 79)

11. Solve load step 1. (p. 79)

Load Step 2

12. Set DOF displacement for pin. (p. 79)

13. Define pull-out analysis options. (p. 80)

14. Write results to file. (p. 80)

15. Solve load step 2. (p. 80)

Postprocessing

16. Expand model from quarter symmetry to full volume. (p. 81)

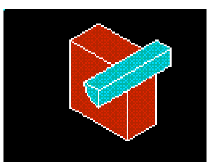
17. Observe interference fit stress state. (p. 81)

6.1.3. Input Geometry

6.1.3.1. Step 1: Read in the model of the pin and block.

Read in the file containing the quarter-symmetry representation of the pin and block.

1. **Utility Menu> File> Read Input from ...**
2. File name: `block.inp` (Click the link to download the file in .zip format.)
3. **OK**



6.1.4. Define Material Property and Element Type

6.1.4.1. Step 2: Define material.

Define the material property (p. 6).

1. **Main Menu> Preprocessor> Material Props> Material Models**
2. Double-click **Structural**, then **Linear**, **Elastic**, and **Isotropic**.
3. **EX** = **`36e6`**
4. **PRXY** = **`0.3`**.
5. Click **OK**.
6. Select **Material> Exit** from the menu bar.

6.1.4.2. Step 3: Define element types.

Define the element type (p. 5).

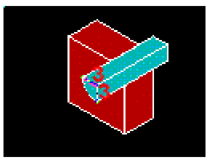
1. **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**
2. Click **Add**.
3. Select **Structural Mass - Solid**.

4. Select **Brick 8node 185** (right column).
5. Click **OK**.
6. Click **Close**.

6.1.5. Generate Mesh

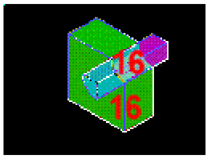
6.1.5.1. Step 4: Mesh solid volume.

1. **Main Menu> Preprocessor> Meshing> MeshTool**
2. Select **Lines - Set**.
3. Pick the horizontal and vertical lines on the front edge of the pin.

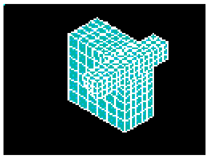


4. Click **OK**.
5. Enter **3** for **No. of element divisions**.
6. Uncheck **SIZE,NDIV can be changed** to indicate **No**.
7. Click **OK**.
8. Select **Lines - Set**.
9. Pick the curved line on the front of the block.

10. Click **OK**.
11. Enter **4** for **No. of element divisions**.
12. Click **OK**.
13. Select **Volumes** from the **Mesh** drop-down menu.
14. Select **Hex**.
15. Select **Sweep**.
16. Click the **Sweep** button.
17. Pick the pin and block volumes.



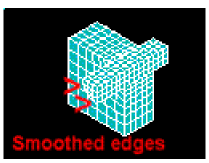
18. Click **OK** and **Close** any warning messages that appear.



19. **Close** the MeshTool.

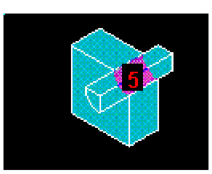
6.1.5.2. Step 5: Smooth element edges for graphics display.

1. **Utility Menu> PlotCtrls> Style> Size and Shape**
2. From the **Facets/element edge** drop-down menu, select **2 facets/edge**.
3. Click **OK**.



6.1.5.3. Step 6: Create contact pair using Contact Wizard.

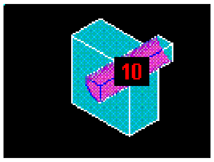
1. **Main Menu> Preprocessor> Modeling> Create> Contact Pair**
2. Select the **Contact Wizard** button (located in the upper left corner of the Contact Manager).
3. Select **Areas** from the **Target Surface** field.
4. Select **Flexible** from the **Target Type** field.
5. Click **Pick Target**.
6. Pick surface of pin hole on block as the target.



