

6. Results Combination in Hexagonal Shell

Applicable CivilFEM Product: All CivilFEM Products

Level of Difficulty: Moderate

Interactive Time Required: 20 minutes

Discipline: Load combinations results

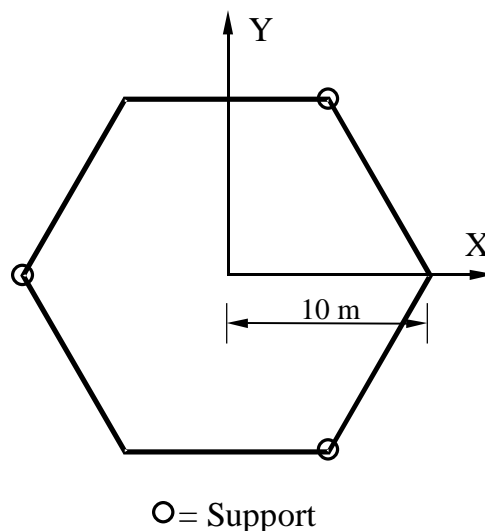
Analysis Type: Linear static

Element Type Used: SHELL181

CivilFEM Features Demonstrated: Combination Module

Problem Description

In an hexagonal shell supported on three alternate vertices and submitted to a uniform distributed load that can be place at any region of its area, we need to find the maximum deflection at each point of the shell and where the distributed load must be placed in order to produce maximum deflection at the free vertex.



■ Given

The structure geometry is shown in the previous figure. The following is a list of all input parameters:

Shell thickness:	0.5 m
Young Modulus:	1.0E7 kN/m ²
Load:	10.0 kN/m ²

■ Approach and Assumptions

A 3D analysis will be performed. We will use elastic shell elements.

■ Summary of Steps

Preprocessing

1. Specify title
2. Set units
3. Define element type
4. Define real constants
5. Define material
6. Define model geometry
7. Mesh
8. Save the database

Solution

9. Apply displacement constraints
10. Define and solve load states

Postprocessing

11. Define targets
12. Define combination rules
13. Carry out combinations
14. Read in results
15. Plot of the maximum deflection at each point of the shell
16. Calculation of the load distribution to obtain the maximum deflection at a vertex
17. Exit the ANSYS program

Interactive Step-by-Step Solution

■ Preprocessing

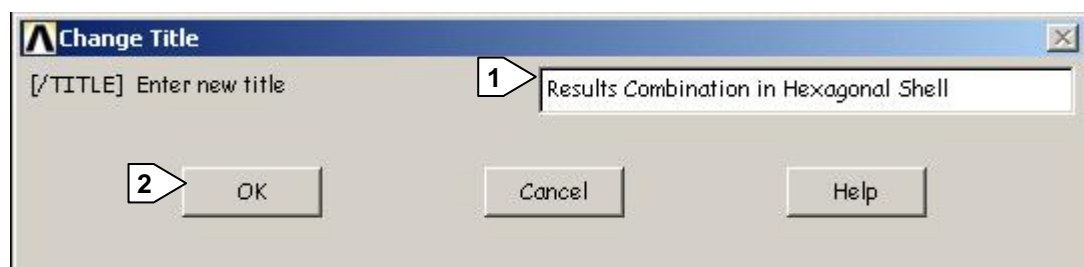
1. Specify title

Although this step is not required for a CivilFEM analysis, we recommend that you make it part of all your analysis.

Utility Menu: **File** → **Change title**

1 Enter the title: “Result combination in hexagonal shell”

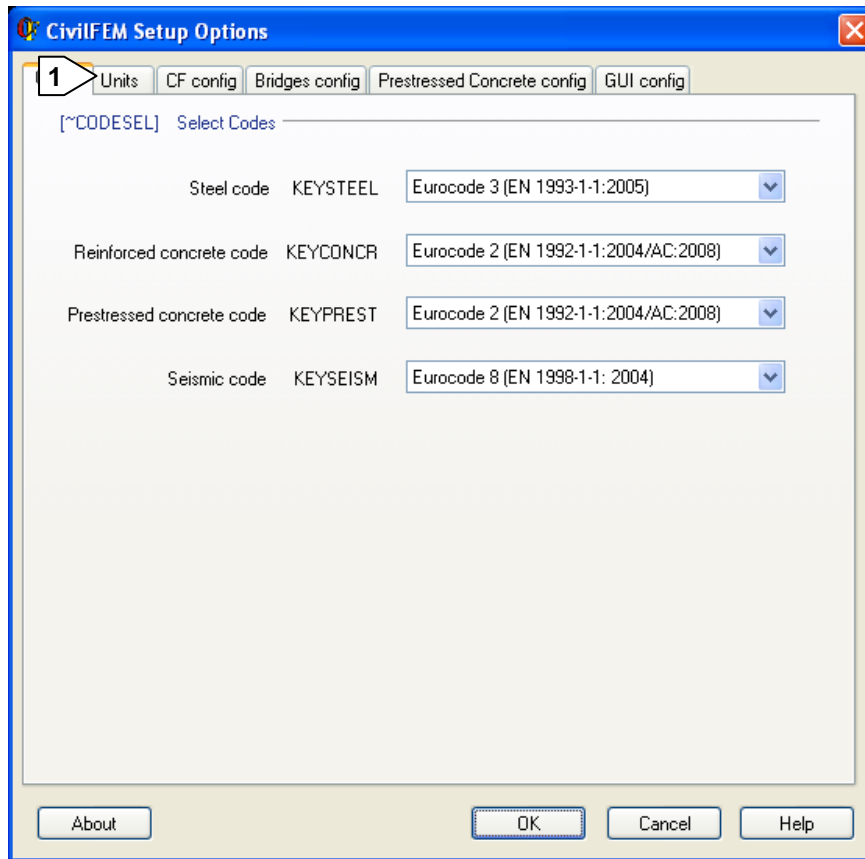
2 OK to define the title and close the dialog box.



