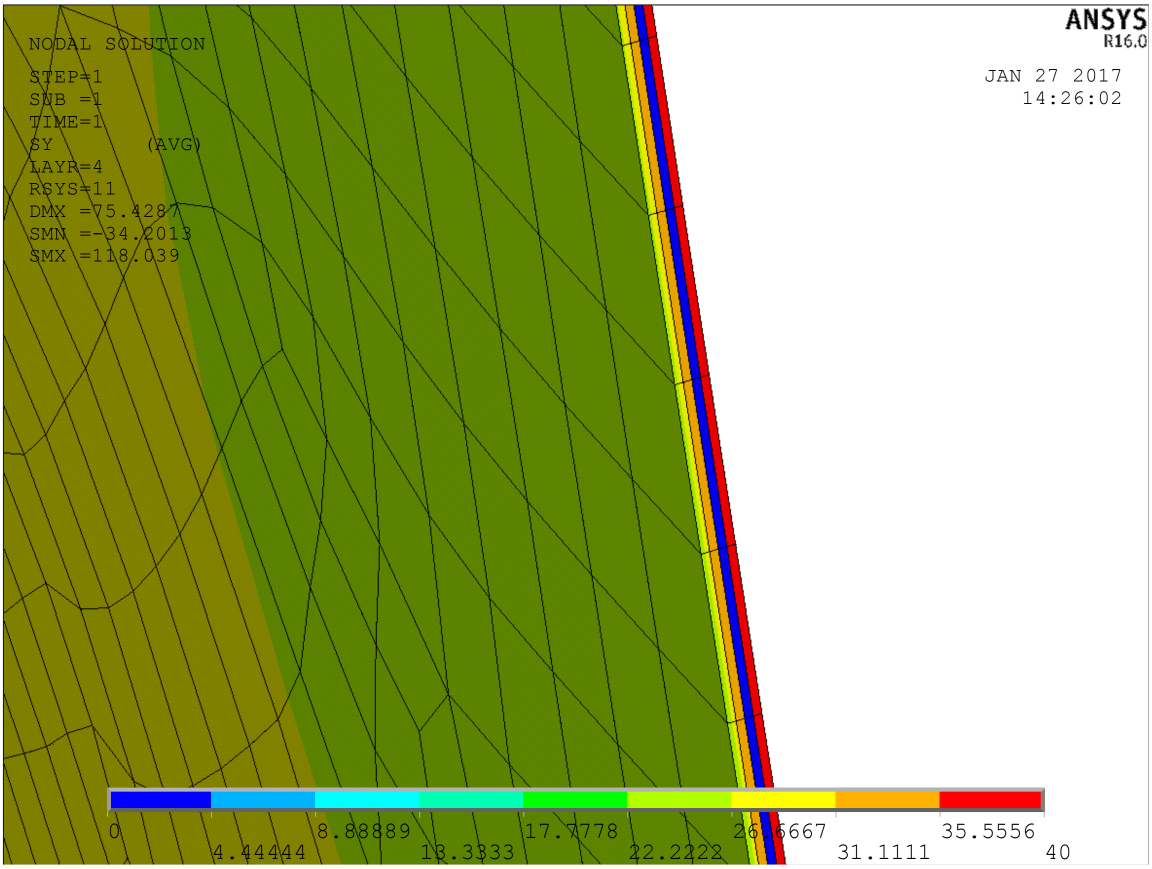

A practical guide for modelling of laminated composites in ANSYS classic
Arno Landewers, VIRO, Vlaardingen, Netherlands

Although modelling of anisotropic layered materials in ANSYS is quite straightforward, many tutorials only discuss cases where all elements are nicely aligned with the global coordinate system, so that every user with basic ANSYS knowledge can produce results without bothering for orientations. However, in most practical situations, elements are not all aligned, and geometries are three-dimensional instead of a flat plane. This basic guide concentrates on some more practical aspects of the modelling and post-processing of laminated composites in ANSYS Classic (APDL).

