

A Practical Starters Guide for Modelling Plate Structures in ANSYS Classic

Arno Landewers (Viro, Engineering Analysis Departement, Vlaardingen, Netherlands)

A starting ANSYS user may find it difficult to gather the necessary information to model from a clean sheet a structure build up from plate material. Available manuals are usually focussing on all available functions in ANSYS (which are a lot), while in practise only a few are used. Furthermore, manuals tend to give examples on imported models, volume models or beam models, as plate structures are only used in specific fields.

Below a short and practical guide for the modelling of a plate structure in ANSYS Classic is given.

It is strongly recommended to explore the commands mentioned further using ANSYS help (**Help/Help topics/Mechanical APDL/Command Reference**). Below guide is only intended as a first start; discussed methods can (and should be) optimized for personal use.

1. Modelling of geometry

Modelling is done in the pre-processor (**Main menu/pre-processor** or **/prep7** at the command line).

1.1. Units

ANSYS does not use units, you just choose suitable ones. For example: when input length units are chosen in meters, and loads kN, then stress results are calculated in kN/m², and deflections in m. As in general rules give allowable stress in N/mm² (MPa), it is recommended to work in mm and N.

1.2. Plate modelling: from keypoint to line to area

In general the modelling starts with definition of keypoints using coordinates. The keypoints are used to make lines, and the lines are used to make areas. To these areas properties (called *reals*, in this case representing a plate thickness) and element types have to be assigned, which is sufficient to mesh the area.

1.3. Create keypoints

Use local reference systems. The global system is made active using command **csys,0**, the local system with **csys,4**. Keypoints can be made by the command **k,,x,y,z**, where **x,y,z** are the coordinates in the active coordinate system. After changing to **csys,4** (local system), the origin of the local system can be put into a keypoint by **kwpa,p**, and a pick on the keypoint. The local system can be put in between two keypoints by picking both keypoints. Using **wpoffs,x,y,z** allows you to move the local coordinate system to position **x,y,z** (coordinates in local system). So there are two methods to create keypoints in space: by **k,,x,y,z**; with **x,y,z** are the keypoint coordinates; or using the **wpoffs** command (which puts the coordinate system on the keypoint location), combined with the command **k,,0,0,0** (creating a keypoint in the local system origin). The "wpoffs" method is recommended for more complex geometries. Keypoints created are assigned a keypoint number, which are displayed in the select keypoint menu after **ksel,p** is entered.

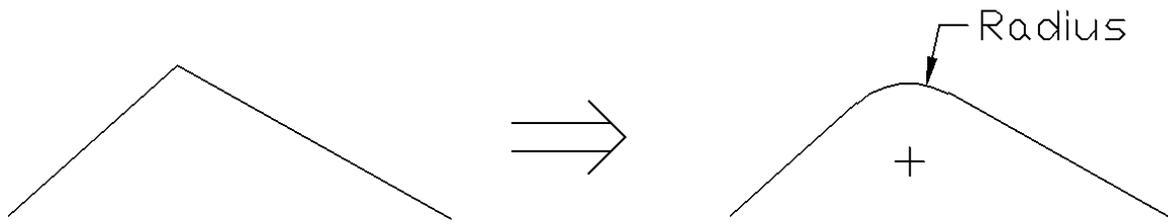
With **kwplan,p**, the local system can be lined up with keypoints. First pick origin, then pick keypoint in x-direction, then pick keypoint in normal direction. A coordinate system can be rotated using **wprota,thxy,thyz,thzx**, where **thxy**, etc are the rotation angles (in degrees) between the axes. Keypoints can be deleted by **kdele, p** (pick keypoint to be deleted, or enter keypoint number instead of **p**).