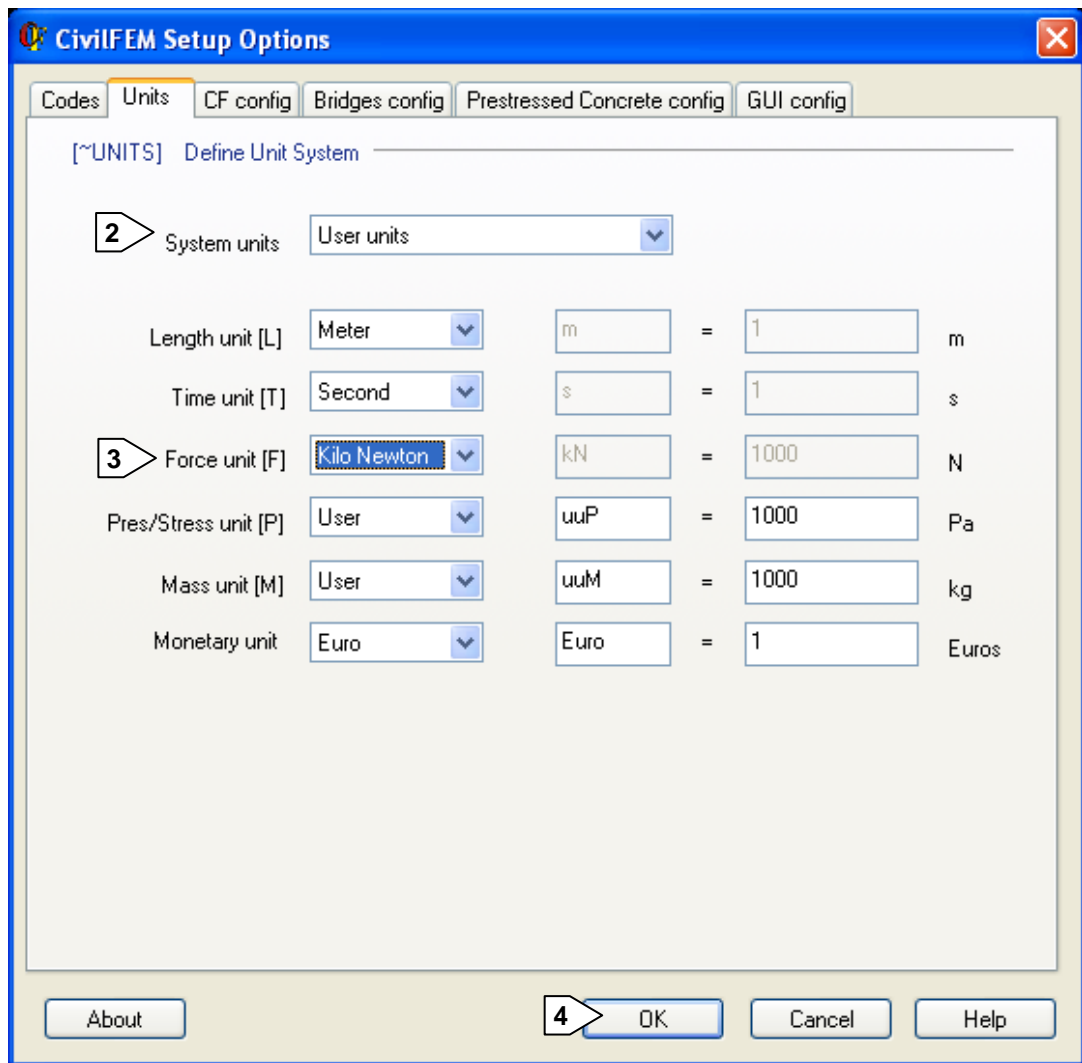
2. Set units

Utility Menu:– CivilFEM – **Civil Setup**

1 Pick on the Units tab



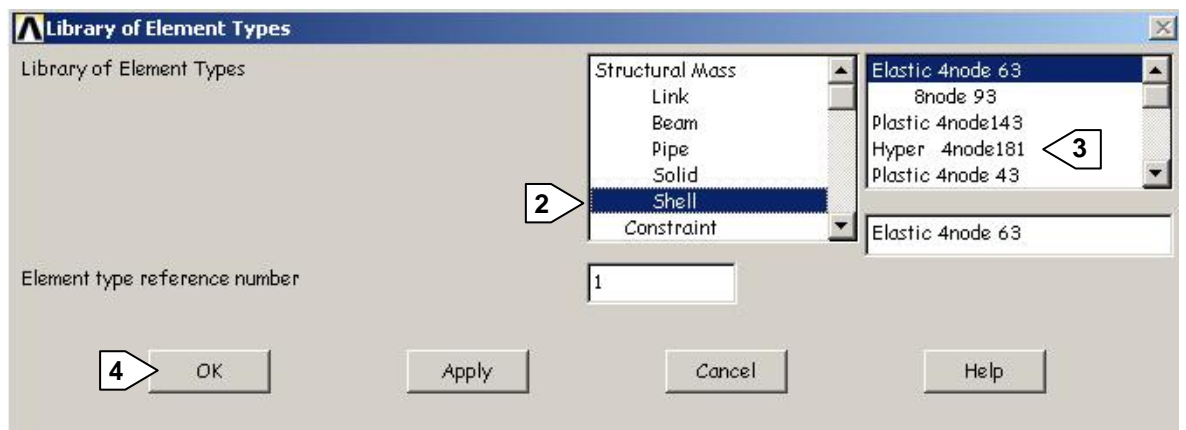
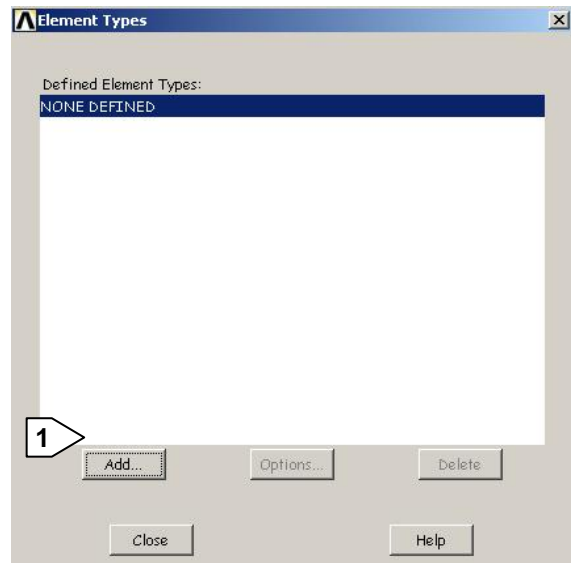
- 2 Select User units
- 3 In this analysis, we will select meters, seconds and kilo Newton.
- 4 OK to accept units and close the units dialog box



3. Define element type

Main Menu: **Preprocessor** → **Element Type** → **Add/Edit/Delete...**

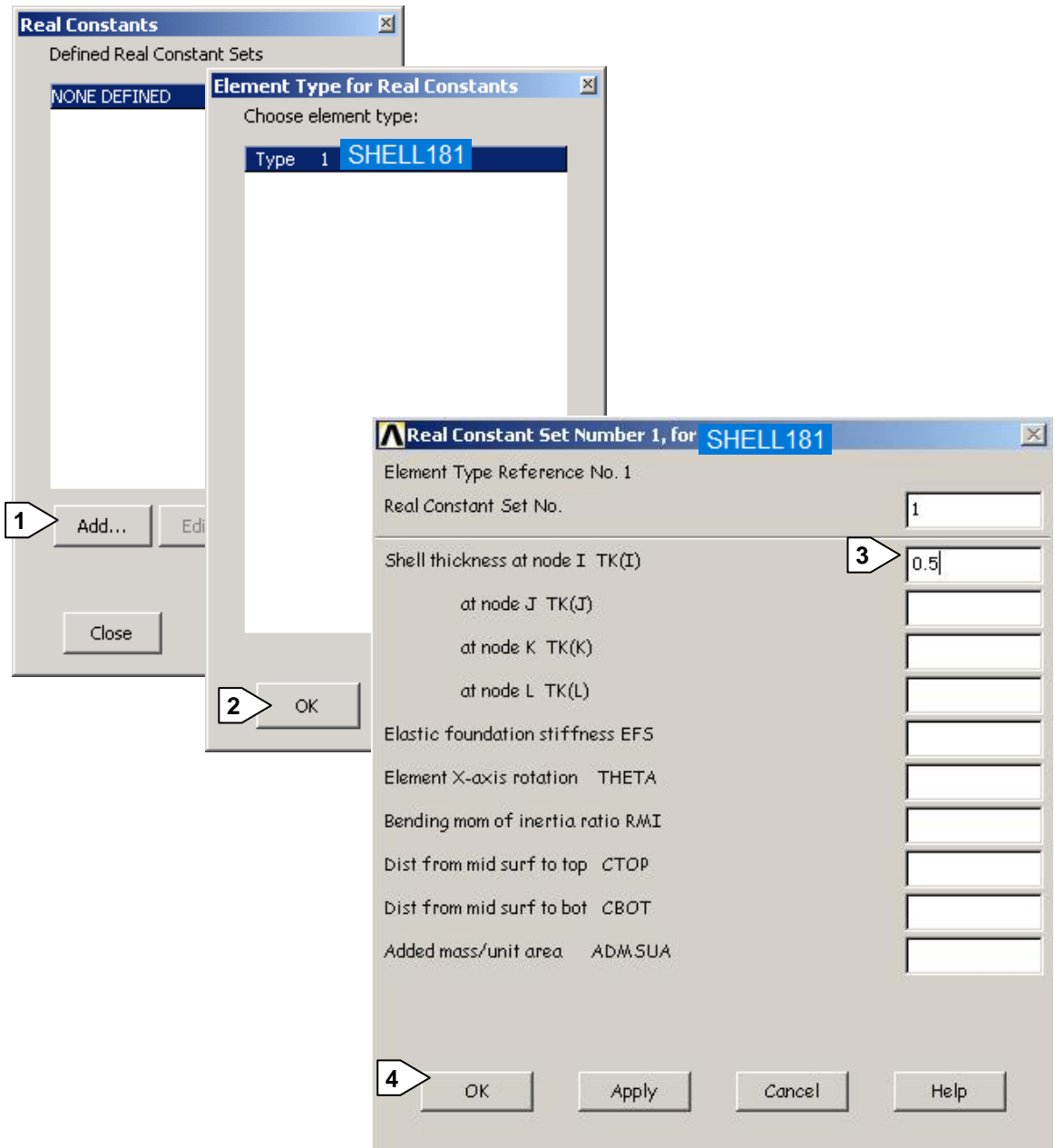
- 1 Pick Add...
- 2 Select Structural Shell
- 3 Choose Hyper 4node 181
- 4 OK
- 5 Close