Element types
Within ANSYS there are many different elements that can be used to model composites. Here the commonly used SHELL181 structural shell element is used. The actual laminates are defined using the SECDATA function. The full APDL syntax to create a composite with 4 layers of 1 mm with different orientations is as below:

```
ET,1,SHELL181          ! definition of element type
OUTRES,ALL,ALL         ! all solution data written to database
KEYOPT,1,3,2           ! full integration
KEYOPT,1,8,2           ! layer results written to database
SECTYPE,1,SHELL
SECDATA,1,1,90  ! Layer 1: thickness 1 mm, material # 1, orientation 90 deg
SECDATA,1,2,0   ! Layer 2: as layer 1, orientation 0 deg
SECDATA,1,3,45  ! Layer 3: as layer 1, orientation 45 deg
SECDATA,1,4,-45 ! Layer 4: as layer 1, orientation -45 deg
```

The material definition should contain values for elasticity modulus, shear modulus and Poisson's ratio in all three directions, where the reference system is the orientation of the layer (specified in SECDATA). Material properties, stresses, and strains for layered elements are based on these layer coordinate systems, which are directly association with the element coordinate system. Therefore, notion (and bookkeeping) of the actual orientation of the elements is essential.

Element orientation

The element coordinate system orientation is default. For the (4-nodes) SHELL181 element default x-axis direction is parallel to the element side formed by the first two nodes (i.e. the two nodes with the lowest node numbers), the y-axis points from the first node to the last node (i.e. the node with the highest number).

Element coordinate systems can be displayed as a triad with the /PSYMB,ESYS,1 (with /eshape,0, so without display of thickness real value), as illustrated in below figure 1. Individual layers can be selected by the LAYER command.

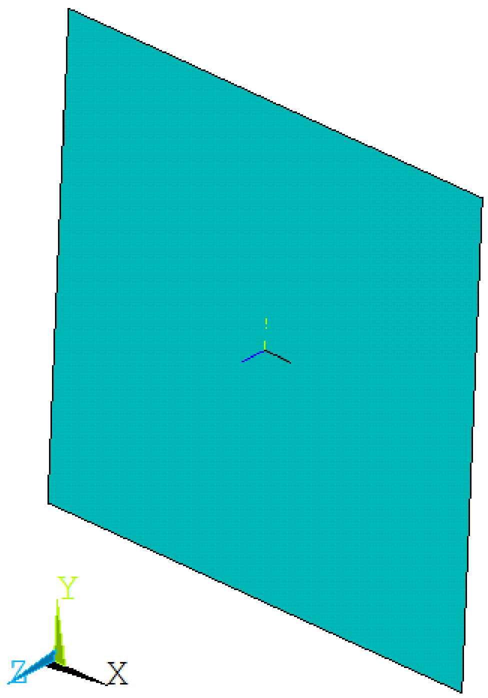

Figure 1. Element coordinate system as displayed in the center of the element. Colors refer to orientations: black for x, green for y and blue for z. These colors are also used for the global system, which can be displayed using the /TRIAD command.

The orientation can be changed by making it parallel to a previously defined local coordinate system, using the ESYS command. Note that this has to be set before meshing. A local coordinate system parallel to the plane of the elements can be defined, using the CSKP (orientation using keypoints) or CS (orientation using nodes) commands. This is illustrated in below figures 2 and 3.

