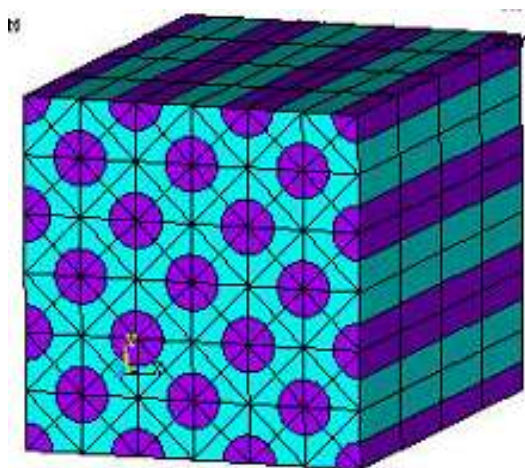
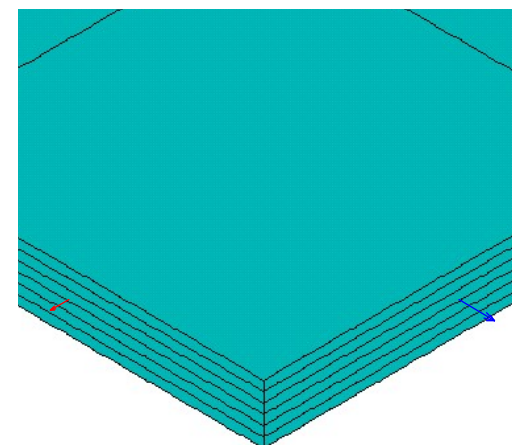


# **Obliczenia numeryczne kompozytów**

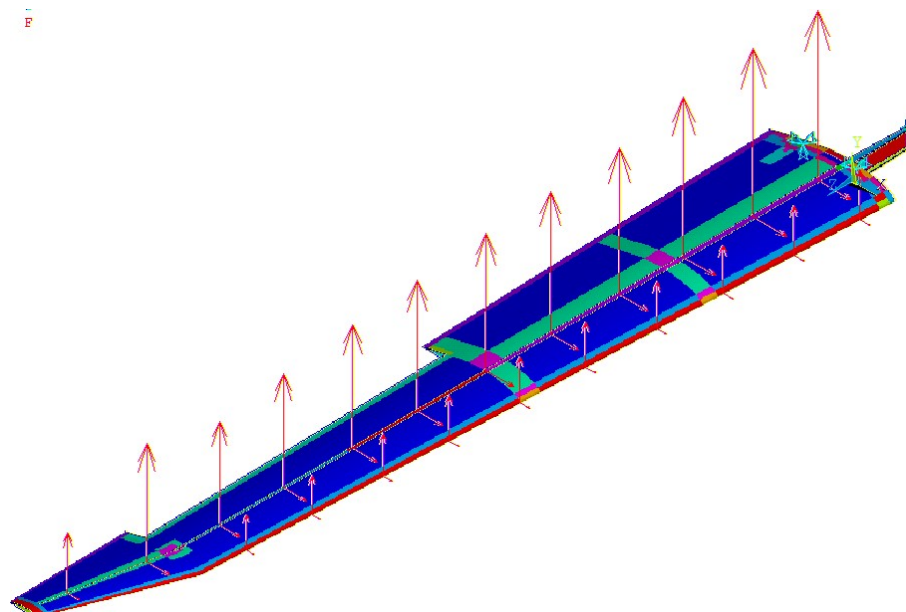
# Różne poziomy analizy konstrukcji kompozytowej



Model mikromechaniczny kompozytu

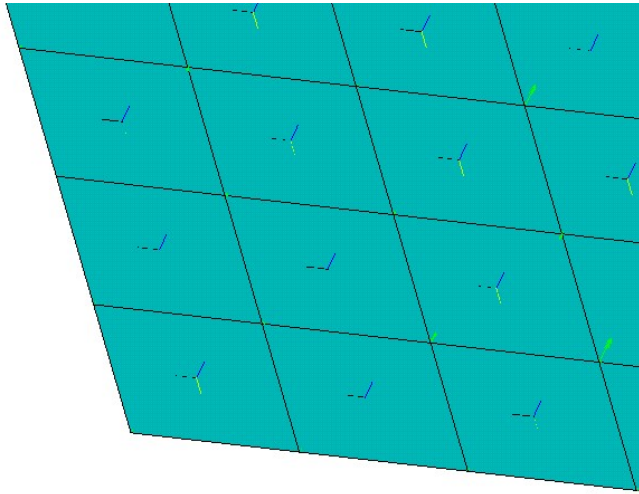


Model powłoki wielowarstwowej

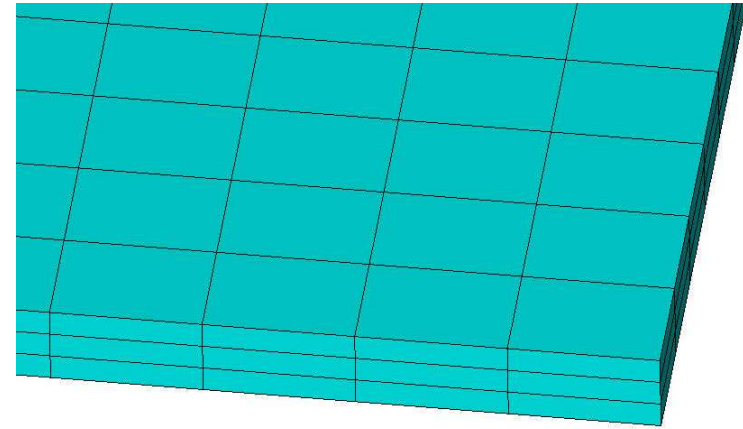


Model konstrukcji kompozytowej

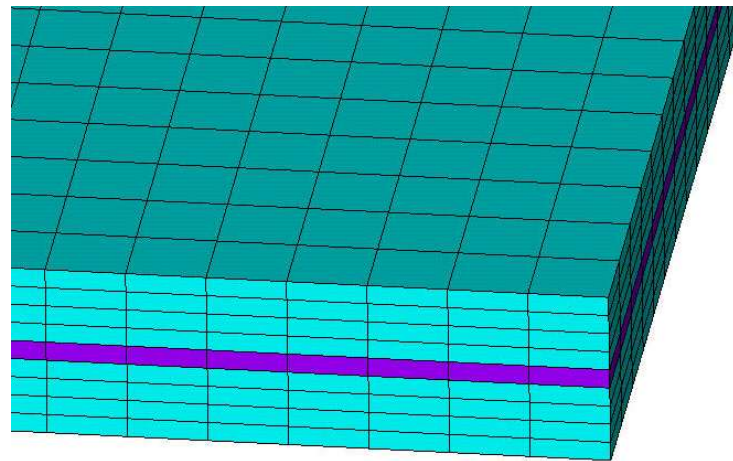
# Rodzaje elementów



Powłokowe (SHELL 181, SHELL 281)