4. Define real constants

Main Menu: **Preprocessor** → **Real Constants** → **Add/Edit/Delete...**

- 1 Pick Add...
- 2 OK
- 3 Enter 0.5 as TK(I)
- 4 OK
- 5 Close



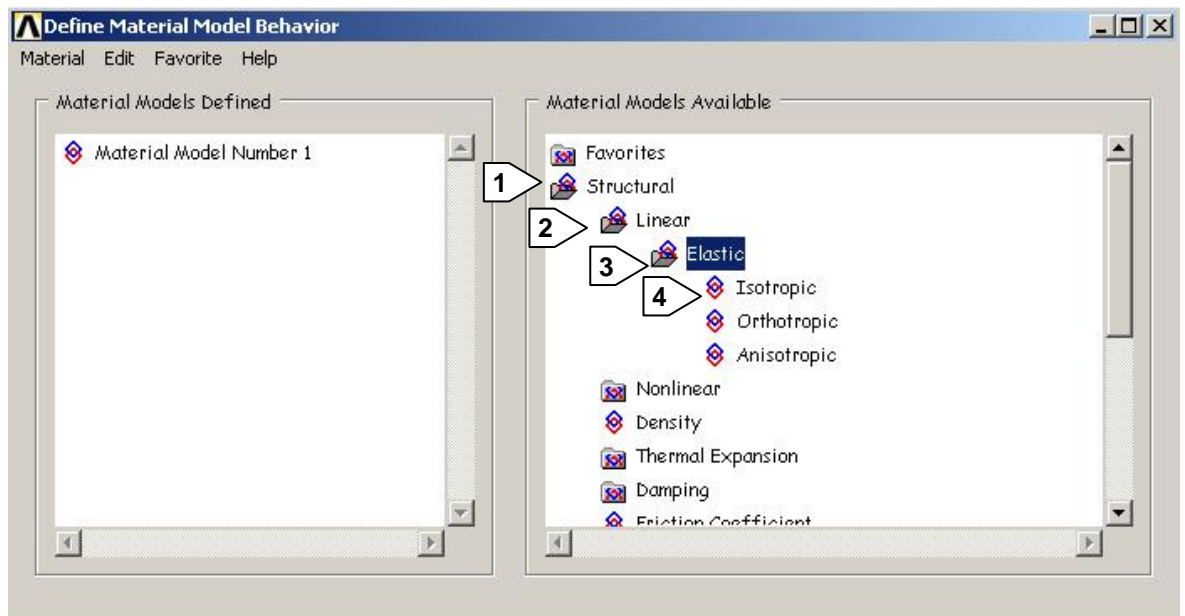


5. Define material

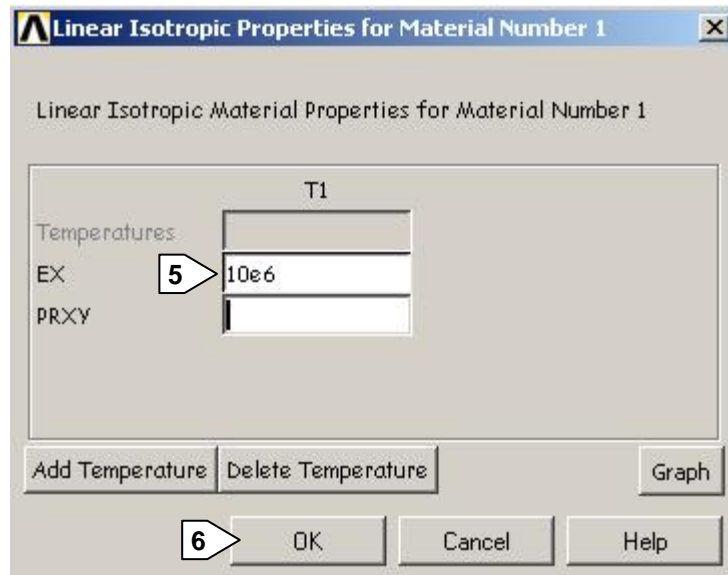
In this analysis we are going to define the material as a generic material. To do so, the only necessary property is the Young's modulus.

Main Menu: **Preprocessor** → **Materials props** → **Material models**

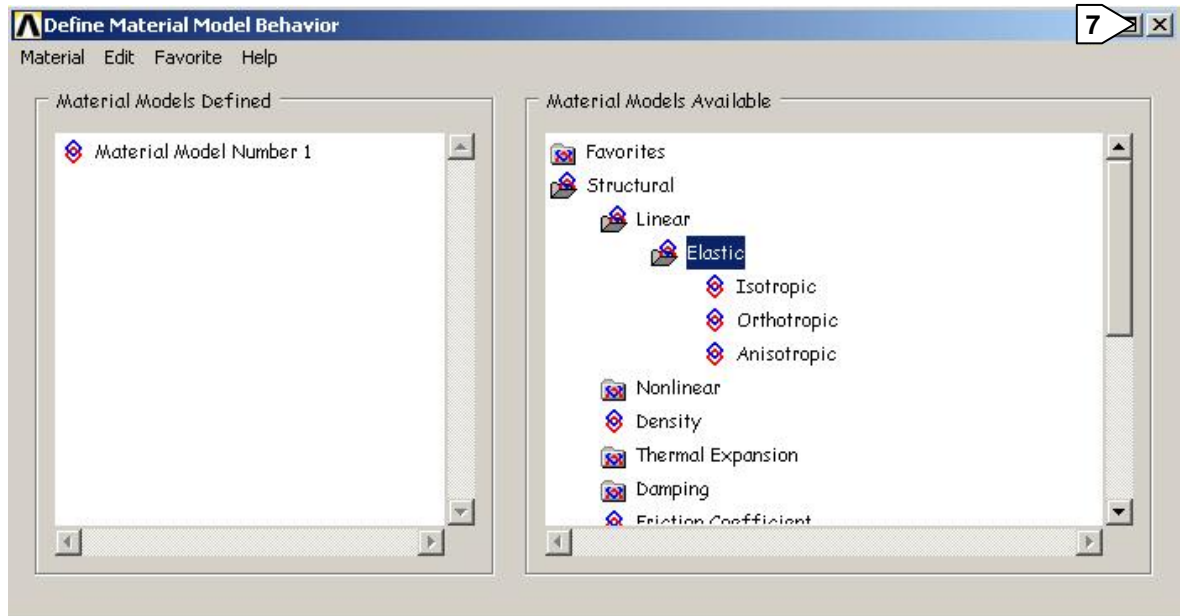
- 1 Double click the Structural folder
- 2 Double click the linear folder
- 3 Double click the elastic folder
- 4 Choose isotropic



