Erling Wiig

MSC/PATRAN LAMINATE MODELER COURSE PAT 325

Workbook

P3*V8.0*Z*Z*Z*SM-PAT325-WBK

Table of Contents

Page

Erling Wiig

LESSON 1

Composite Model of Loaded Flat Plate

What you will learn in this exercise

7 MSC/Patran's capabilities with Composites
8 Create a orthotropic material, a lamina and a layup
9 Postprosess calculations done on a composite model
10 The shortcomings in traditional composites calculations

THE MODEL

We will model a 1x1 meter plate, we will use millimetres as length measurement.
The plate is 4 mm thick and is a laminate made up of 16 plies with equal thickness. The laminate is uniform on the plate. The plies have two orientations: 0 and 90 degrees, i.e. parallel to the plate edges.

The material properties of the lamina are:
E-modulus: E_{11} = 181000 MPa, E_{22} = 10300 MPa
Shear modulus: G_{12} = 7170 MPa, G_{23} = 5000 MPa, G_{13} = 7170 MPa
Poisson Ratio: 0.28
Density: 1.6E-09 kg/mm^3

The plate is fixed along one end and supported vertically in one of the other two corners.
The plate is loaded with a uniform pressure of 0.1k Pa, giving an total force acting on the plate of 100 Newtons.

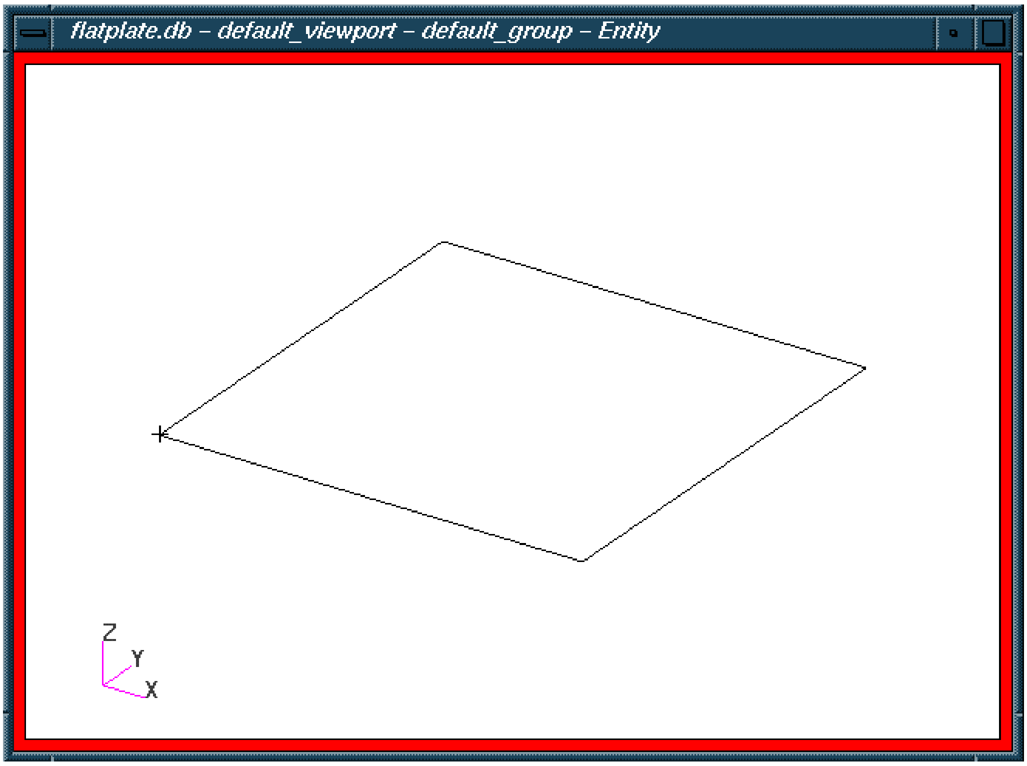
We want to investigate the occuring stresses and maximum deflection of the plate.

1 Open a new filebase using the default template database

File, New...
New Database Name: enter “flatplate”
Ok
Ok

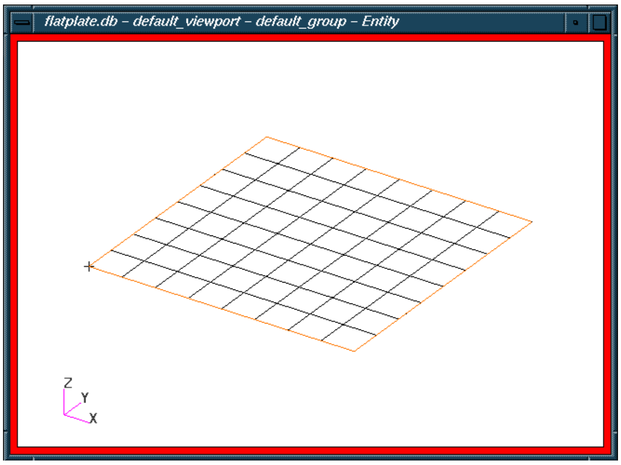
2 Create a plate 1 m square using units of Newton and Millimeter

♦ Geometry
Action : Create
Object : Surface
Method : XYZ
Vector Coordinates List : enter “<1000 1000 0>”
Origin Coordinates List : enter “[0 0 0]”
Apply



This is all the geometry needed for this model.