7. Click **OK**.
8. Click **Next**.
9. Select **Areas** from **Contact Surface** field.

10. Click **Pick Contact**.

11. Pick surface area of pin as the contact.



12. Click **OK**.

13. Click **Next**.

14. Select **Include Initial penetration**.

15. Select **1** from **Material ID** drop-down menu.

16. Enter **`0.2`** for the **Coefficient of friction**.

17. Click **Optional settings**.

18. Enter **`0.1`** for **Normal penalty stiffness**.

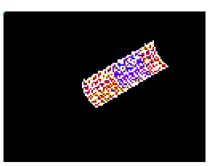
19. Select the **Friction** tab.

20. Select **Unsymmetric** from the **Stiffness matrix** drop-down menu.

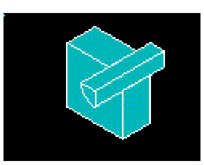
21. Click **OK**.

22. Click **Create**.

23. Click **Finish** and close the Contact Manager.



24. **Utility Menu> Plot> Areas**



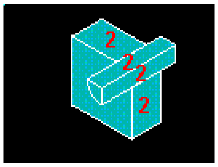
25. **Toolbar>SAVE_DB**.

6.1.6. Specify Solution Criteria

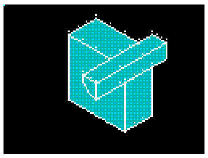
6.1.6.1. Step 7: Apply symmetry constraints on (quartered) volume.

1. **Main Menu> Solution> Define Loads> Apply> Structural> Displacement> Symmetry B. C.> On Areas**

2. Pick the four interior areas that were exposed when original model was quartered.

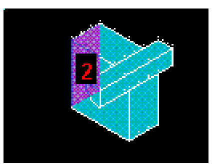


3. Click **OK**.

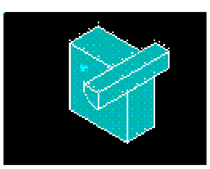


6.1.6.2. Step 8: Define boundary constraints on block.

1. **Main Menu> Solution> Define Loads> Apply> Structural> Displacement> On Areas**
2. Pick left side of block.



3. Click **OK**.
4. Select **All DOF** for **DOFs to be constrained**.
5. Enter `0` for the **Displacement value**.
6. Click **OK** to apply the constraints.



6.1.6.3. Step 9: Specify a large displacement static analysis.

Specify the analysis option as a static analysis in which large deformation effects are to be included. Use the Solution Controls dialog, a central control panel enabling you to adjust the most commonly used settings for a structural static or full transient analysis. The controls consist of tabbed pages, each offering a related set of solution controls.

1. **Main Menu> Solution> Analysis Type> Sol'n Controls**
2. Select **Large Displacement Static** from the **Analysis Options** pull-down menu. Continue with the next step.

6.1.7. Load Step 1

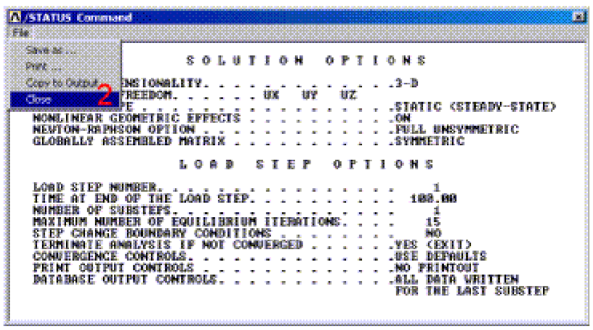
6.1.7.1. Step 10: Define interference fit analysis options.

For both load steps, ramped (rather than stepped) loading is applied automatically.

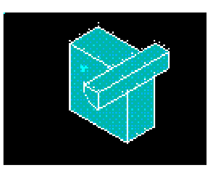
1. Enter **100** as the **Time at end of load step**.
2. Select **Off** from the **Automatic time stepping** drop-down menu.
3. Enter **1** as the **Number of substeps**.
4. Click **OK** to apply the settings.
5. **Toolbar>SAVE_DB**.