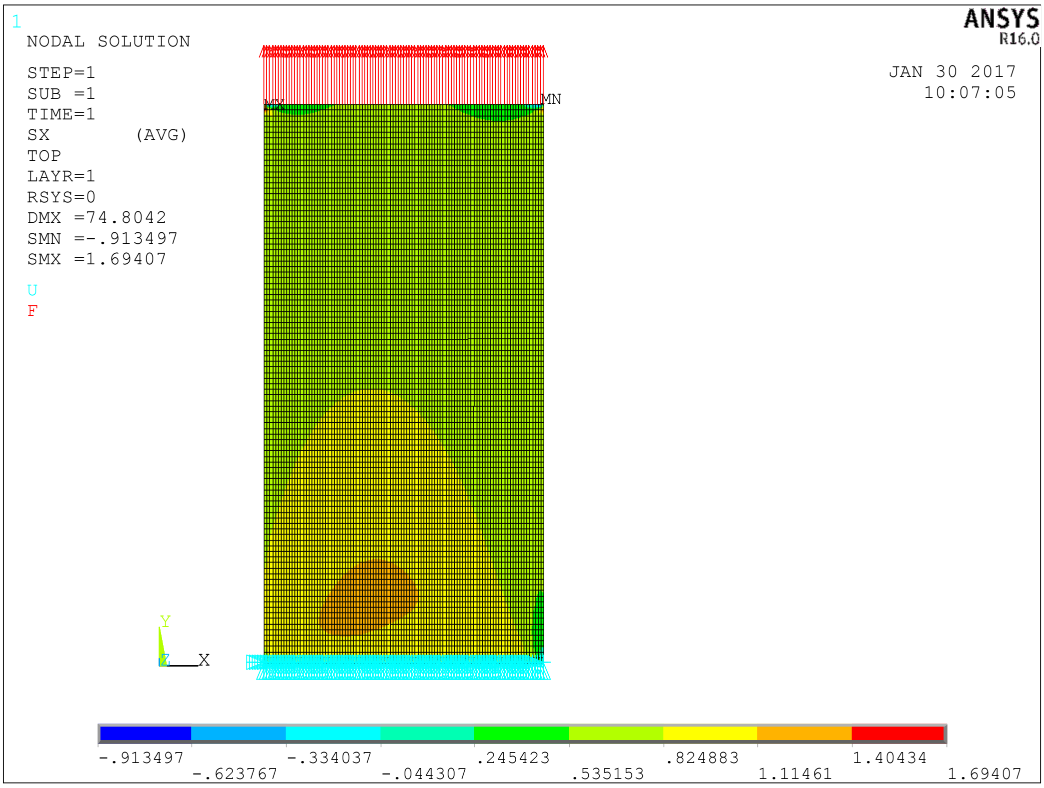

Figure 2. Elements "automatically" oriented according global coordinate system.

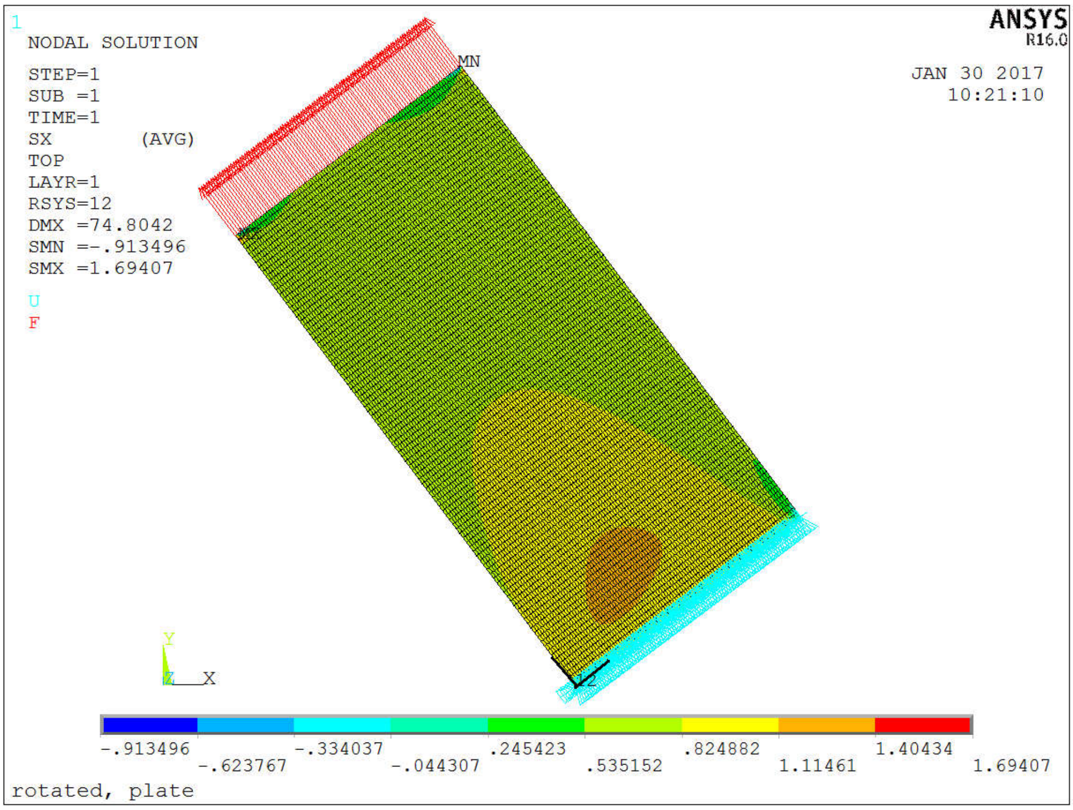

Figure 3. Situation with plate rotated in global system. Plate rotation took place before the creation of the elements, so a local coordinate system (here: system 12) had to be defined.