3 Create a mesh



♦ Finite Elements
Action : Create
Object : Mesh
Method : Surface
Global Edge Length : enter "125"
Element Topology : select Quad4
Mesher : select Isomesh
Surface List : pick Surface 1
Apply

4 Define loads and boundary conditions

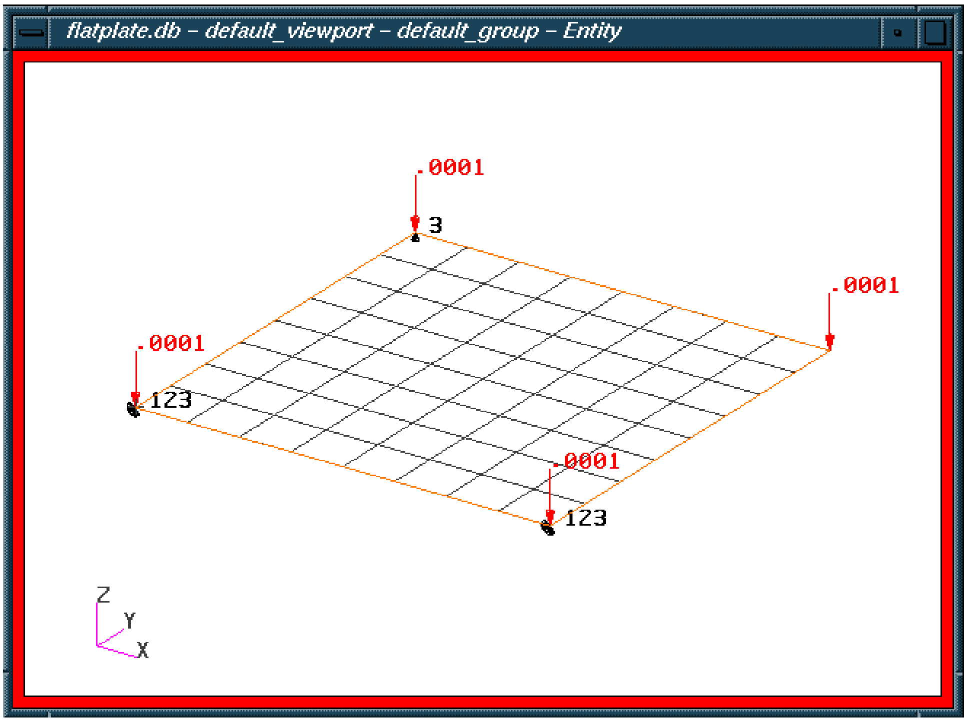
♦ Loads/BCs
Action : Create
Object : Displacement
Method : Nodal
New Set Name : enter "Fixed Line"
Input Data ...
Translations <T1 T2 T3>: enter "<0,0,0>"
Ok
Select Application Region...
Application Region : pick Surface 1.4 (edge)
Add
Ok
Apply

Action : Create
Object : Displacement

Method : Nodal
New Set Name : enter “Supported Point”
Input Data ...
Translations <T1 T2 T3> : enter “<,,0>”
Ok
Select Application Region...
Application Region : pick Point 2
Add
Ok
Apply

Action : Create
Object : Pressure
Method : Element Uniform
New Set Name : enter “Pressure Load”
Target Element Type : 2D
Input Data ...
Top Surf Pressure : enter “0.0001”
Ok
Select Application Region...
Application Region : pick Surface 1
Add
Ok
Apply

Note the pressure 0.0001 which is in MegaPascals.



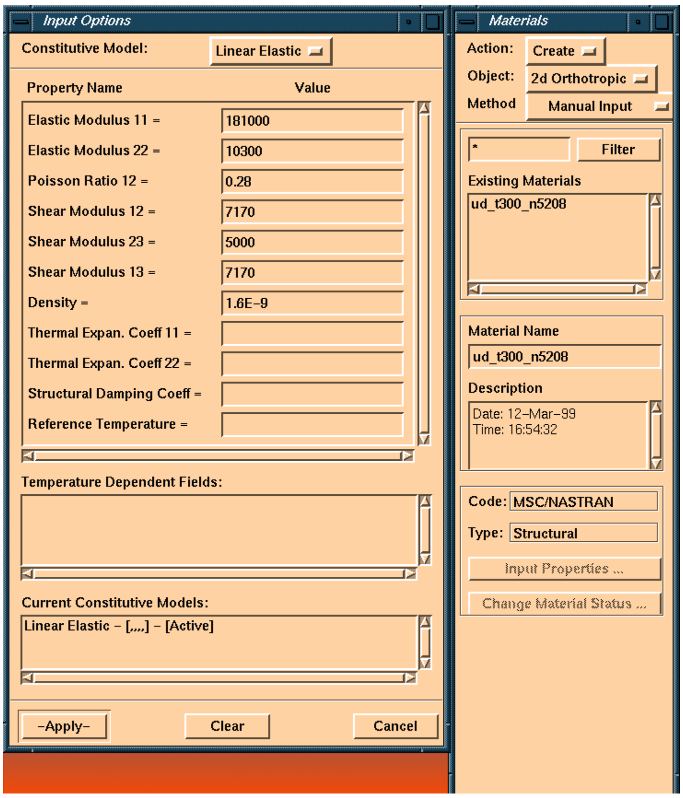
Until now this exercise has been quite straight forward, but the next step is to define the laminate.