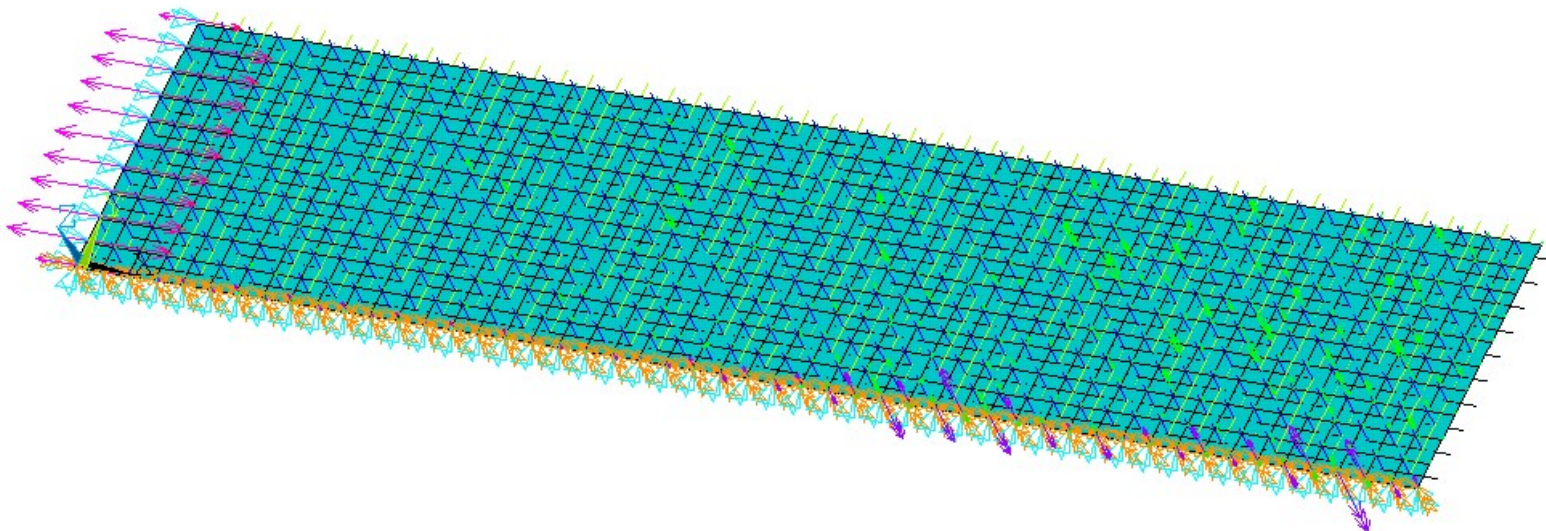
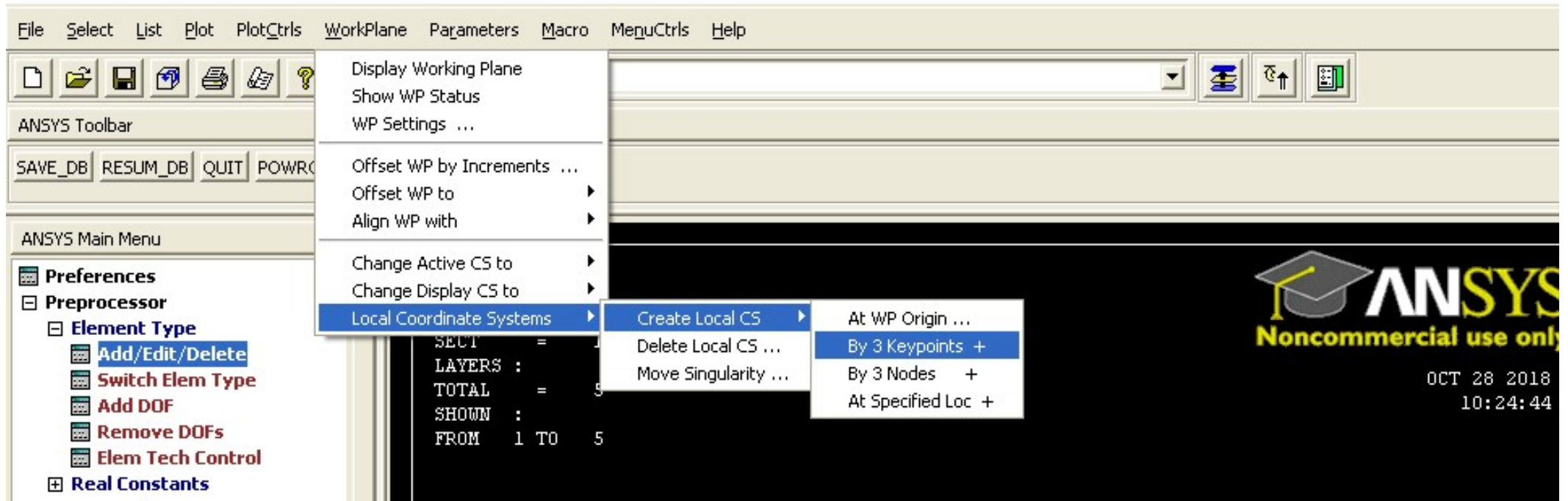