6.1.7.2. Step 11: Solve load step 1.

1. **Main Menu> Solution> Solve> Current LS**
2. Review the information in the status window, then **File> Close** to close the window.



3. Click **OK** to begin the solution. Select **Yes** if a Verify window appears, and ignore any warning messages, but do not close the warning message window yet.
4. **Close** the note window when solution is done.
5. **Utility Menu> Plot> Replot**. Continue with the next step.



6.1.8. Load Step 2

6.1.8.1. Step 12: Set DOF displacement for pin.

To observe the effects of pulling the pin out of the block, apply a displacement value of 1.7 to all nodes on the front of the pin.

1. **Utility Menu> Select> Entities**
2. First drop down menu = **Nodes**.

3. Second drop-down menu = **By Location**.
4. Select **Z coordinates**.
5. Enter **4.5** for **Min, Max**.
6. Click **OK**.
7. **Main Menu> Solution> Define Loads> Apply> Structural> Displacement> On Nodes**
8. Click **Pick all**.
9. Select **UZ** for the **DOFs to be constrained**.
10. Enter a **Displacement value** of **1.7**.
11. Click **OK**.

6.1.8.2. Step 13: Define pull-out analysis options.

1. **Main Menu> Solution> Analysis Type> Sol'n Controls**
2. Enter **200** as the **Time at end of load step**.
3. Select **On** from the **Automatic time stepping** drop-down list.
4. Enter **100** for the **Number of substeps**.
5. Enter **10000** for the **Max no. of substeps**.
6. Enter **10** for the **Min no. of substeps**. Continue with the next step.

6.1.8.3. Step 14: Write results to file.

1. Select **Write every Nth substep** from the **Frequency** drop-down list.
2. Enter **-10** in the **where N equals (=)** field.
3. Click **OK**.
4. **Utility Menu> Select> Everything**
5. **Toolbar>SAVE_DB**.

6.1.8.4. Step 15: Solve load step 2.

Mechanical APDL may generate several warning messages while solving the second load step, but they do not impede the solution.

By default, Mechanical APDL displays only the first five warning messages. If more warnings occur (which may happen during the solution of this load step), they are not displayed, nor is the "Solution is Done!" message displayed.

To ensure that the "Solution is Done!" message appears, change the setting that controls the number of messages that are displayed from 5 to 100. Type the following command in the Input Window, then press **Enter**:

```
/NERR,100,100,OFF
```

The command also ensures that your Mechanical APDL analysis does not end if an error occurs during solution.

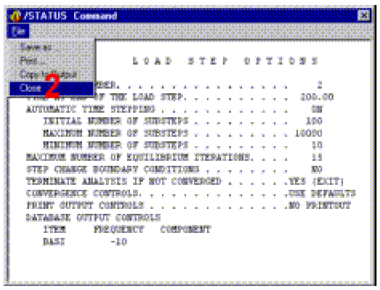
You can then proceed with the following steps to obtain the solution:

1. **Main Menu> Solution> Solve> Current LS**
2. Review the information in the status window, then choose:

   File> Close (Windows),

   or

   Close (Linux), to close the window.

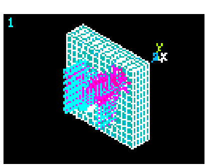
   

3. Click **OK** to begin the solution. Ignore any warning messages, but do not close the warning message window yet.
4. **Close** the note window when solution is done.

6.1.9. Postprocessing

6.1.9.1. Step 16: Expand model from quarter symmetry to full volume.

1. **Utility Menu> PlotCtrls> Style> Symmetry Expansion> Periodic/Cyclic Symmetry**
2. Check **1/4 Dihedral Sym**.
3. Click **OK**.
4. **Utility Menu> Plot> Elements**

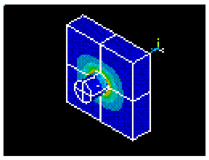
   

5. **Toolbar>SAVE_DB**.