- 5 Enter 10E6 for EX
- 6 Ok



7 Close the window to define material properties



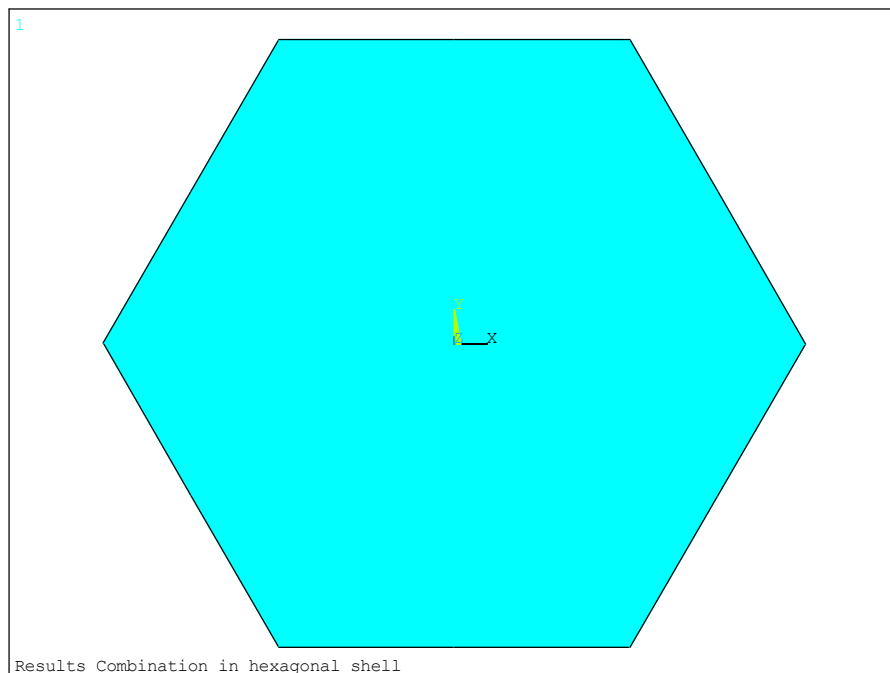
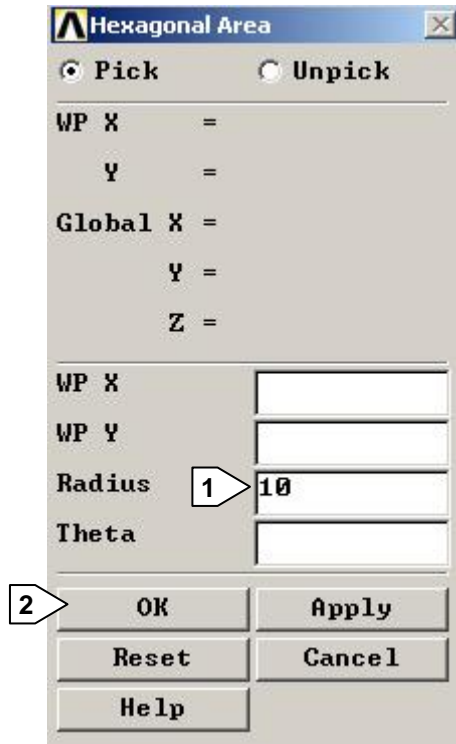
6. Define model geometry

We are going to define the geometry of the hexagonal shell through solid modeling.

Main Menu: **Preprocessor** → -Modeling- **Create** → -Areas- **Polygon** → **Hexagon +**

1 Enter 10 as vertex radius

2 OK



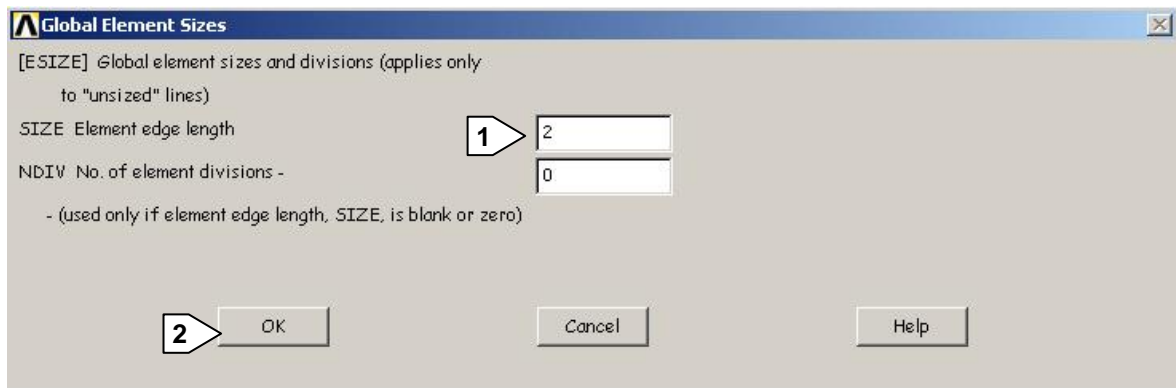
7. Mesh

We first specify the maximum element size and then mesh.

Main Menu: **Preprocessor** → -Meshing- **Size Cntrls** →
-ManualSize- -Global- **Size**

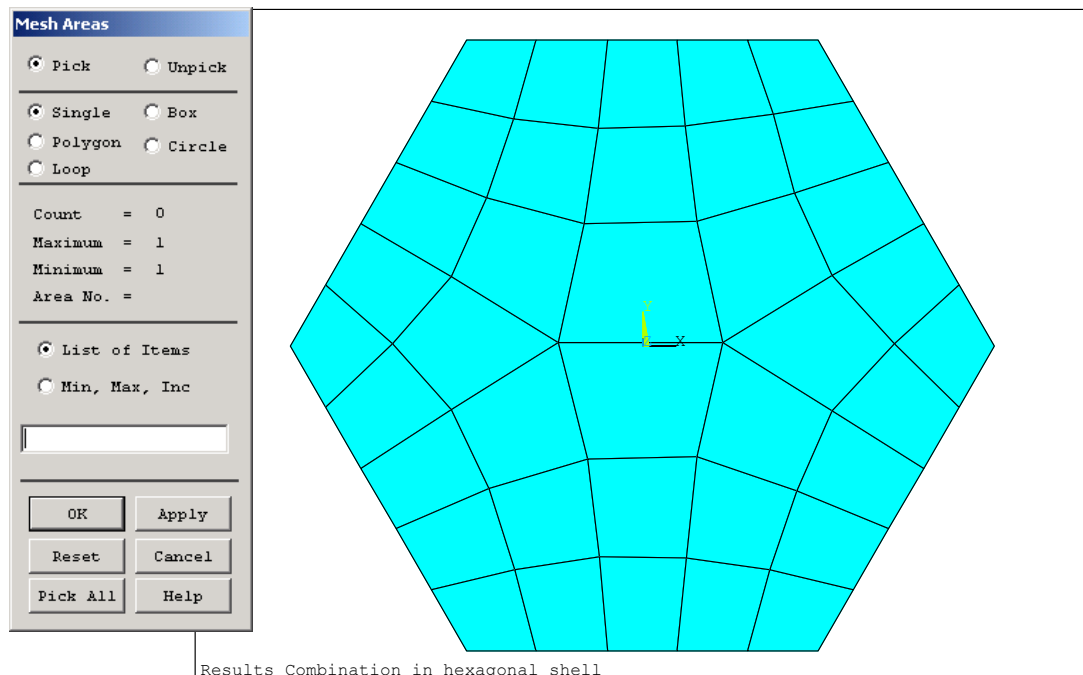
1 Enter 2 as element edge length

2 OK



Main Menu: **Preprocessor** → -Meshing- **Mesh** → -Areas- **Free +**

3 Pick All



8. Save the database

Before moving to the next step, we will save all we have done so far. The save operation will save the database to file.db and file.cfdb