5 Define lamina material properties

Now we input the orthotropic material properties valid for the lamina.

♦ Materials
Action : Create
Object : 2d Orthotropic
Method : Manual Input
Material Name: enter "ud_t300_n5208"
Input Properties...
Constitutive Model: Linear Elastic
Elastic Modulus 11: enter "181000"
Elastic Modulus 22: enter "10300"
Poisson Ratio 12: enter "0.28"
Shear Modulus 12: enter "7170"
Shear Modulus 23: enter "5000"
Shear Modulus 13: enter "7170"
Density: enter "1.6E-09"
Apply
Cancel

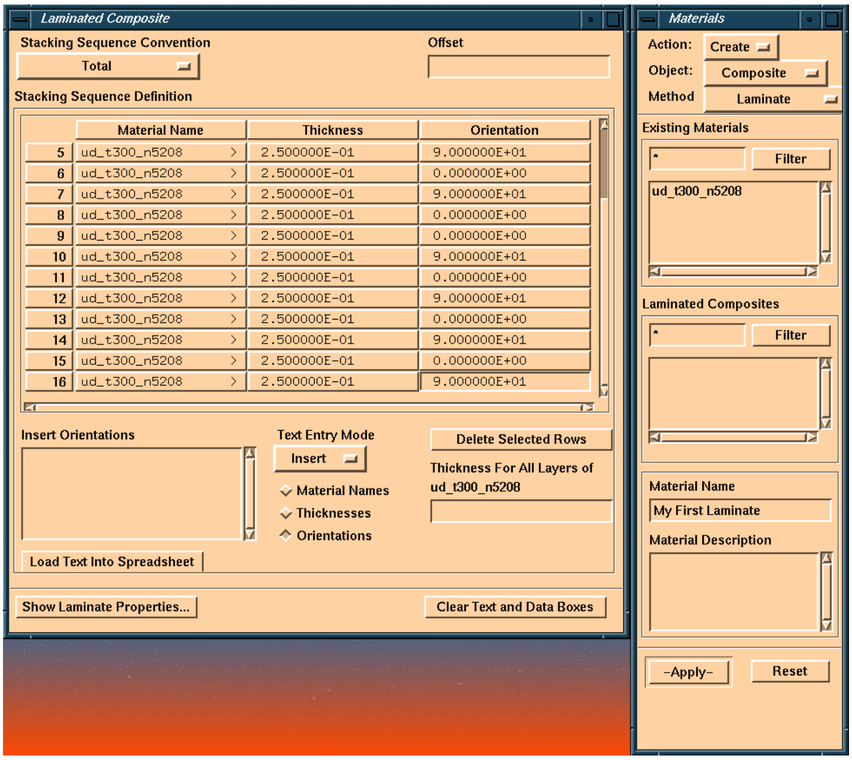
We will also use this material in later exercises, so a session file that makes this material is included in the work files for this course, named "materials.ses".



6 Define laminate properties

Now we build the laminate out of the lamina we just defined. Take notice of how the laminate is defined in the spreadsheet.

Action : Create
Object : Composite
Method : Laminate
Material Name : enter "My first Laminate"
Text Entry Mode : Insert, Material Names
Enter "16(ud_t300_n5208)" in the input window
Press "Load Text Into Spreadsheet"
Text Entry Mode : Overwrite Thicknesses
Enter "16(0.25)" in the input window
Press "Load Text Into Spreadsheet"
Text Entry Mode : Overwrite Orientations
Enter "4(90/0)" in the input window
Press "Load Text Into Spreadsheet"
Text Entry Mode : Overwrite Orientations
Enter "4(0/90)" in the input window



Press “Load Text Into Spreadsheet”
Apply

7 Define element properties

The composite laminate can now be associated to the model.

♦ Properties
Action : Create
Object : 2D
Type : Shell
Property name : enter “Shell Property”
Option 1: Laminate
Option 2 : Standard Formulation
Input Properties...
Material Name: select “My_First_Laminate” from list below
Material Orientation : Vector, Coord 0.1
Ok
Application Region : pick Surface 1

Add
Apply

Now we have related the fiber direction to the global coordinate x-axis. Half of our fibres are rotated 90 degrees related to this direction.