1.4. Create lines

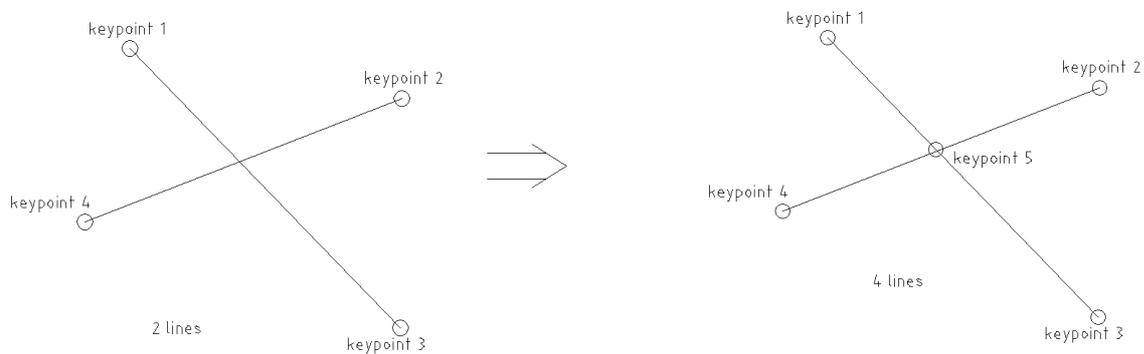
Lines can be created in between keypoints, by **l,p**, then picking the keypoints, or, when you know the keypoint numbers (say 3 and 4), by typing **l,3,4** (a line between keypoint 3 and 4 is created).

Circles can be created using the **Preprocessor/Modelling/Create/Arcs/Full circle** menu (or the "circle" command). Enter the coordinates in active system, then enter the radius. The circle is created in the xy plane of the active system, z is normal to the circle, and is default build-up of 4 lines.

In the same menu the "line fillet" option can be found; it creates fillets between lines by picking both lines and entering a fillet radius; see also below figure.



Make use of crossing lines, see below figure. Cut them using “line partition” (menu **Preprocessor/Modelling/Operate/Booleans/Partition/Lines**). Click the crossing lines and hit “apply”, and new lines and keypoints are created (see below figure). Crossing lines should always be cut, otherwise areas made using these line (and the mesh on these areas) are not connected.



Lines can be deleted using **ldelete, p** (pick lines to be deleted).

Other useful tools: (can all be found in “modelling”)

Move

Copy

Reflect (mirroring)

Using **lplo, all**, modelled lines can be displayed. With **lplo, p** selected lines can be displayed.

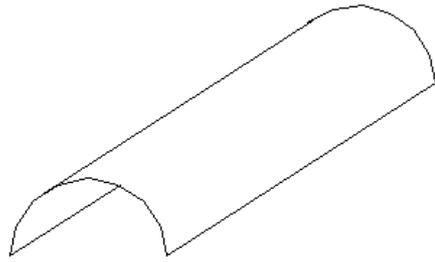
1.5. Create areas

Areas are created using an enclosed pattern of lines, using the command **al, p** (pick lines, then “apply”).

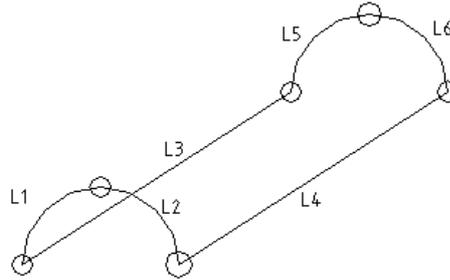
Note: bended areas can only be created using 4 lines; see example below.

Areas can be deleted using **adele, p**.

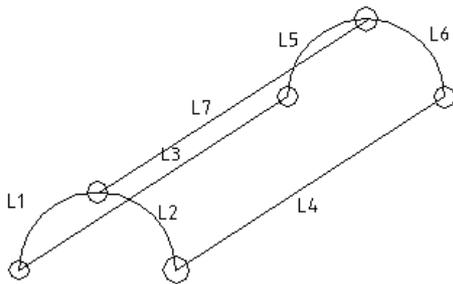
A hole can be created by subtracting areas. Make an area, ignoring the hole. Then make an area of the cut-out. Use “subtract areas” via the menu **Preprocessor/Modeling/Operate/Booleans/Subtract/Areas**. Pick the base area, then pick the cut-out area and hit “apply”.



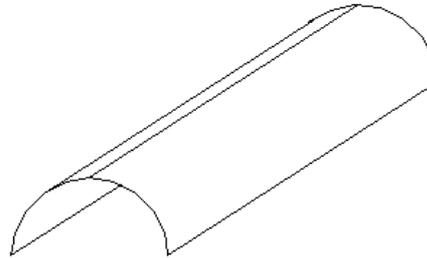
Area to be created



Impossible to model; contains too much line for one area.
One line missing (circle-halves build up by two lines)



Add 7th line



Area to be created, contains two areas

All selected areas can be displayed using **aplot,all**.