6.1.9.2. Step 17: Observe interference fit stress state.

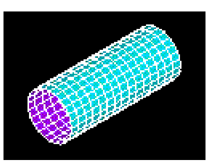
1. **Main Menu> General Postproc> Read Results> By Load Step**

2. Load step number = **1**.
3. Click **OK**.
4. **Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu**
5. Select **Stress** and then **von Mises stress**.
6. Click **OK**.

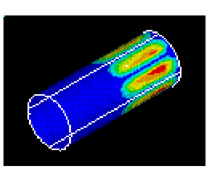


6.1.9.3. Step 18: Observe intermediate contact pressure on pin.

1. **Main Menu> General Postproc> Read Results> By Time/Freq**
2. Enter **`120`** as the **Value of time or freq**.
3. Click **OK**.
4. **Utility Menu> Select> Entities**
5. In the first drop-down list, select **Elements**.
6. In the second drop-down list, select **By Elem Name**.
7. Enter **`174`** as the **Element name**.
8. Click **OK**.
9. **Utility Menu> Plot> Elements**

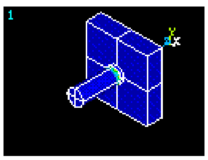


10. **Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu**
11. Select **Contact**.
12. Select **Contact Pressure**.
13. Click **OK**.



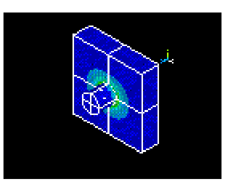
6.1.9.4. Step 19: Observe pulled-out stress state.

1. **Utility Menu> Select> Everything**
2. **Main Menu> General Postproc> Read Results> By Load Step**
3. Enter **2**as the **Load step number**.
4. Click **OK**.
5. **Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu**
6. Select **Stress** and then **von Mises stress**.
7. Click **OK**.



6.1.9.5. Step 20: Animate pin pull-out.

1. **Utility Menu> Plot Ctrls> Animate> Over Results**
2. Select **Load Step Range** for the **Model result data**.
3. Check **Include last SBST for each LDST**.
4. Check **Auto contour scaling** (On).
5. (In the left column) **Contour data for animation** = **Stress**.
6. (In the right column) **Contour data for animation** = **von Mises SEQV**.
7. Click **OK**.



8. Make selections in the Animation Controller (not shown), if necessary, then **Close**.

6.1.9.6. Step 21: Plot reaction forces for pin pull-out.

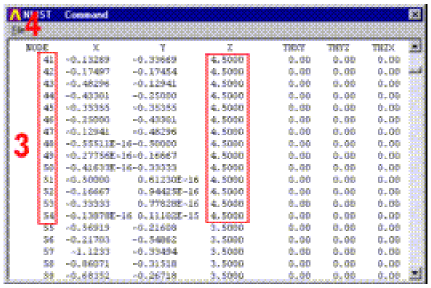
1. **Utility Menu> List> Nodes**
2. Click **OK**.
3. Make note of all the node numbers whose Z coordinates are 4.5.

   Your node numbers may be different from those shown here.

4. **File> Close** (Windows),

   or

   Close (Linux), to close the window.



5. **Utility Menu> Plot> Volumes**

6. **Main Menu> TimeHist Postproc**

7. Select **Add Data** (left most button).

8. Select **Reaction Forces**, **Structural Forces**, and then **Z-Component of Force**.

9. Click **OK**.

10. Pick a node on the front surface of the pin whose number corresponds to one of the nodes listed above for z = 4.5. (Hold down the left mouse button and drag the mouse cursor across the front of the pin. The highlighted node numbers appear in the picking menu. Upclick on the one you want to select.)

11. Click **OK**.

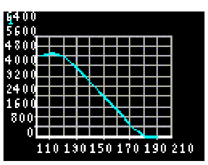
12. Verify that the node number you picked above is displayed in the Node number field.

    Your node number may be different from the one shown here.