Toolbar: **CFSAVE**

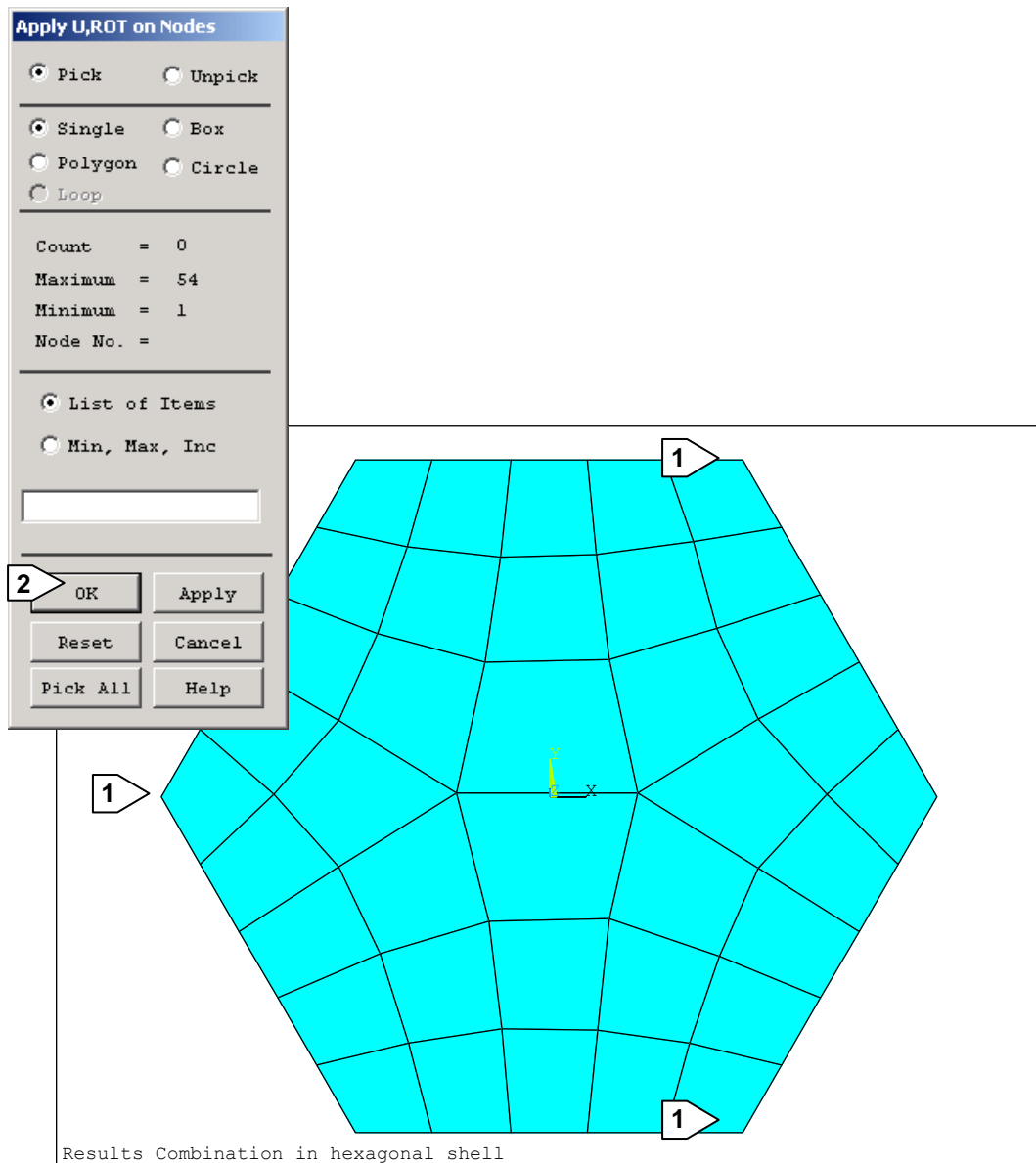
■ Solution

9. Apply displacement constraints

We apply the boundary conditions by restraining the displacement of alternate vertices.

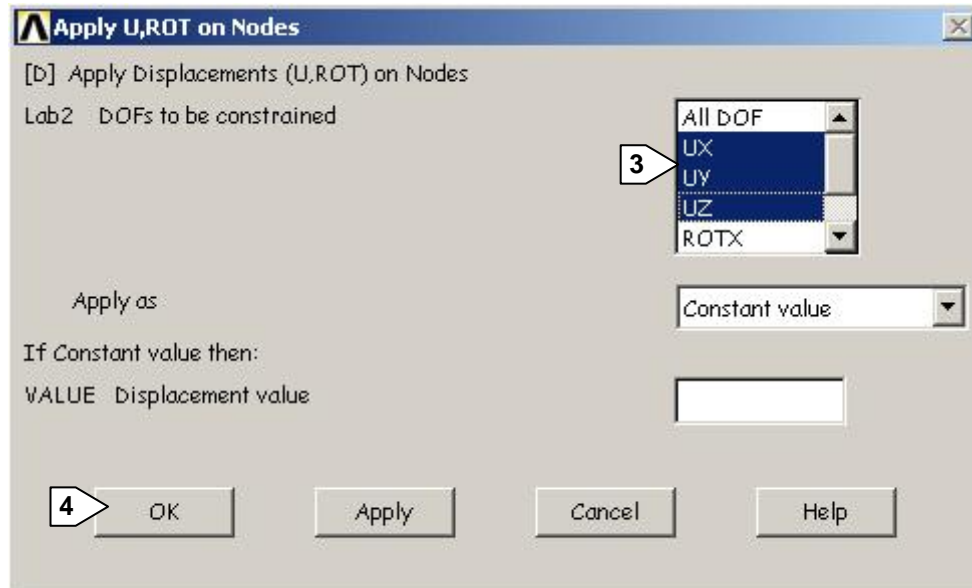
Main Menu: **Solution** → **Define Loads** → **Apply** → – Structural – **Displacement** → – **On nodes**

- 1 Pick the nodes of three alternate vertices
- 2 OK



3 Select UX, UY, UZ

4 OK



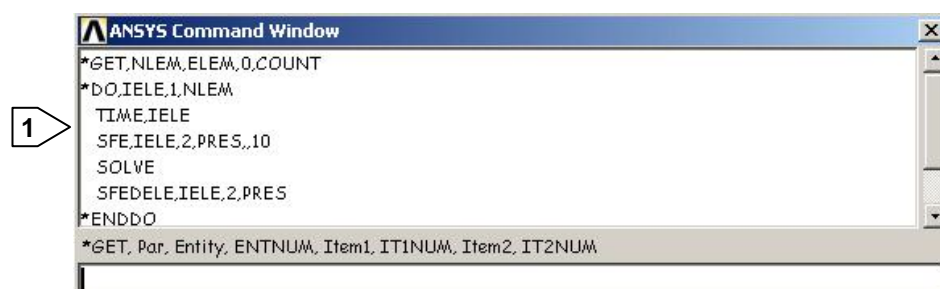
10. Define and solve load states

To know the maximum deflections in each point of the shell and how we must load it in order to obtain the maximum deflection at a point, we need to load and solve each of the shell elements separately. Therefore, every calculated *Load Step* can be taken as a starting state for the following combinations.

To solve each one of the mentioned states in an efficient way, we will use some simple commands in APDL written directly in the ANSYS command window.

1 Type the following lines:

```
*GET,NLEM,ELEM,0,COUNT
*DO,I,1,NLEM
  TIME,I
  SFE,I,2,PRES,,10
  SOLVE
  SFEDELE,I,2,PRES
*ENDDO
```



■ Postprocessor

The following steps need to be taken to accomplish the results combination:

- Target definition
- Combination rules definition
- Combination calculation
- Results review

11. Define targets

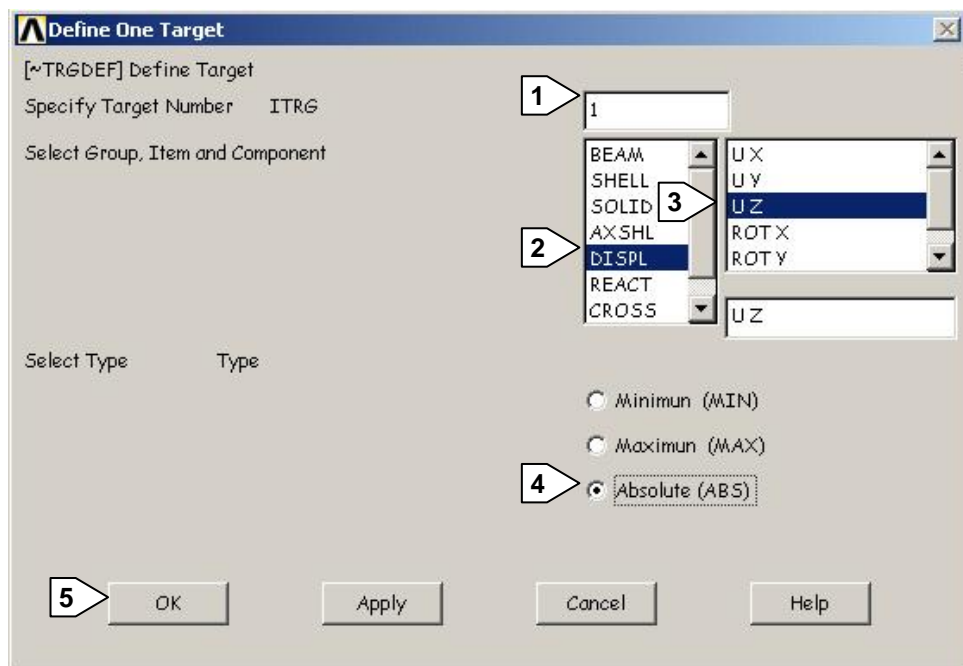
First of all, we must initialize the Combination Module in order to erase any previous Combinations. Then, the Targets (objectives to reach) and the Combinations will be defined. The Combination Module will utilize the Combinations (Start States and coefficients) to reach the Target values.