To create elements which are aligned with a (rotated) area, as shown in figure 3, we need to create a local coordinate system. The operation (including the rotation) in APDL looks like:

```
CSYS,2 !select global cylindrical coordinate system
ASEL,ALL !select area (for pick: ASEL,p)
AGEN,2,ALL,,,0,37,,10,,1 ! rotate area (including lines and keypoints) with 37 deg in csys,2
CSKP,12,0,1,2,4 ! create local orientation system 12, origin in keypoint 1, x-axis keypoints 1-2, y-axis keypoints 1-4
CSYS,12 ! change to reference system 12
ALLSEL
ESYS,12 ! set element coordinate system
AMESH,ALL ! mesh all areas
NROTAT,ALL ! rotate all node orientations to reference system 12 (so constraints and loads directions are correct)
```

Note that when the NOELEM = 0 option in the AGEN command is used (generate nodes and elements associated with the original area in case of rotation after meshing), the generated elements remain to have their original (i.e. non-rotated) orientation. This also accounts for the NGEN command.

Only after the display coordinate system is changed to the local system 12 (by RSYS,12), results display (and printouts) will be in the system aligned with the plate/layer, see figure 3.

In the example shown in figure 4, the elements are not "automatically" aligned with the global system. Before meshing, a coordinate system to which the elements have to be aligned is to be specified. Note that the global system (csys,0) cannot be used, as this option is used to keep the element orientations as default.

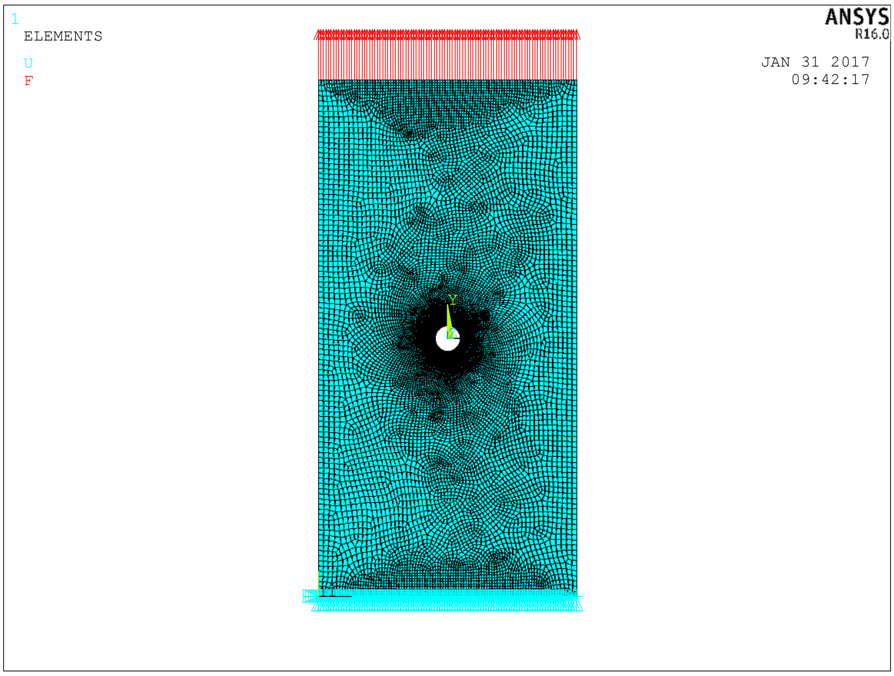

Figure 4. Situation with elements which are not aligned (with any coordinate system)

Similar to the method described above, in APDL this looks like (including keypoint and area generation):

```
ndiv=100 !number of elements
a=1000  ! plate width
b=2000  ! plate height
r=100  ! circle radius

! geometry
K,1,-a/2,-b/2,0
K,2,a/2,-b/2,0
K,3,a/2,b/2,0
K,4,-a/2,b/2,0  ! generate plate corner keypoints 1 to 4
CSYS,0 ! set coordinate system to global
A,1,2,3,4 ! create rectangular plate area
K,5,0,0,0  ! generate circle centre keypoint, which is in plate center (equal to global coordinate system origin)
CIRCLE,5,r/2 ! generate circle
A,6,7,8,9 ! create circle area
ALLSEL
ASBA,1,2 ! substract circle area from rectangular plate area

 ! meshing
CSKP,11,0,1,2,4 ! create local coordinate system 11, origin in keypoint 1, x-axis along 1-2, y-axis 2-4
ESYS,11 ! sets element coordinate systems to local system 11
ESIZE,,ndiv
ALLSEL
AMESH,all
```

After solving, stresses can be assessed per layer as shown in figure 5.