13. Click **Graph data** (third button from left) .

14. Close all **EXTREM Command** windows.

15. **File> Close** the **Time History Variables** window.



6.1.9.7. Step 22: Exit the program.

1. Toolbar: **Quit**.

2. Select **Quit - No Save!**

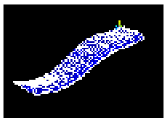
3. Click **OK**.

Although you have exited the Mechanical APDL program, you can still view animations via the ANIMATE program (p. 97). The program runs only on Windows computers and is useful for:

- Viewing Mechanical APDL animations on a Windows computer regardless of whether the files were created in Windows (AVI files) or Linux (ANIM files).
- Converting ANIM files to AVI files.
- Sending animations via email or sharing via the Web.

Chapter 7: Modal Tutorial

Modal Analysis of a Model Airplane Wing



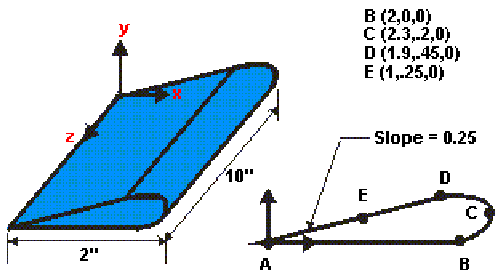
7.1. Modal Analysis of a Model Airplane Wing

7.1.1. Problem Specification

Applicable Products: (p. 5)	ANSYS Multiphysics, ANSYS Mechanical, ANSYS Structural
Level of Difficulty: (p. 6)	Easy
Interactive Time Required: (p. 6)	30 to 45 minutes
Discipline: (p. 5)	Structural
Analysis Type: (p. 5)	Modal
Element Types Used: (p. 5)	PLANE182, SOLID185
Features Demonstrated: (p. 5)	Extrusion with a mesh, selecting, eigenvalue modal analysis, animation
Help Resources: (p. 6)	Modal Analysis, PLANE182 and SOLID185

7.1.2. Problem Description

This is a simple modal analysis of a wing of a model airplane. The wing is of uniform configuration along its length and its cross-sectional area is defined to be a straight line and a spline. It is held fixed to the body of the airplane on one end and hangs freely at the other.



Objective: Find the wing's natural frequencies and mode shapes.

7.1.2.1. Given

The wing is made of low density polyethylene with a Young's modulus of 38x10^3 psi, Poisson's ratio of 0.3, and a density of 8.3E-5 lb_f-sec^2/in^4.

7.1.2.2. Approach and Assumptions

Assume that the side of the wing connected to the plane is completely fixed in all degrees of freedom. The wing is solid and material properties are constant and isotropic.

Use solid modeling to generate a 2-D model of the cross-section of the wing. Create a reasonable mesh and extrude the cross-section into a 3-D solid model (to be meshed automatically).

7.1.2.3. Summary of Steps

Use the information in this description and the steps below as a guideline in solving the problem on your own. Or, use the detailed interactive step-by-step solution by choosing the link for step 1.

Input Geometry

1. Read in geometry input file. (p. 89)

Define Materials

2. Set preferences. (p. 89)

3. Define constant material properties. (p. 89)

Generate Mesh

4. Define element type. (p. 90)

5. Mesh the area. (p. 90)

6. Extrude the meshed area into a meshed volume. (p. 91)

Apply Loads

7. Unselect 2-D elements. (p. 92)

8. Apply constraints to the model. (p. 92)

Obtain Solution

9. Specify analysis types and options. (p. 93)

10. Solve. (p. 93)

Review Results

11. List the natural frequencies. (p. 94)

12. Animate the five mode shapes. (p. 94)

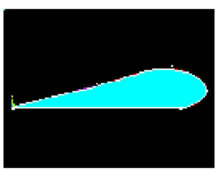
13. Exit the program. (p. 95)

7.1.3. Input Geometry

7.1.3.1. Step 1: Read in geometry input file.

Read in the file contaning the model.