Main Menu: – CivilFEM – **Civil Postproces** → **Combine Results**
→ **Initialize**

Now we are going to define the target as maximum displacement in the Z direction in absolute value:

Main Menu: – CivilFEM – **Civil Postproces** → **Combine Results**
→ TARGETS: **Def One Target**

- 1 Specify Target Number 1
- 2 Select DISPL as Group
- 3 Select UZ as Item and Component
- 4 Pick type Absolute
- 5 Ok



12. Define combination rules

In this case, the combination rule is compatible because any possible subset of starting states can be taken into account.

Main Menu: – CivilFEM – **Civil Postprocess** → **Combine Results** → **Combination**

This opens the Combination Explorer

- 1 Enter combination title: Compatible load on shell
- 2 Specify Combination Rule 1

New Combination - (fill the properties and press the button)

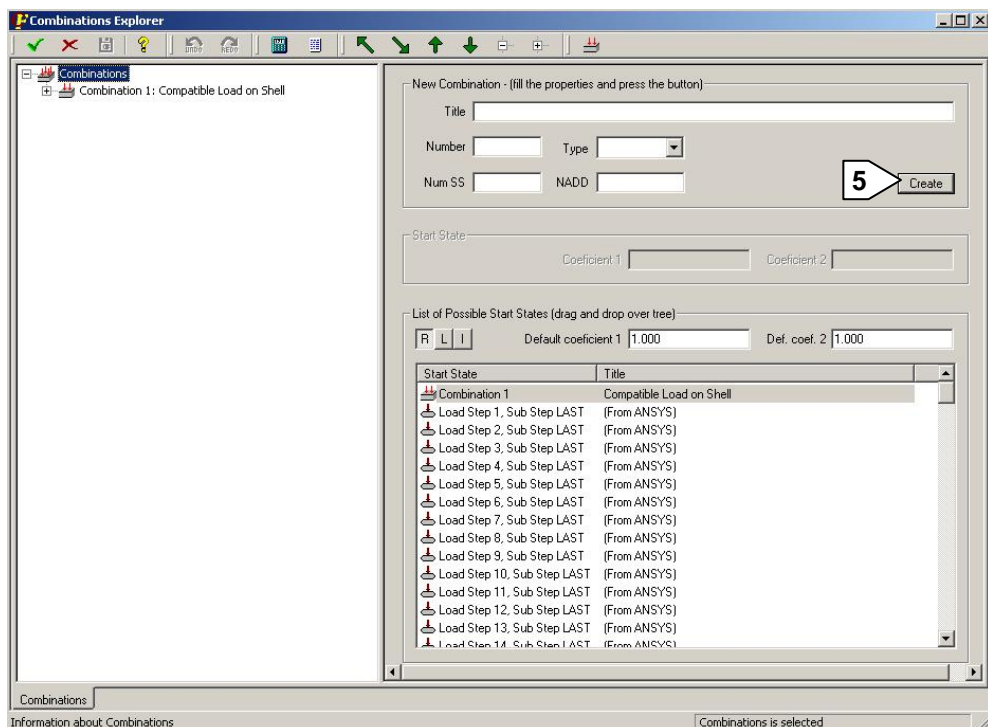
1 Title Compatible Load on Shell

2 Number 1 3 Type COMPATIB Compatible Start States. Addition of any subsets of the Start States defined (none, one, several or all of them).

4 Num SS 52 NADD

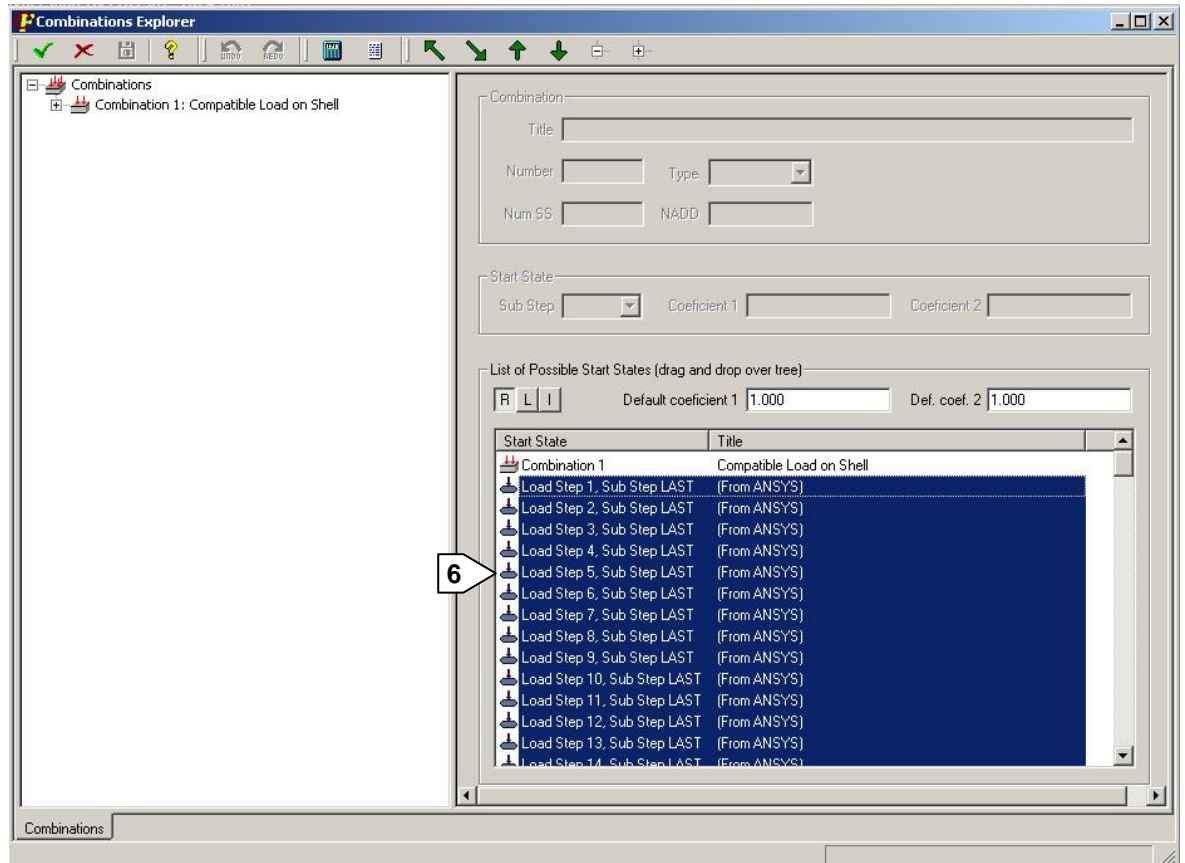
5 Create

- 3 Select type of combination rule compatible
- 4 Enter number of Start States: The number of start states coincides with the number of elements previously saved in the NEL parameter
- 5 Create combination