Bryłowo-powłokowe (SOLSH 190)



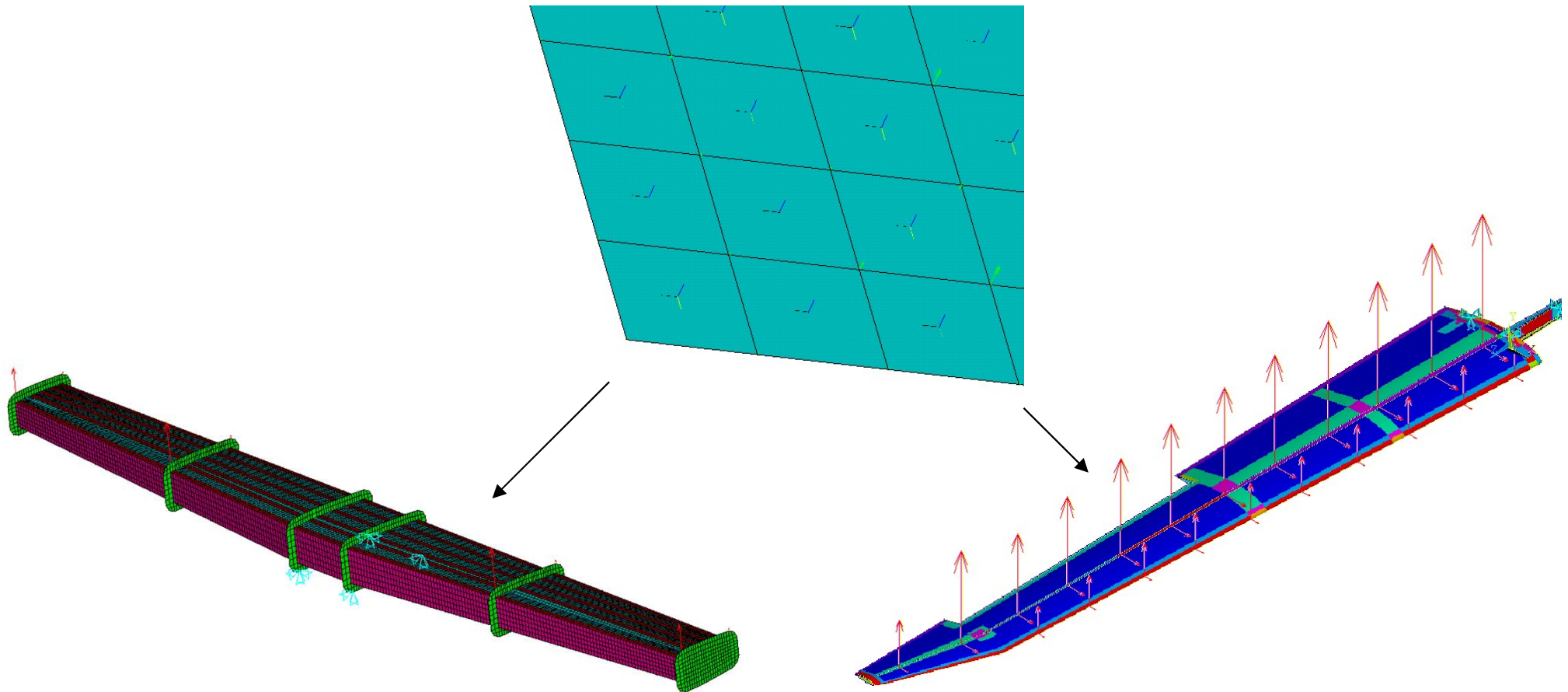
Bryłowe (SOLID 185, SOLID 186)