1. **Utility Menu> File> Read Input from ...**
2. File name: `wing.inp` (Click the link to download the file in .zip format.)
3. [OK]



7.1.4. Define Materials

7.1.4.1. Step 2: Set preferences.

Set preferences (p. 7) to filter quantities pertaining to this discipline only.

1. **Main Menu> Preferences**
2. (check) “Structural”
3. [OK]

7.1.4.2. Step 3: Define constant material properties.

1. **Main Menu> Preprocessor> Material Props> Material Models**
2. (double-click) “Structural”, then “Linear”, then “Elastic”, then “Isotropic”
3. “EX” = 38000
4. “PRXY” = 0.3
5. [OK]

6. (double-click) “Density”
7. “DENS” = 8.3e-5
8. [OK]
9. **Material> Exit**

7.1.5. Generate Mesh

7.1.5.1. Step 4: Define element types.

Define two element types: a 2-D element and a 3-D element. Mesh the wing cross-sectional area with 2-D elements, then extrude the area to create a 3-D volume. The mesh will be extruded along with the geometry so that 3-D elements are created in the volume.

1. **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**
2. [Add...]
3. “Structural Solid” (left column)
4. “Quad 4node 182” (right column)
5. [Apply] to choose the Quad 4 node (PLANE182)
6. “Structural Solid” (left column)
7. “Brick 8node 185” (right column)
8. [OK] to choose the Brick 8 node (SOLID185)
9. [Options] for Type2 SOLID185
10. Choose “Simple Enhanced Str” for the element technology.
11. [OK]
12. [CLOSE]
13. Toolbar: **SAVE_DB**

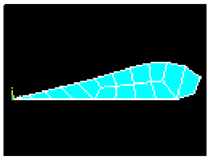
7.1.5.2. Step 5: Mesh the area.

Specify mesh controls to obtain the given mesh density.

1. **Main Menu> Preprocessor> Meshing> Mesh Tool**
2. “Size Controls Global” = [Set]
3. “Element edge length” = 0.25
4. [OK]
5. [Mesh]

6. [Pick All]

7. [Close] Warning.



8. [Close] Meshtool

9. Toolbar: **SAVE_DB**

For this analysis, the maximum node limit of ANSYS ED was taken into consideration. That is why the 4-node PLANE182 element, rather than the 8-node PLANE183 element was used.

The mesh contains a PLANE182 triangle, resulting in a warning. If you are not using ANSYS ED, you can use PLANE183 during the element definitions to avoid the warning.

Your mesh may vary slightly from the mesh shown. As a result, you may see slightly different results during postprocessing. For a discussion of results accuracy, see Planning Your Approach.

7.1.5.3. Step 6: Extrude the meshed area into a meshed volume.

The 3-D volume is generated by first changing the element type to SOLID185, defined as element type 2, then extruding the area into a volume.

1. **Main Menu> Preprocessor> Modeling> Operate> Extrude> Elem Ext Opts**

2. (drop down) "Element type number" = 2 SOLID185

3. "No. Elem divs" = 10

4. [OK]

5. **Main Menu> Preprocessor> Modeling> Operate> Extrude> Areas> By XYZ Offset**

6. [Pick All]

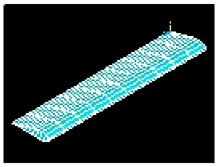
7. "Offsets for extrusion" = 0, 0, 10

8. [OK]

9. **Utility Menu> PlotCtrls> Pan, Zoom, Rotate**

10. [Iso]

11. [Close]



12. Toolbar: **SAVE_DB**

7.1.6. Apply Loads

7.1.6.1. Step 7: Unselect 2-D elements.

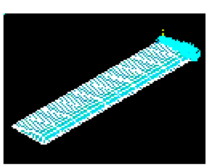
Before applying constraints to the fixed end of the wing, deselect all PLANE182 elements used in the 2-D area mesh, as they are not used for the analysis.