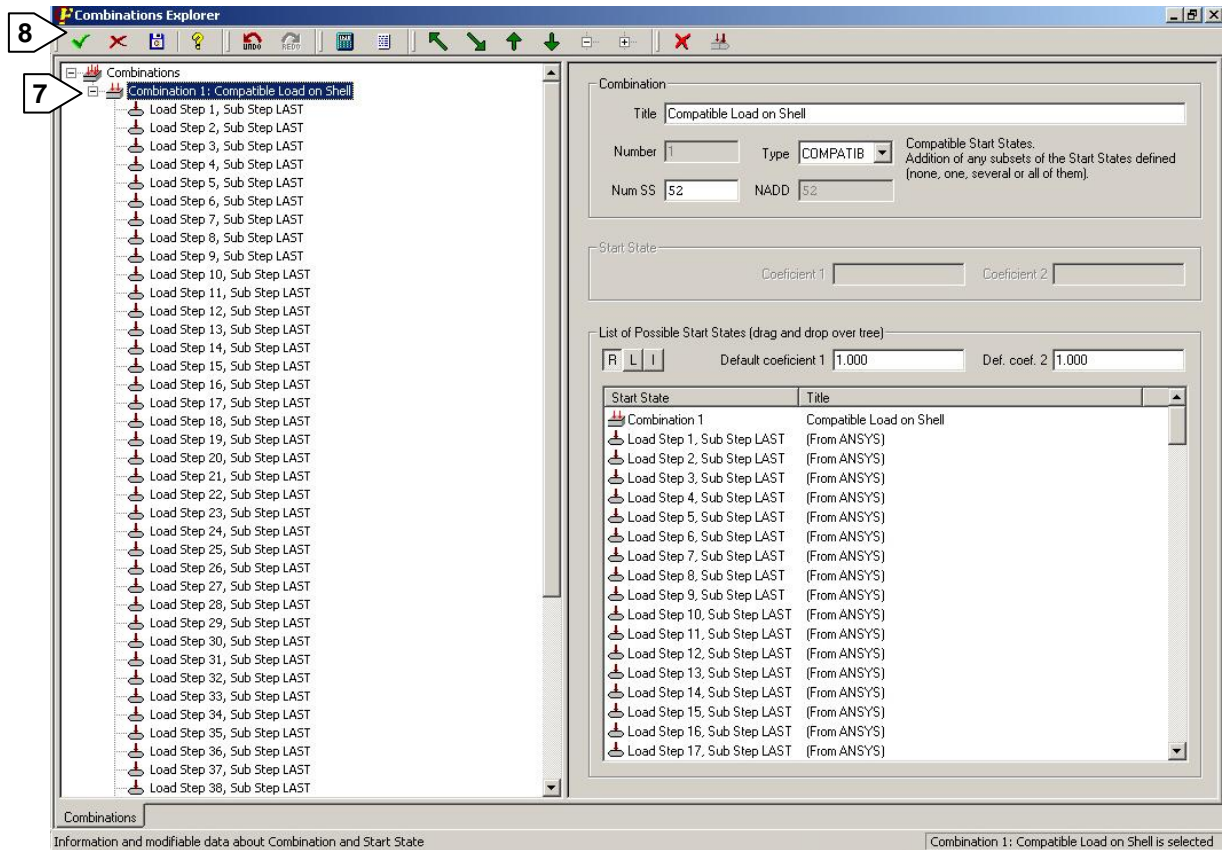
Now we are going to specify the starting states that come into play in the combination rule 1 previously defined, so in Combinations Explorer:

- 6 Select from list the NEL start states : from “**Load Step 1, Sub Step LAST**” to “**Load Step NEL, Sub Step LAST**”



- 7 Drag and drop the NEL start states over **Combination 1** tree

- 8 OK

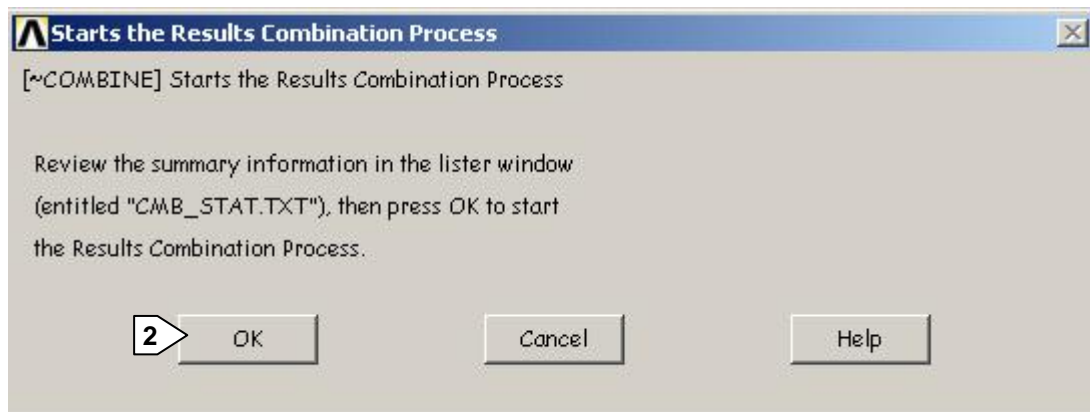
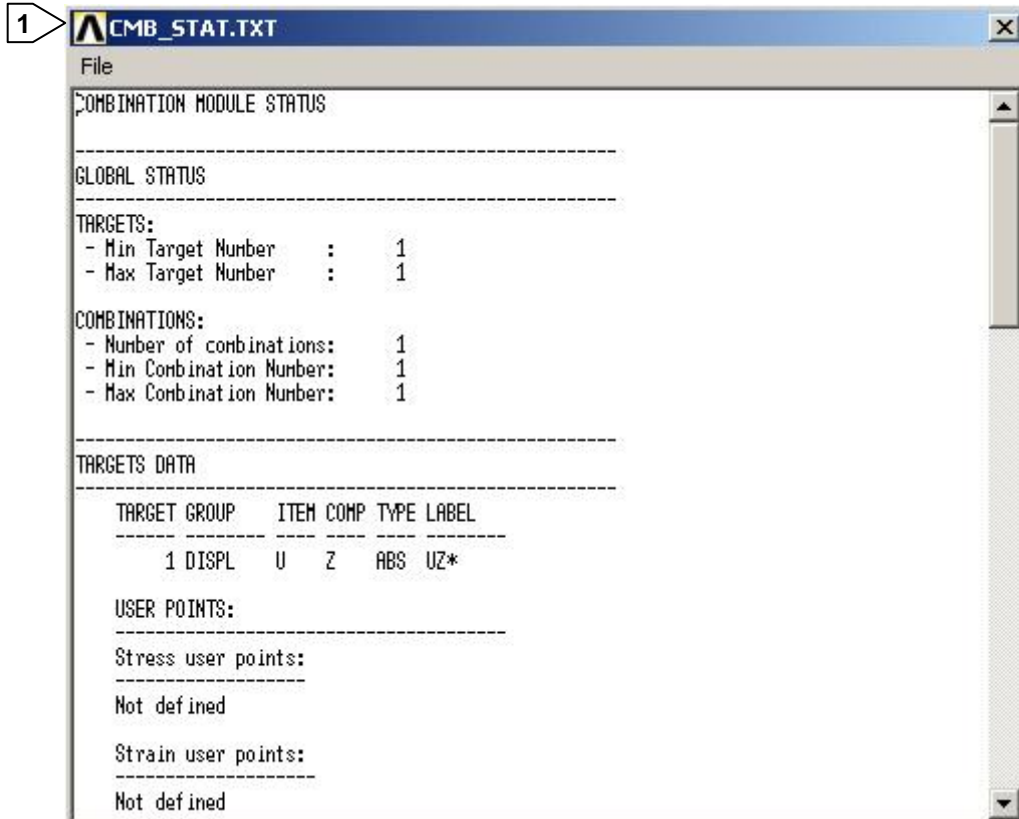


13. Carry out combinations

Prior to completing the combinations CivilFEM shows a window with the global status in order to review all data before carrying out the combinations.