# Elementowe układy współrzędnych



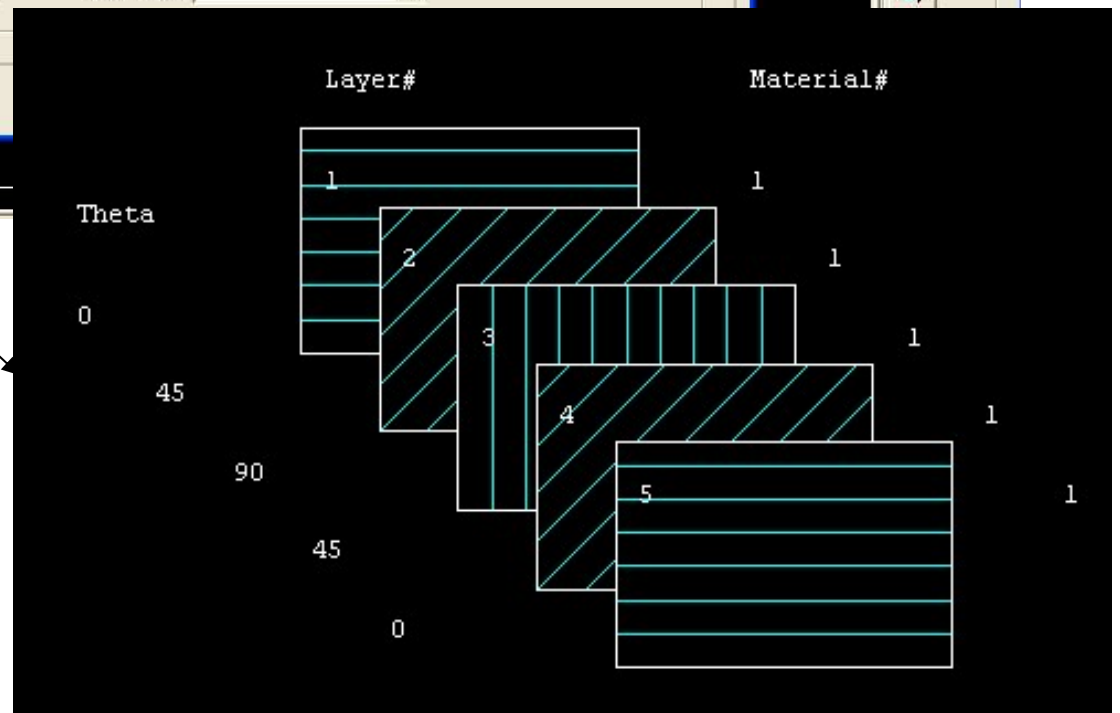
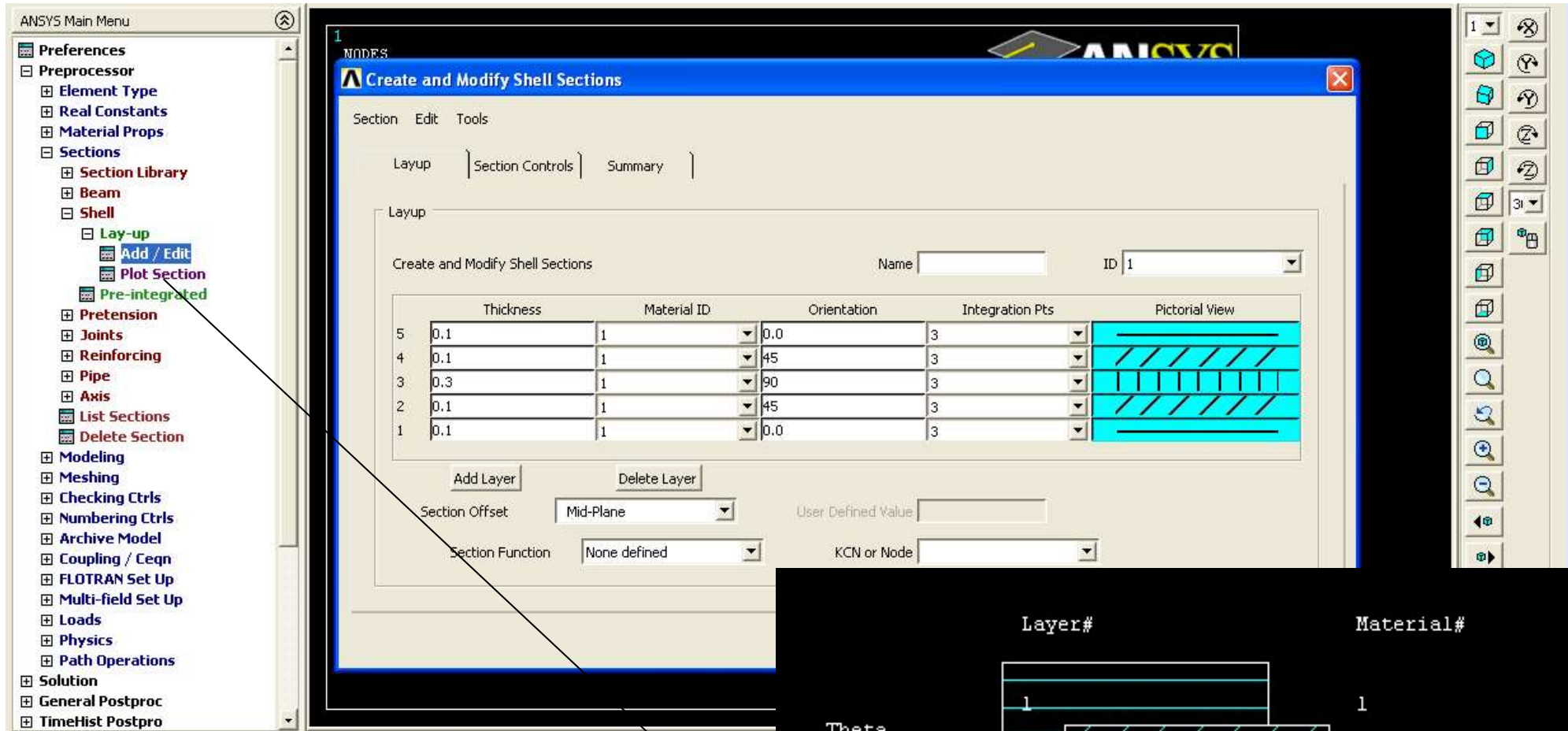
Kierunki warstw są orientowane zgodnie z elementowymi układami współrzędnych.

# Elementy powłokowe



- Jedyna metoda obliczeń dużych struktur
- Nie odwzorowują zjawisk w kierunku grubości kompozytu
- Nie pozwalają na modelowanie zjawisk lokalnych

# Sections



# Offset

ANSYS Main Menu

- Preferences
- Preprocessor
  - Element Type
  - Real Constants
  - Material Props
  - Sections
    - Section Library
    - Beam
    - Shell
      - Lay-up
        - Add / Edit
        - Plot Section
        - Pre-integrated
    - Pretension
    - Joints
    - Reinforcing
    - Pipe
    - Axis
    - List Sections
    - Delete Section
  - Modeling
  - Meshing
  - Checking Ctrl
  - Numbering Ctrl
  - Archive Model
  - Coupling / Ceqn
  - FLOTRAN Set Up
  - Multi-field Set Up
  - Loads
  - Physics
  - Path Operations
- Solution
- General Postproc
- TimeHist Postpro

1 LAYER STACKING

### Create and Modify Shell Sections

Section Edit Tools

Layup | Section Controls | Summary

Layup

Create and Modify Shell Sections Name: ID: 1

	Thickness	Material ID	Orientation	Integration Pts	Pictorial View
5	0.1	1	0	3	
4	0.1	1	45	3	
3	0.3	1	90	3	
2	0.1	1	45	3	
1	0.1	1	0	3	

Add Layer Delete Layer

Section Offset: Top-Plane (selected), Mid-Plane, Bottom-Plane

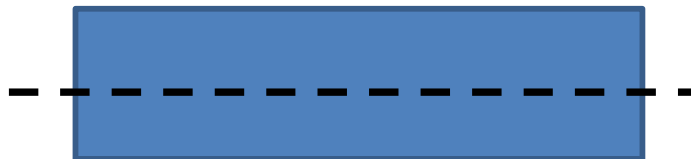
Section Function: Top-Plane (selected), Bottom-Plane, User Defined Function