1. **Utility Menu> Select> Entities**
2. (first drop down) "Elements"
3. (second drop down) "By Attributes"
4. (check) "Elem type num"
5. "Min,Max,Inc" = 1
6. (check) "Unselect"
7. [Apply]

7.1.6.2. Step 8: Apply constraints to the model.

Constraints are applied to all nodes located where the wing is fixed to the body. Select all nodes at z = 0, then apply the displacement constraints.

1. (first drop down) "Nodes"
2. (second drop down) "By Location"
3. (check) "Z coordinates"
4. "Min,Max" = 0
5. (check) "From Full"
6. [Apply]
7. **Main Menu> Preprocessor> Loads> Define Loads> Apply> Structural> Displacement> On Nodes**
8. [Pick All] to pick all *selected* nodes.
9. "DOFs to be constrained" = All DOF
10. [OK] Note that by leaving "Displacement" blank, a default value of zero is used.

    Reselect all nodes.
11. (second drop down) "By Num/Pick"

12. [Sele All] to immediately select all nodes from entire database.

13. [Cancel] to close dialog box.

14. Toolbar: **SAVE_DB**

7.1.7. Obtain Solution

7.1.7.1. Step 9: Specify analysis type and options.

Specify a modal analysis type.

1. **Main Menu> Solution> Analysis Type> New Analysis**

2. (check) "Modal"

3. [OK]

4. **Main Menu> Solution> Analysis Type> Analysis Options**

5. (check) "Block Lanczos" (Block Lanczos is the default for a modal analysis.)

6. "No. of modes to extract" = 5

7. "No. of modes to expand" = 5

8. [OK]

9. [OK] All default values are acceptable for this analysis.

10. Toolbar: **SAVE_DB**

7.1.7.2. Step 10: Solve.

1. **Main Menu> Solution> Solve> Current LS**

2. Review the information in the status window, then choose:

   File> Close (Windows),

   or

   Close (Linux), to close the window.

3. [OK] to initiate the solution.

4. [Yes]

5. [Yes]

   Based on previous discussions, the warnings are accepted. Messages appear in the verification window because PLANE182 elements have been defined but not used in the analysis; instead, they were used to mesh a 2-D cross-sectional area.

6. [Close] to acknowledge that the solution is done.

7.1.8. Review Results

7.1.8.1. Step 11: List the natural frequencies.

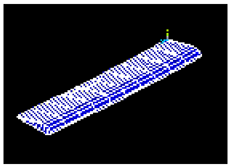
1. **Main Menu> General Postproc> Results Summary**
2. [Close] after observing the listing.

7.1.8.2. Step 12: Animate the five mode shapes.

Set the results for the first mode to be animated.

1. **Main Menu> General Postproc> Read Results> First Set**
2. **Utility Menu> PlotCtrls> Animate> Mode Shape**
3. [OK]

   Observe the first mode shape:

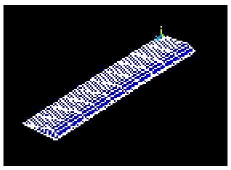
   

4. Make selections in the Animation Controller (not shown), if necessary, then choose Close.

   Animate the next mode shape.

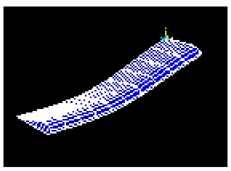
5. **Main Menu> General Postproc> Read Results> Next Set**
6. **Utility Menu> PlotCtrls> Animate> Mode Shape**
7. [OK]

   Observe the second mode shape:

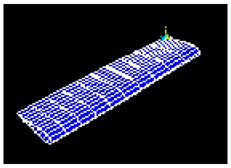
   

   Repeat red steps 4 through 7 above, and view the remaining three modes.