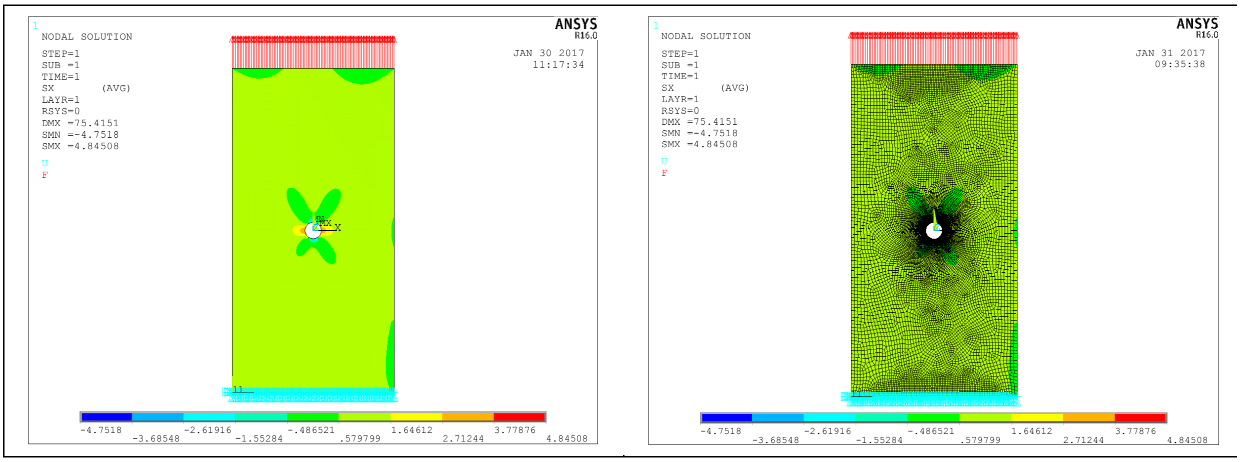

Figure 5. Stress results for elements aligned with coordinate system 11. Note that both RSYS,0 and RSYS,11 could be used.

Also here, after rotation of the plate and subsequent element generation according to above described procedure an equivalent result is obtained provided element orientations and results display are associated with the correct (aligned to the plate/layer) coordinate system.

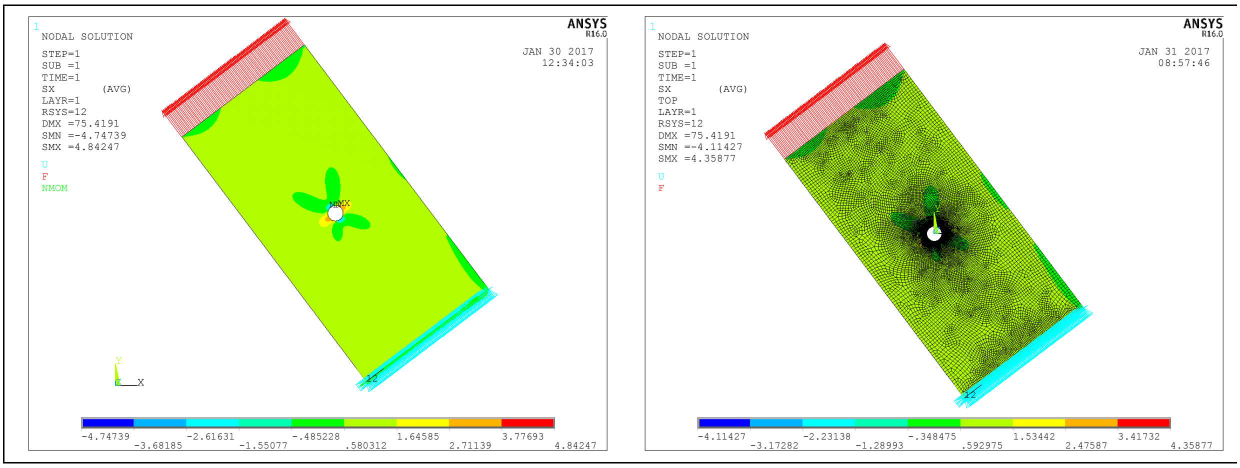

Figure 6. Stress results for rotated plate. With proper use of element orientations, results are as expected equal to the results show in figure 5.

So the general message is that assessing component results only makes sense in case element orientation and results display/printout are in line with these components.

Structural calculations for laminated composites
After modelling, taking above considerations into account, the actual calculations are equal to calculations done for non-laminated (isotropic) materials, including buckling and non-linear analyses.

Note that for assessing delamination, a different modelling approach has to be used, where the actual laminates are modelled separately, and interface elements (like INTER205) have to be used.

Failure criteria
For laminated composites, failure criteria different from isotropic materials are used. ANSYS nodal solution results give under "failure criteria" for instance the common used Maximum Stress and Tsai-Wu criterion. However, these criteria can only be used after manual input of allowable stresses and strains, using the FC command.