User Defined Value: 0.0

KCN or Node: Global Cartesian

OK Cancel Help

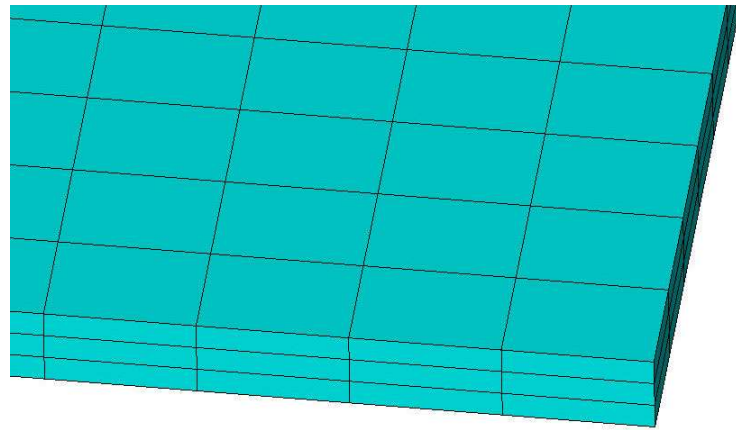
Middle plane



Top plane



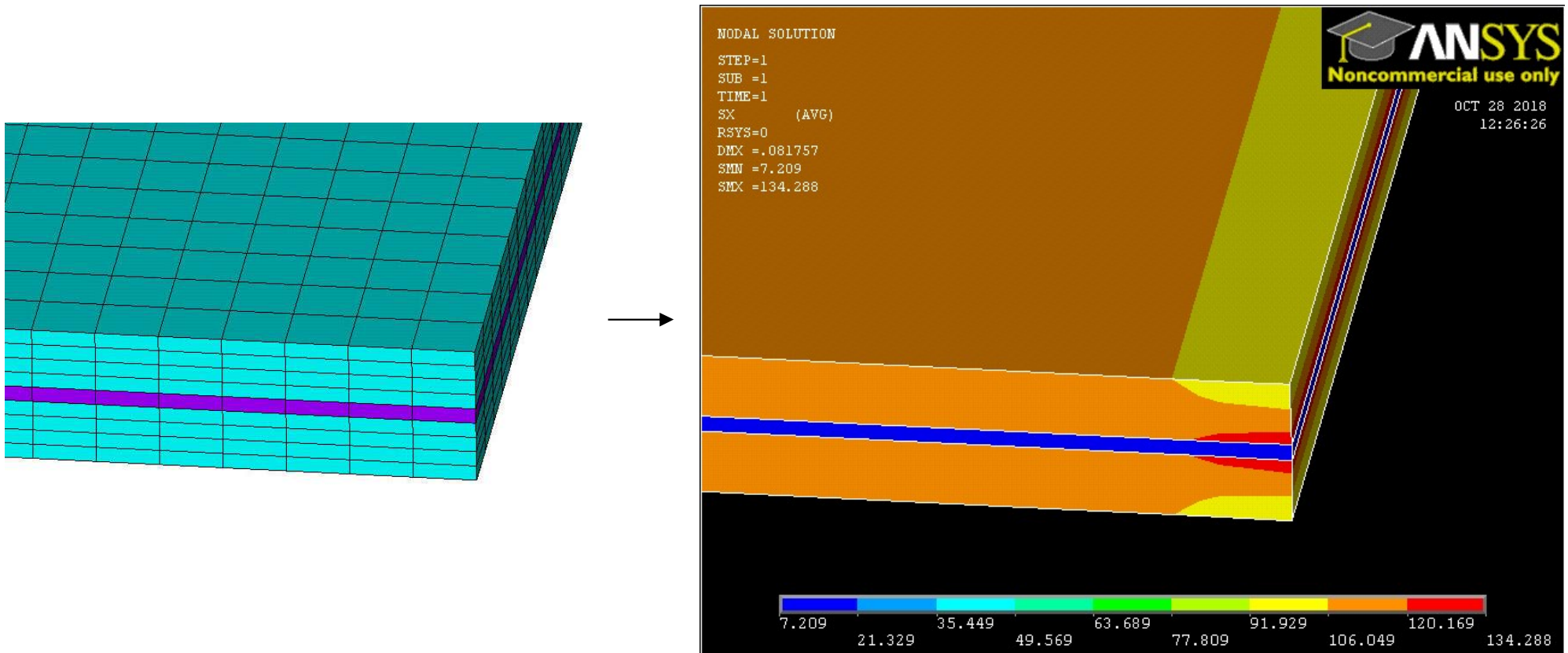
# Elementy bryłowo-powłokowe



- Pozwalają na modelowanie kilku warstw materiału jedną warstwą elementów.
- Używamy, gdy nie interesują nas naprężenia, ale np. deformacje.