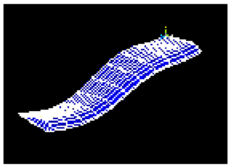
   Observe the third mode shape:

   Observe the fourth mode shape:



Observe the fifth mode shape:



7.1.8.3. Step 13: Exit the program.

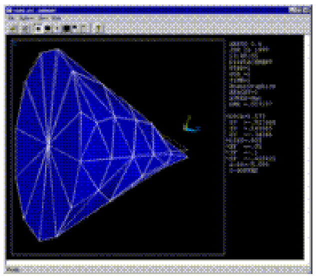
1. Toolbar: **QUIT**
2. (check) “Quit - No Save!”
3. [OK]

Although you have exited the Mechanical APDL program, you can still view animations via the ANIMATE program (p. 97). The program runs only on Windows computers and is useful for:

- Viewing Mechanical APDL animations on a Windows computer regardless of whether the files were created in Windows (AVI files) or Linux (ANIM files).
- Converting ANIM files to AVI files.
- Sending animations via email or sharing via the Web.

Chapter 8: ANIMATE Program

Using ANIMATE to View Animations on a Windows Computer



The standalone ANIMATE program enables you to view Mechanical APDL animation files on a Windows computer regardless of whether the files were created on a Windows (AVI files) or Linux (ANIM files) computer. You can run ANIMATE without Mechanical APDL. Its small 250kB file size makes it easy to transport.

If you create and view AVI animation files on a Windows computer, the ANIMATE program provides a better frame speed and window-size control than does Windows Media Player.

For ANIM files created on Linux workstations, you can view the files directly on a Windows computer using the ANIMATE program. On the Windows computer, you can also *convert* an ANIM file to an AVI file (a substantial file-size reduction).

When using ANIMATE to view 3-D model animations saved as ANIM files, you can pan, zoom, or rotate the model *while* the animation is in progress.

Try the ANIMATE Program

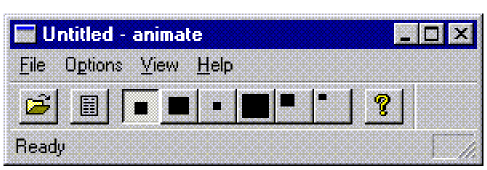
To experiment with ANIMATE, use a Windows computer and have an AVI (Windows) or ANIM (Linux) animation file ready.

If you have just created an animation in Mechanical APDL and want to use it with ANIMATE now, save the animation to a file and name it: **Utility Menu> PlotCtrls> Animate> Save Animation**

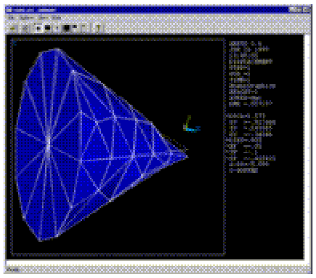
For Linux users, create the ANIM file using pixmap animation (rather than display list animation): **Utility Menu>PlotCtrls>Device Options**, then check Pixmaps for Animation mode. Transfer the file to a Windows file system (via FTP, SAMBA, or some other file-system-transfer utility).

Experiment with ANIMATE program as follows:

1. Start `Animate.exe`.

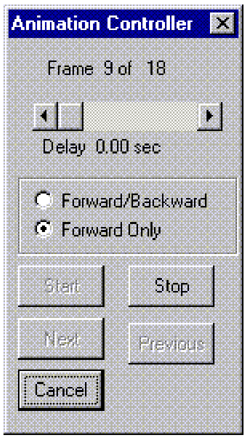


2. Select any AVI or ANIM animation file created using pixmap animation.



3. Click **Controller**.

   Result: The animation controller appears.



4. Experiment with the controls on the Animation Controller.

5. With the animation still running, experiment with window sizes:

   Click on the various window buttons on the toolbar.

6. If viewing an AVI file, you have completed this procedure.

   End the ANIMATE program: **File** > **Exit**

   If you are viewing an ANIM file, continue with the remaining steps.

7. Convert the ANIM file to an AVI file: **File** > **Save as**

   Specify a file name and click **Save**.

8. End the ANIMATE program: **File** > **Exit**

9. Compare the size of the original ANIM file with the size of the AVI file. Note the smaller file size of the AVI file.