8 Verify Laminate Directions

It is a good rule to check your model before an analysis is run, especially when dealing with laminates. Remember that layer 1 is at the bottom of the elements. First we need to check the element normals, to determine what direction is up.

♦ Finite Elements
Action : Verify
Object : Element
Test: Normals
Display Control : Draw Normal Vectors
Apply

Verify that all vectors are pointing in the positive z-axis direction

♦ Properties
Action: Show
Existing Properties: Select "Orientation Angle"
Display Method: Select "Vector Plot"
Apply

We now see the initial reference direction. Note that we cannot see the individual fibre directions. We can only check them in the laminate spreadsheet. You can also select to show the thickness, but this is of little interest in this case.

9 Set up and run the analysis

♦ Analysis
Action : Analyse
Object: Entire Model
Type : Full Run
Subcase Create ...
Available Subcases : select “Default”
Output Requests...
Form Type: Advanced
Output Requests: select “Stress”
Composite Plate Opt : Ply & Elem. Stresses
Modify
Ok
Apply
Cancel
Apply

The analysis should run through in a few seconds time.

10 Read in MSC/Nastran analysis results

Action : Read Output2
Object : Result Entities
Method : Translate
Select Result File...
Selected Results File : pick “flatplate.op2”
Ok
Apply

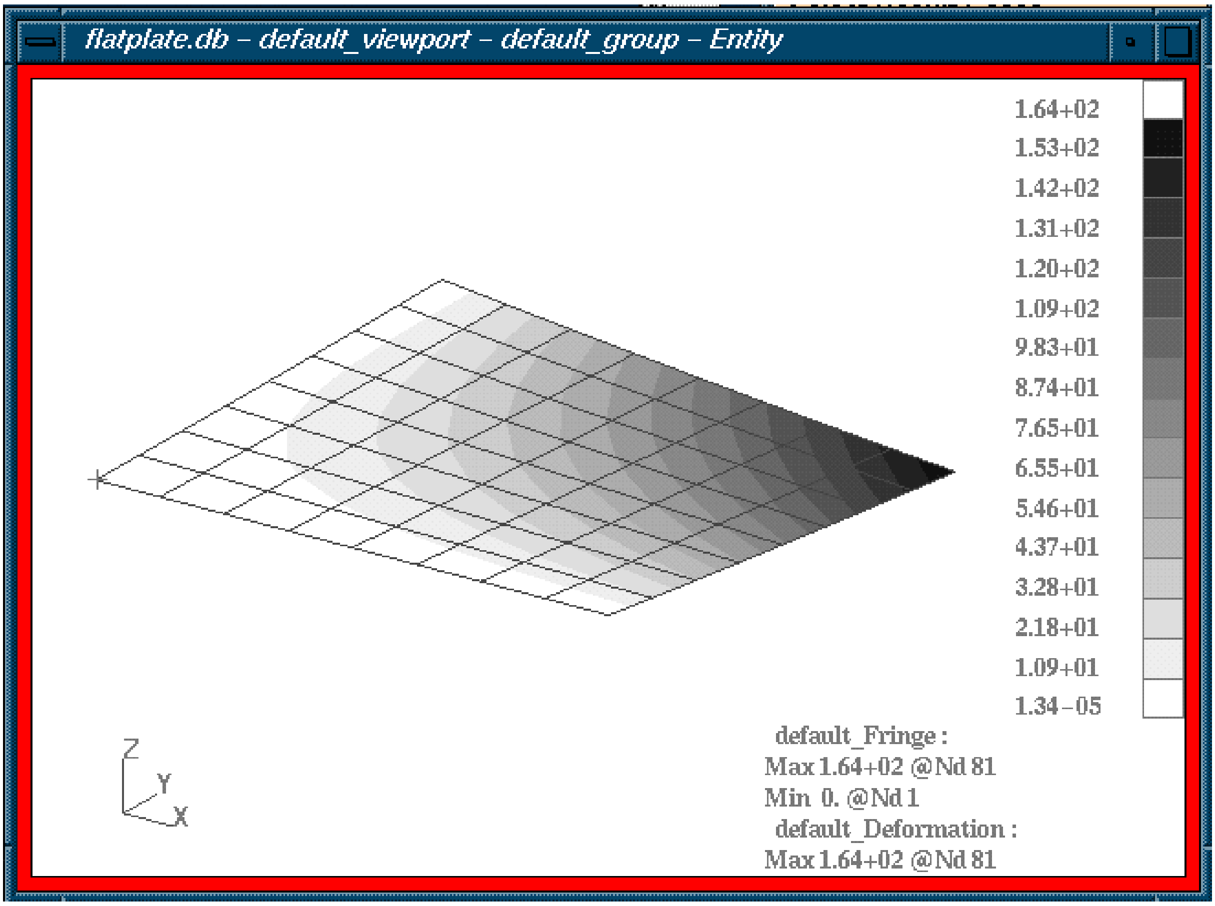
10 View the Results

First we want to look at the stresses in one of the layers. For illustrations sake, we choose layer 12, and plot stresses in the x-direction.