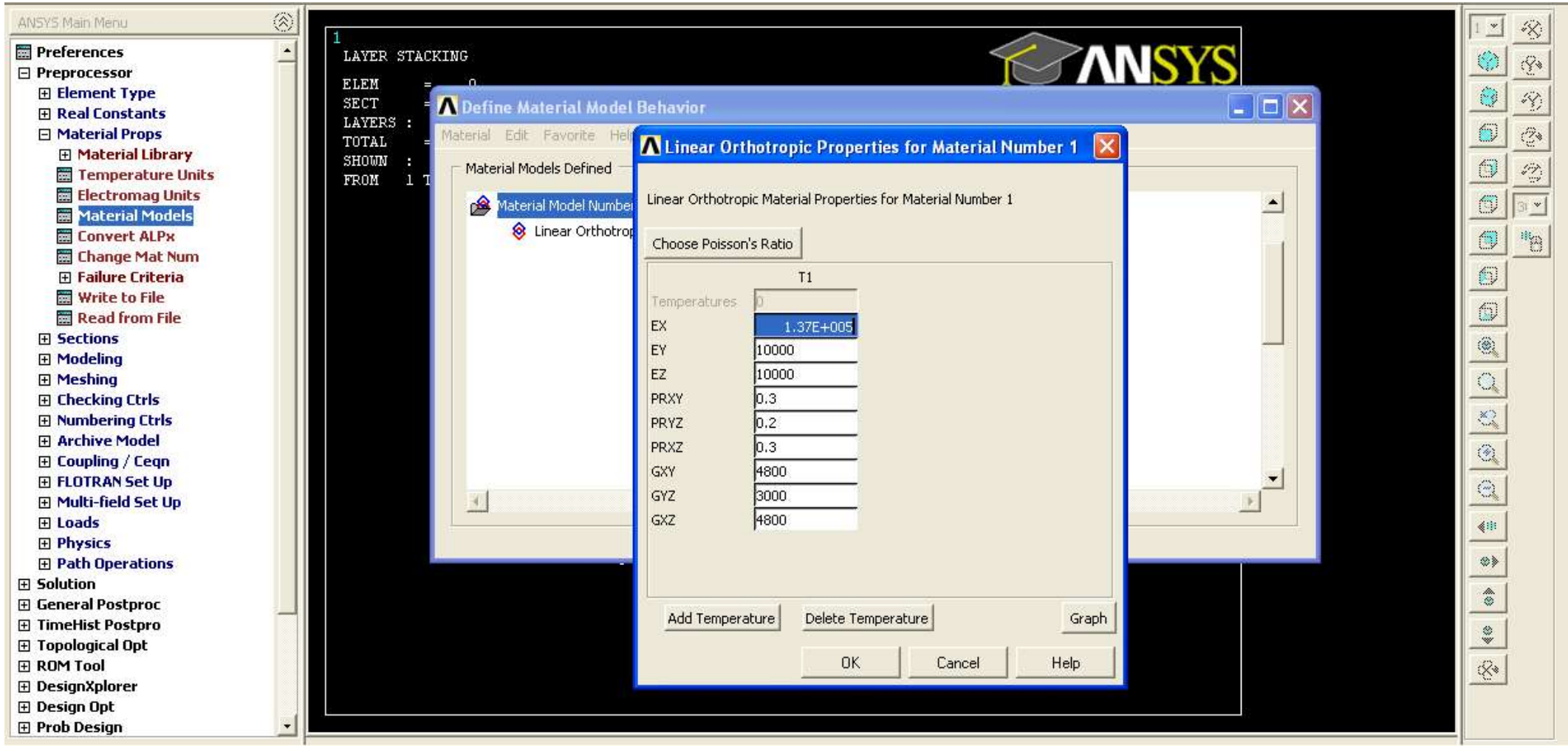


# Elementy bryłowe



- Każda warstwa materiału powinna być modelowana oddzielną warstwą elementów – duży model.
- Dobre odwzorowanie naprężeń.

# Materiał ortotropowy



- Materiał ortotropowy jest zawsze liniowo-sprężysty.
- Zawsze trzeba podać 9 stałych materiałowych.
- Stałe materiałowe najwygodniej jest wstawiać w megapascalach.

# Dane materiałowe

	Kompozyt szklano-epoksydowy		Kompozyt węglowo-epoksydowy	
	Jednokierunkowy	Tkaninowy	Jednokierunkowy	Tkaninowy
$E_{11T}/E_{11C}$	43/42 [GPa]	20/17 [GPa]	130/115 [GPa]	70/60 [GPa]
$E_{22T}/E_{22C}$	8/10 [GPa]	19/16 [GPa]	9/10 [GPa]	65/55 [GPa]
$\nu_{12}$	0,28	0,2	0,25	0,15
$G_{12}$	4 [GPa]	4,2 [GPa]	4,4 [GPa]	5,5 [GPa]
$X_{T1}$	1100 [MPa]	600 [MPa]	2000 [MPa]	800 [MPa]
$X_{C1}$	900 [MPa]	550 [MPa]	1300 [MPa]	700 [MPa]
$X_{T2}$	35 [MPa]	550 [MPa]	80 [MPa]	750 [MPa]
$X_{C2}$	150 [MPa]	500 [MPa]	250 [MPa]	650 [MPa]
S	60 [MPa]	55 [MPa]	95 [MPa]	80 [MPa]