1.6. Selecting/reselecting

Use **asel,,p** to select areas (by pick); the selected areas only will be displayed after **/rep**.

To unselect areas from a structure: **asel,u,p**; the picked areas are removed from the display after **/rep**.

To add areas to the selection: **asel,a,p**. **Allsel**: select all areas.

The same functionality is available for lines, using **lsel** instead of **asel**.

Via the select/entities menu (on top of the screen) the same can be achieved.

2. Preprocessing: meshing

2.1. Create elements types

Element types to be used (in our case one of the offered shell elements) have to be defined. ANSYS identifies element types with a number. For example: beam188, link10 or shell63 (if which link10 and shell63 are not officially supported anymore by ANSYS, they can however still be used without any problem and are still described at the Internet).

2.2. Create material properties

Most convenient is to use the **mp** command (or alternatively: **/Preprocessor/Material Props/Material Models**). Several materials can be defined, identified with a number. Defining structural steel this way (as material number 1):

mp,ex,1,2.1E5 ! modulus of elasticity in N/mm²

mp,dens,1,0.785e-05 ! density in kg/mm³

mp,prxy,1,0.3 !Poisson's ratio

2.3. Create reals

A real is a property for an element type. A "real 30" can for example refer to shell elements with thickness 30 mm, or pipe elements with diameter 30 mm.

Reals are defined in the **Preprocessor/Real constants/add real** menu (or alternatively the **r** command, see ANSYS help). Hitting the "add" button, an element type can be chosen. After that a (arbitrary) real constant number can be chosen (it is strongly recommended, to ease the bookkeeping, to choose a number reflecting the main property; for example real 25 for a plate with thickness 25, or real 763 for a tube with diameter 763) and the properties can be entered.

So in general different reals from the same element type exist in a model (for example reals for shell elements with thickness 10 mm, 15 mm and 25 mm).

2.4. Actual meshing

Meshing (so the creation of nodes and elements) is done using the **Preprocessor/Meshing** menu. In **Mesh Attributes**, reals can be assigned to areas. Using "picked areas" in the Mesh Attributes menu, areas which should have a certain real assigned to can be picked, and after hitting "apply" a form appears where material number, real constant number and element type can be assigned (note that real number and element type number should match with the combinations defined for the reals!).

After assigning reals to all areas, the meshing can be started by the command, **amesh,p** (picked areas meshed), or **amesh,all** (all areas meshed, only recommended for simple structures). The element size can be set using **esize, element size** (for example: **esize, 30** creates 30 x 30 mm for shells).

Using **aclear, p**, a mesh can be deleted from picked areas (and **aclear, all** on all areas).

The mesh (elements) can be displayed using **eplo**. The element mid-planes are displayed.

3. Assigning constraints and loads

Constraints and loads are assigned to nodes (so after meshing). Point loads and moments are assigned using **f,p**; after picking the nodes, a load direction can be chosen and the value can be entered. Constraints (seen by ANSYS as displacements with value 0) can be added using **d,p**; after picking of nodes the directions (or rotation directions) can be chosen.

Accelerations are assigned using **acel,ax,ay,az**, where ax...az are the accelerations in x, y and z-direction. With y the vertical, gravitation is modelled using **acel, 0, 9.81, 0**.

Loads and constraints can also be applied using the **Preprocessor/Loads/Define Loads/Apply/Structural** menu, where also possibilities exist for applying pressure loads or line loads.

3. Solving

Solving is executed in the solution module (**/solu**).

For standard solving, type solve on the command line, or use the menu Solution/Solve/Current LS. Be sure everything is selected (use **allsel**). ANSYS indicates whether the model is solved, or gives error messages.

4. Postprocessing

Postprocessing is done in the general Postproc module (**/post1**).

Display of reaction loads via **prrsol** command. Gives a list of reaction loads for all constrained nodes; at the end of the list summed totals are given.

Displaying of (von Mises) stresses in colour plot via command **plnsol,s,eqv**. The scale can be adjusted via the command **/contour,,9,0,,vmax**, where 9 is the number of contours (9 is max), 0 the minimum stress, vmax the maximum stress. Display is adjusted only after **/rep** is entered. Deformations can be displayed via **plnsol,u**. All options can also be find via **General Postproc/Plot Results/Nodal Solu**. Via **List Results** numerical results are tabulated.