♦ Results
Select Result Case : pick "Default, Static Subcase"
Select Fringe Result : pick "Stress Tensor"
Position : select "Layer 12"
Close *(the layer selection menu)*
Option : Maximum
Close
Quantity: select "X Component"
Select Deformation Result: select "Displacements, Translational"
Apply

We also want to investigate the deflection of the plate

Select Fringe Result : Displacements, Translational
Apply



11 EXTRA, If time

Another laminate builder tool is situated in the Utilities menu, Materials, Laminate Builder tool. Check it out.

If you have any questions, please do not hesitate to ask!

Do not remove this exercise, it will be used later!

LESSON 2

Failure Criteria for Flat Plate

What you will learn in this exercise

11 How to Define a Composite Failure Criterion
12 Set up a Margin of Safety analysis
13 Postprosess Margin of Safety results
14 Edit existing Composite

Additional Material information

Material allowables are needed for the Margin of Safety Analysis, the following values can be used for the material:

Tension Stress Limit 11	1500
Tension Stress Limit 22	40
Compress Stress Limit 11	1500
Compress Stress Limit 22	246
Shear Stress Limit	68
Interaction Term	-0.5
Bonding Shear Stress	50

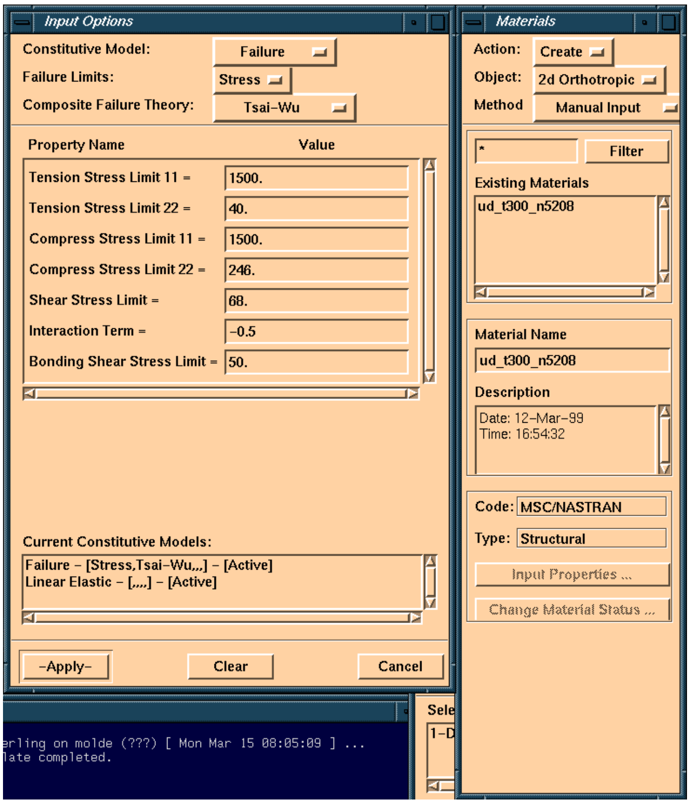
1 Open the last exercise database

File, Open...
Existing Database Name : select "flatplate.db"
Ok

2 Define Constitutive Failure Model

♦ Materials
Action : Create
Object : 2d Orthotropic
Method : Manual Input
Material Name : select "ud_t300_n5208"
Constitutive Model : Failure
Failure Limits : Stress
Composite Failure Theory : Tsai-Wu
Tension Stress Limit 11 : enter "1500"
Tension Stress Limit 22 : enter "40"
Compress Stress Limit 11 : enter "1500"
Compress Stress Limit 22 : enter "246"

Shear Stress Limit : enter “68”
Interaction Term : enter “-0.5”
Bonding Shear Stress Limit : enter “50”
Apply
Cancel