# Dane materiałowe

## Kompozyt jednokierunkowy

$$E_x > E_y = E_z$$

$$G_{xy} = G_{xz} > G_{yz}$$

$$v_{xy} = v_{xz} > v_{yz}$$

Nieznane:  $G_{yz}$  i  $v_{yz}$

## Kompozyt tkaninowy

$$E_x = E_y > E_z$$

$$G_{xy} > G_{xz} = G_{yz}$$

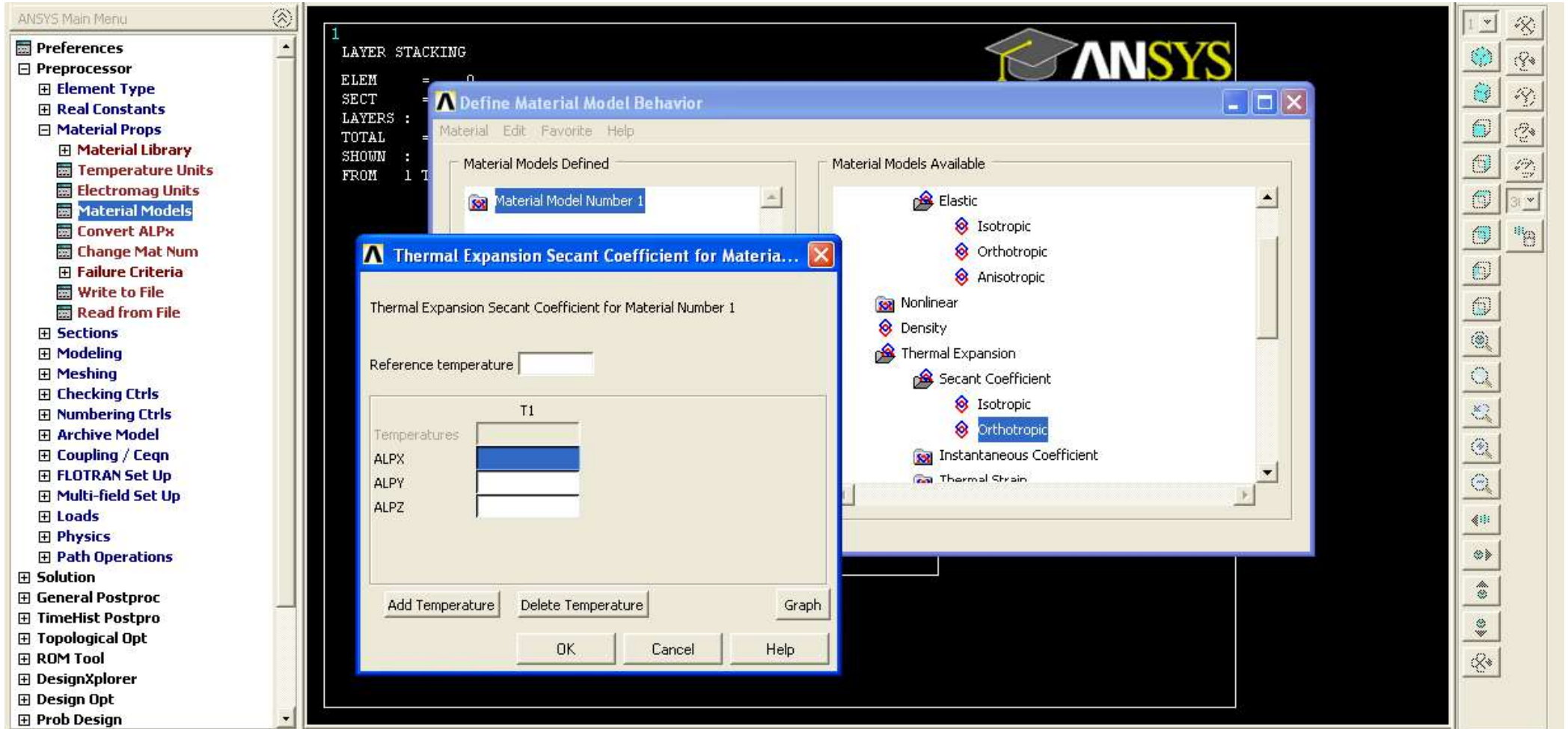
$$v_{xy} > v_{xz} = v_{yz}$$

Nieznane:  $E_z$ ,  $G_{xz} = G_{yz}$  i

$$v_{xz} = v_{yz}$$

**Analiza wrażliwości**

# Rozszerzalność cieplna



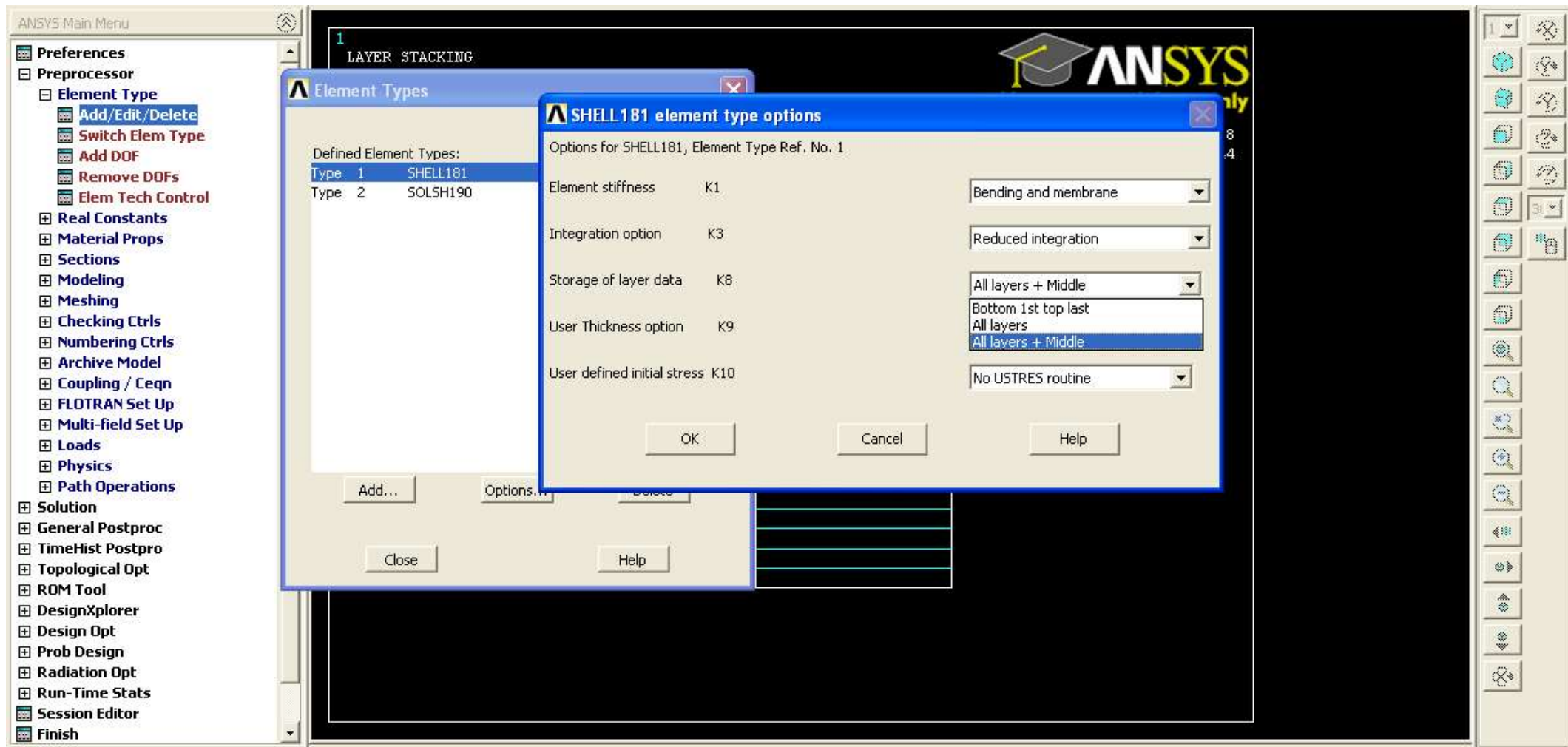
Kompozyt jednokierunkowy

$$\alpha_x < \alpha_y = \alpha_z$$

Kompozyt tkaninowy

$$\alpha_x = \alpha_y < \alpha_z$$

# Opracowanie wyników – elementy powłokowe



Dzięki zaznaczeniu odpowiedniej opcji, można uzyskać wyniki dla każdej z warstw.

# Opracowanie wyników – elementy powłokowe

ANSYS Main Menu

- Preferences
- Preprocessor
- Solution
- General Postproc
  - Data & File Opts
  - Results Summary
  - Read Results
  - Failure Criteria
  - Plot Results
  - List Results
  - Query Results
  - Options for Outp
  - Results Viewer
  - Write PGR File
  - Nodal Calcs
  - Element Table
  - Path Operations
  - Surface Operations
  - Load Case
  - Check Elem Shape
  - Write Results
  - ROM Operations
  - Submodeling
  - Fatigue
  - Safety Factor
  - Define/Modify
  - Nonlinear Diagnostics
  - Reset
  - Manual Rezoning
- TimeHist Postpro
- ROM Tool
- DesignXplorer
- Prob Design
- Radiation Out

SAVE\_DB RESUM\_DB QUIT POWRGRPH

1 NODAL SOLUTION  
STEP=1  
SUB =1  
TIME=1  
UZ  
RSYS=0  
DMX =11.0811  
SMN =-11.0809

Options for Output

Options for Output

[RSYS] Results coord system: As calculated

Local system reference no.:

[AVPRIN] Principal stress calcs

[AVRES] Avg rsults (pwr grph) for

Use interior data:  NO

[/EFACET] Facets/element edge: 1 facet/edge

[SHELL] Shell results are from: Top layer

[LAYER] Layer results are from:

Specified layer number: 3

[FORCE] Force results are: Total force

OK Cancel Help

-11.0809 -9.84972 -8.61851 -7.38729 -6.15608 -4.92486 -3.69365 -2.46243 -1.23122 0

Dzięki zaznaczeniu odpowiedniej opcji, można uzyskać wyniki dla każdej z warstw.

# Kryteria wytrzymałościowe

The image displays two overlapping windows from the ANSYS software interface. The top window is titled "Add/Edit Failure Criteria" and shows settings for material 1. The bottom window is titled "Contour Nodal Solution Data" and shows a list of items to be contoured, with "Failure Criteria" selected.

**ANSYS Main Menu**

- Preferences
- Preprocessor
- Solution
- General Postproc
  - Data & File Opts
  - Results Summary
  - Read Results
  - Failure Criteria
    - Temp Variation
    - Add/Edit
    - Delete
    - List
    - Criteria Check
  - Options for Outp
  - Results Viewer
  - Load Case
  - Check Elem Shape
  - Write Results
  - ROM Operations
  - Fatigue
  - Define/Modify
  - Manual Rezoning
- TimeHist Postpro
- Topological Opt
- ROM Tool
- DesignXploror
- Design Opt
- Prob Design
- Radiation Opt
- Run-Time Stats
- Session Editor
- Finish

**Add/Edit Failure Criteria**

[FC] Temperature-independent failure criteria for material 1

	X	Y	Z
Strain in Tension			
Strain in Compression			
	XY	YZ	XZ
Strain in Shear			
	X	Y	Z
Stress in Tension			
Stress in Compression			
	XY	YZ	XZ
Stress in Shear			
	XY	YZ	XZ
Stress Coupling Coefficients			

SAVE\_DB RESUM\_DB QUIT POWRGRPH

**ANSYS Main Menu**

- Preferences
- Preprocessor
- Solution
- General Postproc
  - Data & File Opts
  - Results Summary
  - Read Results
  - Failure Criteria
  - Plot Results
    - Deformed Shape
    - Contour Plot
      - Nodal Solu
      - Element Solu
      - Elem Table
      - Line Elem Res
    - Vector Plot
    - Plot Path Item
    - Concrete Plot
    - ThinFilm
  - List Results
  - Query Results
  - Options for Outp
  - Results Viewer
  - Write PGR File
  - Nodal Calcs
  - Element Table
  - Path Operations
  - Surface Operations
  - Load Case
  - Check Elem Shape
  - Write Results
  - ROM Operations
  - Submodeling

**Contour Nodal Solution Data**

Item to be contoured

- Elastic Strain
- Plastic Strain
- Creep Strain
- Thermal Strain
- Total Mechanical and Thermal Strain
- Swelling strain
- Energy
- Failure Criteria
  - Maximum Stress
  - Tsai-Wu Strength Index
  - Inverse of Tsai-Wu Strength Ratio Index
  - Maximum Strain
- Body Temperatures

Undisplaced shape key: Deformed shape only

Scale Factor: True Scale 1

Additional Options

OK Apply Cancel Help



# Struktury przekładkowe

ANSYS Main Menu

- Preferences
- Preprocessor
  - Element Type
  - Real Constants
  - Material Props
  - Sections
    - Section Library
    - Beam
    - Shell
      - Lay-up
        - Add / Edit
        - Plot Section
        - Pre-integrated
    - Pretension
    - Joints
    - Reinforcing
    - Pipe
    - Axis
    - List Sections
    - Delete Section
  - Modeling
  - Meshing
  - Checking Ctrl
  - Numbering Ctrl
  - Archive Model
  - Coupling / Ceqn
  - FLOTRAN Set Up
  - Multi-field Set Up
  - Loads
  - Physics
  - Path Operations
  - Solution
  - General Postproc

1  
ELEMENTS

## Create and Modify Shell Sections

Section Edit Tools

Layup | Section Controls | Summary |

Layup

Create and Modify Shell Sections

Name  ID 1

	Thickness	Material ID	Orientation	Integration Pts	Pictorial View
3	0.25	1	0	3	
2	10	2	0	3	
1	0.25	1	0	3	

Add Layer

Delete Layer

Section Offset Mid-Plane

User Defined Value

Section Function

KCN or Node Global Cartesian

OK

Cancel

Help