3 Do the failure calculation

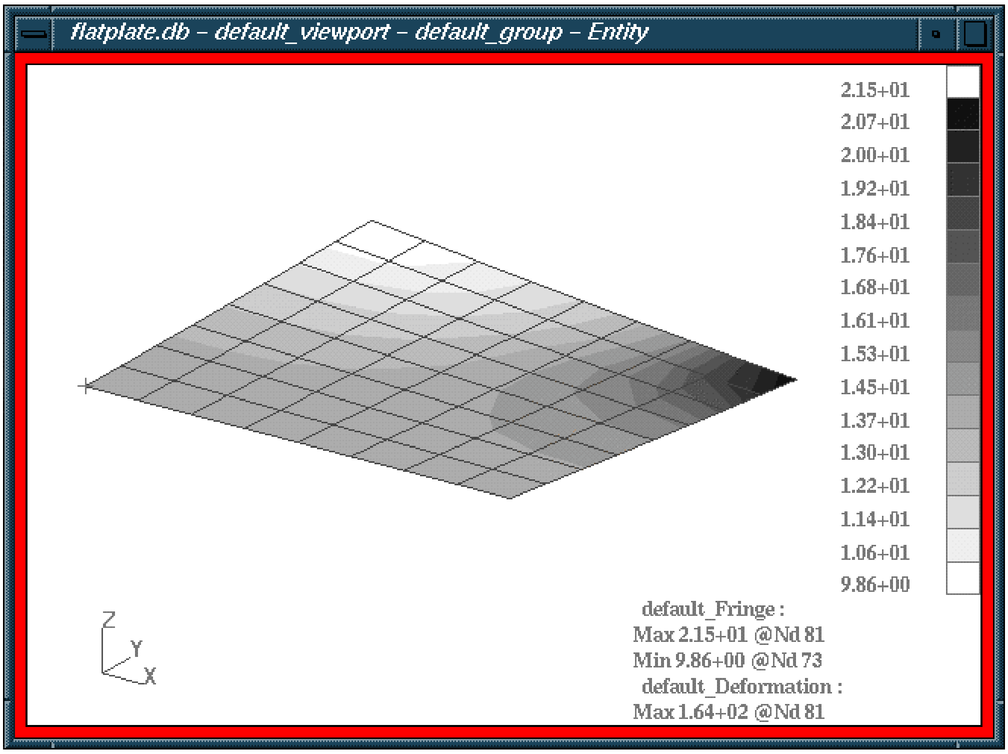
Tools, LAMINATE MODELLER...
New Layup File...
Ok (Accept the default file name)
Action : Create
Object : LM_Results
Method : Failure Calc
Select Loadcase : pick "1- Default"
Select Subcase : pick "1- Static Subcase"
Select Layered Result : pick "3.1- Stress Tensor"
Select Area : select "Elm 1:64"
Criterion : Tsai-Wu
Material Allowables...
Ok (accept default values)
Apply

Take notice of that instead of defining the Constitutive Failure Model under the Material Menu in Patran, you could fill in the same values in the above menu in MSC/Laminate Modeler.

When hitting Apply, the calculation is done immediately and the results are put into the result database.

4 View Failure Results

♦ Results
Select Result Cases : select "Default, Static Subcase"
Select Fringe Result : select "LM_Marg_Saf, ,...."
Apply

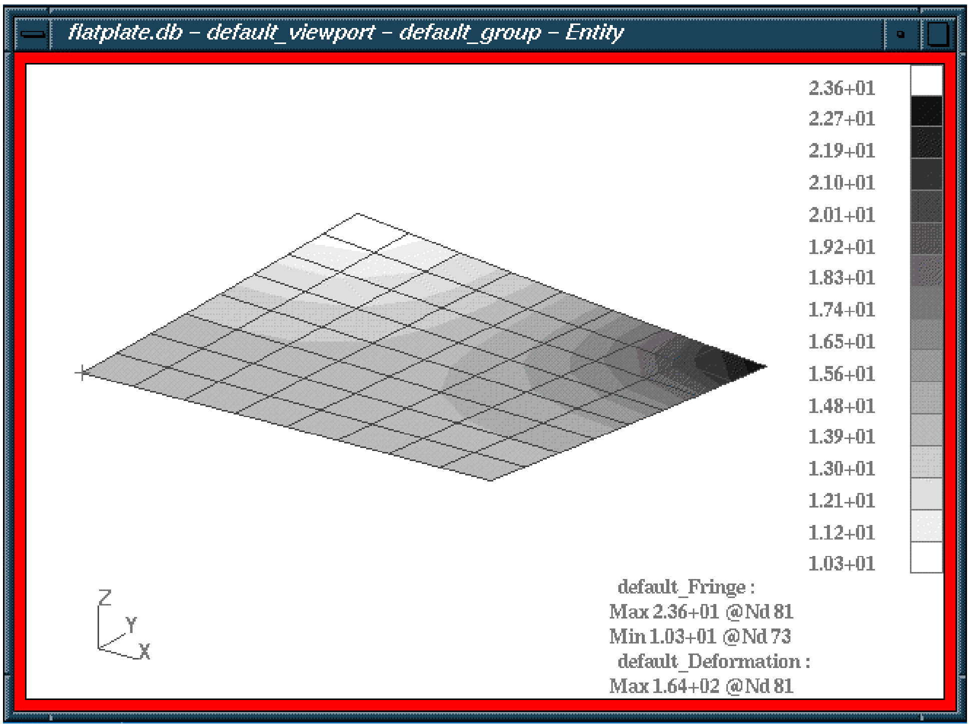


We do have a factor of 10 in safety according to this

5 Alter Failure Model

Now let's repeat step 3 and 4 with other failure models
Tools, LAMINATE MODELLER...
Open Layup File...
Layup file Name: select "flatplate.Layup"

Ok
Action : Create
Object : LM_Results
Method : Failure Calc
Select Loadcase : pick "1- Default"
Select Subcase : pick "1- Static Subcase"
Select Layered Result : pick "3.1- Stress Tensor"
Select Area : select "Elm 1:64"
Criterion : Maximum
Material Allowables...
Ok (accept default values)
Name: maximum
Apply



The different theories should yield:

Maximum	*10.3*
Tsai-Wu	*9.86*
Hill	*10.3*
Hoffman	*9.85*
Hankinson	*10.0*
Cowin	*10.0*

6 Modify the laminate

Try modifying your model to optimize its strength at the same time as we reduce its weight..

Suggestion: Delete ply number 15 and 16. Reorder the remaining plies as follows: 2 (45 / 90 / 0 / 45 / 90 / 0 / 45). Re-run the analysis. The new deflection is now down from 164 to 116 millimetres at the same time as we reduced its weight by 12.5%.

If you have any questions, please do not hesitate to ask!

LESSON 3

Making Plies and Layup on a Fairing Component

What you will learn

15 Make plies in Laminate Modeler
16 Make a layup in Laminate Modeler
17 Get some experience on a more realistic model

1 Make the geometry

Open a new database fairing.db
Generate the goemetry and finite elements by playing the session file "fairing_geom.ses"

2 Apply the loads and boundary conditions

Enter displacement restriction one <0,0,0> on edge "Surface 7.3"
Enter displacement restriction two <0,,> on edge "Surface 7.1"
Create pressure load (2D) with Top pressure 0.1 on "Surface 1:6"

3 Enter the material properties

Define the ply material properties by playing the session file "materials.ses"
Enter the Laminate Modeler and open a new layup file

Define the laminate modeler material from "ud_t300_n5208" with thickness 0.25, all other settings default.

4 Create the plies

Set view angles to –100,-10,40 (Viewing, Angles…)
Create a ply with

18 Application Region: Surface 1 3 2 7
19 Starting point: Node 75
20 Reference Direction: Along fairing, e.g. from Node 75 to Node 108
21 Reference Angle: 0

Repeat for reference angle 45 degrees

Make the 8 next plies as follows, keep the starting point for all, see the below figure for the correct view angles.

Keep the starting point at node 75

Ply #	Reference Angle	Application Region	View Vectors
3	0	Surface 7 5 6 4	(-100,0,20)
4	45	“	
5	0	Surface 3 5	(80,-5,20)
6	45	“	
7	0	Surface 2 3 5 6	
	1.8 45		“
9	0	Surface 1 2 3 4 5 6	
10	45	”	

5 Create the layup
Define the layup by stacking the plies as shown in the table below:

Number	Ply stacking sequence
1-8	1 / 2 / 1 / 2 / 1 / 2 / 1 / 2
9-16	3 / 4 / 3 / 4 / 3 / 4 / 3 / 4
17-24	5 / 6 / 5 / 6 / 5 / 6 / 5 / 6
25-32	7 / 8 / 7 / 8 / 7 / 8 / 7 / 8
33-40	9 / 10 / 9 / 10 / 9 / 10 / 9 / 10

You should now have 40 layers of plies

Use no offset, standard element type, and default tolerance

When hitting apply, thre program informes you that it created 8 laminate materials and 21 property regions.

6 Check you model before solving

Check the plies (fiber direction and application region) using the Laminate Modeler

Action: Show
Object: Ply
Method: Graphics
LM_Ply_Data: *Select one ply*
Apply

Action: Show
Object: LM_Layup
Method: Exploded
Select LM_Plys: *Select several plies or even all*
Apply

7 Create the analysis input deck

Remember to request ply stress as output

8 Analyse and read the results

Run the analysis and read the result
Verify that the laminate materials and properties have been created

9 Postprosess the results
Take a look at the deformations, the different layer stresses etc.

If you have any difficulty with this exercise, examine or play the session file "fairing.ses" after opening a new database.

LESSON 4

Draping a Doubly Curved Surface

What you will learn from this exercise

22 How to make a drape

23 How different inputs, such as starting point and reference angle, affect shear

24 What is meant by step length

4.1 Make the model

Open a new database called “Drapeintro.db”
Play session file “drapeintro_geom.ses” to create geometry and mesh

2 Create the materials

Play session file “materials.ses” to create materials
Create a scissor drape material with default values from LM_Material ud_t300_n5208

3 Create the first ply

Set angles to “0 0 0” to obtain default view
Create a scissor drape ply with start point in the middle of the surface, the reference direction in the global X-direction, reference angle zero, covering the entire surface
Note that shear strain increases away from the starting point and the principal axes, which are geodesic lines by default
Use the graphics control form to plot and hide the draped pattern, element angles, flat pattern, maximum strain value and starting point.
Note the maximum shear angle of 14 degrees.

4 Create the second ply

Change the reference angle to 30 degrees and create another scissor drape ply
Note that this results in lower shear, 9 degrees.
Also, change in ply orientations is not necessarily equal to 30 degrees on individual elements, as a consequence of the nonlinearity of the draping process

5 Create the third ply

Change the starting point to the middle of the lower edge (e.g. node 11) and create another scissor draped ply
Note the increased shear due to greater distances from the start point, maximum 30 degrees.

6 Investigate the function of step length

Change the step length to Implicit 2 (set in Additional Controls, Geometry) and create another similar scissor draped ply.
The default step length is calculated as a function of the area of the surface of the model. This multiplies the default step length by 2

Change the step length to Explicit, 2 (set in Additional Controls, Geometry) and create another scissor draped ply
This changes the step length to exactly 2 units

Note that for small step lengths, the pattern may not cover the entire surface because the number of possible steps is limited due to limited (not unlimited!) computer resources.

If you have any difficulty with this exercise, examine or play the session file "drapeintro.ses" after opening a new database.

LESSON 5

Draping with splits

What you will learn from this exercise

25 How splits are introduced in a ply

26 Capabilities and limitations using splits

1 Create the geometry

In the Goemetry menu

Action: Create
Object: Curve
Method: 2DArc 2Angles
Enable Patran2 convention
Curves pr. arc: enter ”3”
Radius: enter ”500”
Start Angle: enter ”0”
End Angle: enter ”90”

Action: Create
Object: Surface
Method: Revolve
Enable Patran2 convention
Axis: Coord 0.2
Total Angle: enter ”360”
Surface pr Curve: enter “8”
Curve List: Pick curve 1:3

2 Create the mesh

In the Finite Elements Menu

Action: Create
Object: Mesh
Methods: Surface
Global Edge Length: enter “50”
Element Topology: select “Quad4”
Mesher: Select “Paver”
Surface List: select “Surface 1:24”

Do not bother with the mesh quality for this exercise

Action: Equivalence

Object: Nodes
Method: Tolerance Qube
Tolerance Cube: enter “0.01”

3 Create the materials
Play session file “materials.ses” to input the material
Create a LM_Material based on “ud_t300-n5208” with default values

4 Create the plies
View the model from above (Angles 90,0,0)
Set Step Length to Explicit 25. Create A Scissor Drape with starting point in the centre covering all the surfaces. Reference Direction along the global X-axis, Reference Angle 0.

Note that the maximum shear angle is 50 degrees! This is not usable for any real world application.
Do another similar drape with Reference Angle 45 degrees. Note no changes in shear (of course).

5 Introduce Cuts
Introduce four cuts. This is done under Additional Controls, Boundaries, Define Splits.
Add the lover four curves where the shear is excessive to the split application region.
Remake the drape (keep 45 degrees Ref angle)
Maximum shear is now reduced to 43 degrees. This is better than our last try, but still too much.

6 Refine Cuts

Extend the cuts by adding the next four curves and redrape using the same inputs. The maximum shear angle is now 35 degrees. This is still not good, but we will keep it for this exercise. In fact the only solution to our problem is to drape smaller sections of the sphere.

7 Make a new drape

Now redefine the cuts to the other 8 lines in a similar fashion and redrape with an Reference Angle of 0 degrees. This drape is exactly the same other than being rotated 45 degrees about the global y-axis

8 Make the layup

Note that you can use the Show, LM_Ply, Graphics to browse through the existing plies to see which to choose when making the layup. Create a layup with the last two plies explained in this exercise. Stack them in a A/B/A/B fashion until the total number of layers is 16.

9 Make boundary- and loading conditions

Create an displacement restriction of “<,0,>” for the translations applicable for all the lines in the bottom of the sphere.
Create another displacement restriction of “<0,0,0> for the translations and “<0,0,0>” for the rotations for on point anywhere on the sphere lower edge.
Add an internal pressure on all surfaces of 1 (MPa)

10 Do the analysis

Run the analysis and read the results

11 Postprosess

Look at the displacements in the y-direction.
Explain what you see!

Erling Wiig

LESSON 6

Springback analysis of a jet engine cowling

What you will learn from this exercise

27 Importance of springback, and the need to correct for it
28 Thermal induced deflections on a composite model

1 Make the model

Create a new database springback.db
Play sesion file “springback_geom.ses” to generate geometry

2 Make the material

Play session file “materials.ses” to create materials
Create a laminate material using “ud_t300_n5208“ properties and a thickness of 0.25 (mm)

3 Make the plies

Create a ply with origin [1250 0 1250] and initial vector along the global x-axis
Create similar plies, but with different reference angles: 45, 90 and 135 degrees.

4 Make the layup

Create a layup with the 0,45,90,135 plies stacked six times (giving a 24 ply layup)

5 Spesify the boundary conditions

Fix lower edge in translation and rotation
Create a nodal temperature load of –55 over the whole body

6 Analyse the model

Analyse the model, and read in the results

You now simulates what happens when the material cools from curing temperature to room temperature. Because of the orthotropic thermal and mechanical properties the model distorts. This yields a different model shape than the one we made in the mould. Abviously we will have to correct for this in a way such as the deformed shape is the one we want!

7 Verify the results

Plot the thermal induced deformation of the model

If you have any difficulty with this exercise, examine or play the session file "springback.ses" after opening a new database.