Main Menu: – CivilFEM – **Civil Postprocess** → **Combine Results**
 → **CALCULATE: Combine** → **Combine for targets**

- 1 Review the information in the list window
- 2 OK to start the combination process



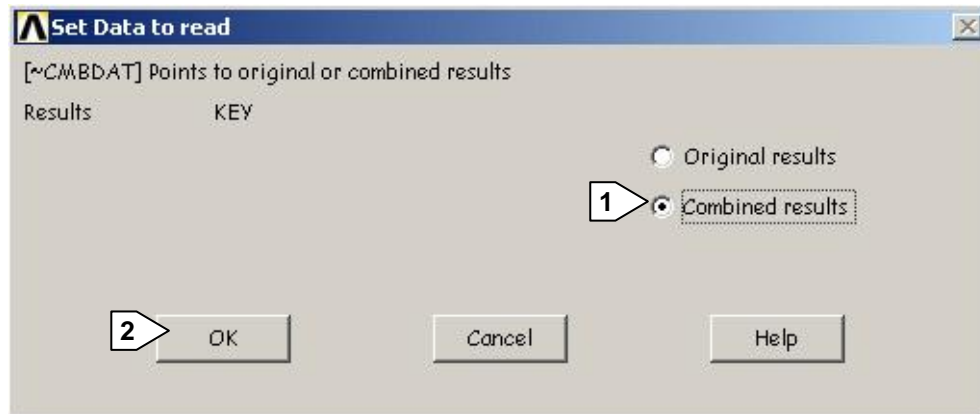
14. Read in results

First of all we need to point to the combined results.

Main Menu: – CivilFEM – **Civil Postprocess** → **Combine Results** → READ RESULTS: **Set data to read**

1 Select combined results

2 OK



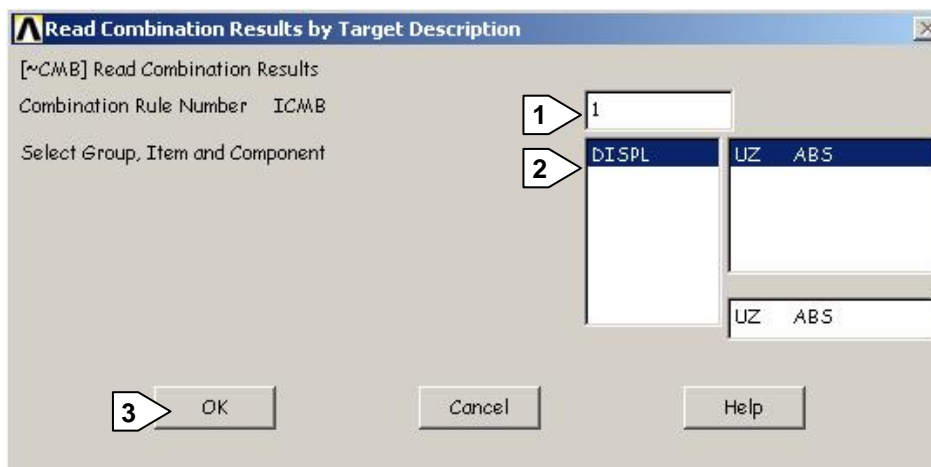
We read the results of combination rule 1 that satisfy the defined target.

Main Menu: – CivilFEM – **Civil Postprocess** → **Combine Results** → READ RESULTS: **By description...**

1 Enter 1 to read result of combination rule 1

2 Select the target (group and components) from which you would like to read the results

3 OK

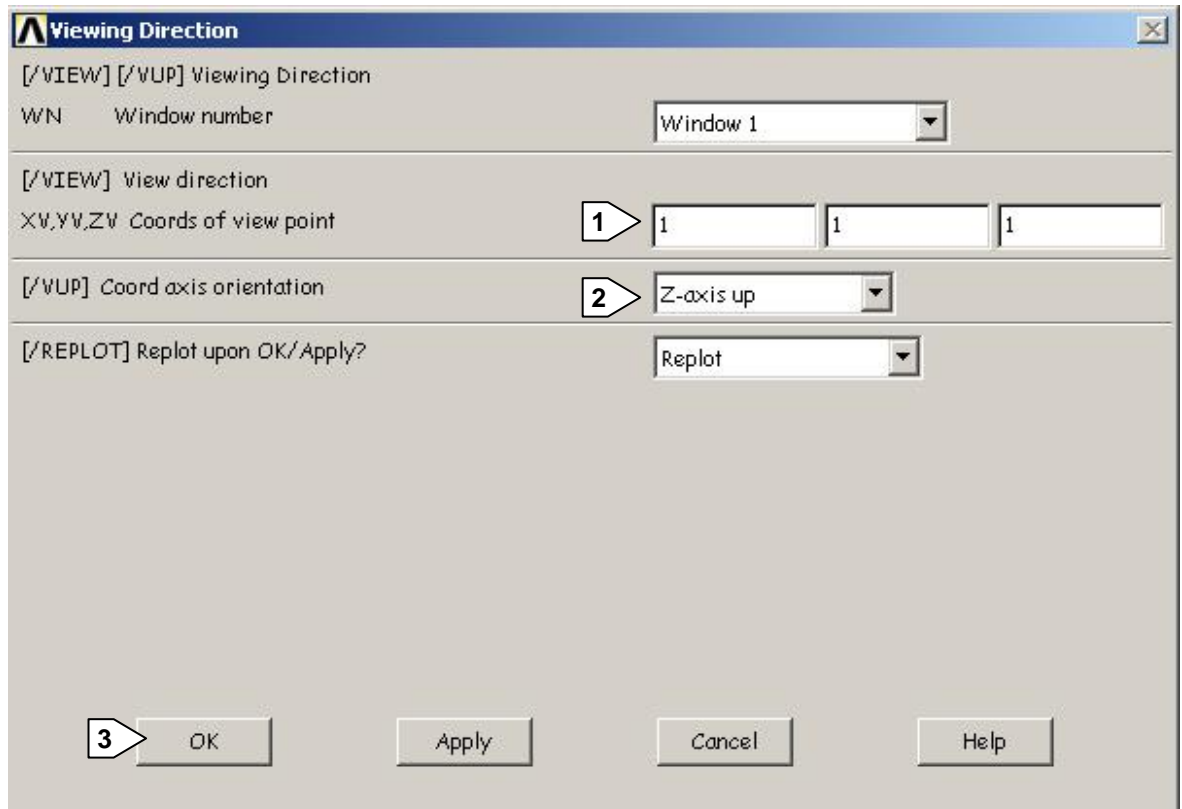


15. Plot of the maximum deflection at each point of the shell

We are going to change the viewing direction of the model to better observe the deflections distribution.

Utility Menu: **PlotCtrls** → **View Settings** → **Viewing Direction...**

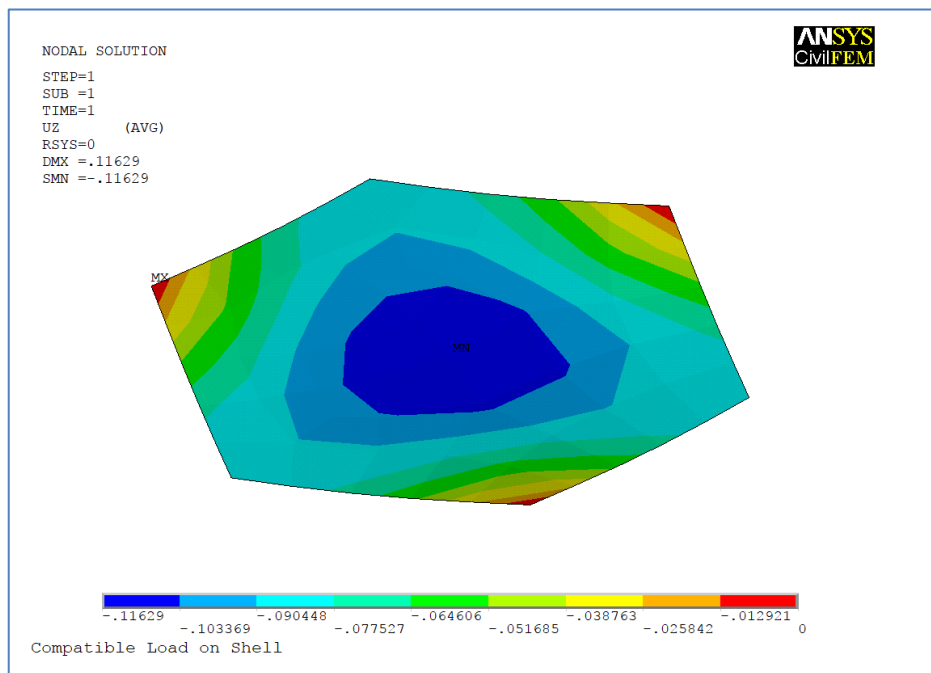
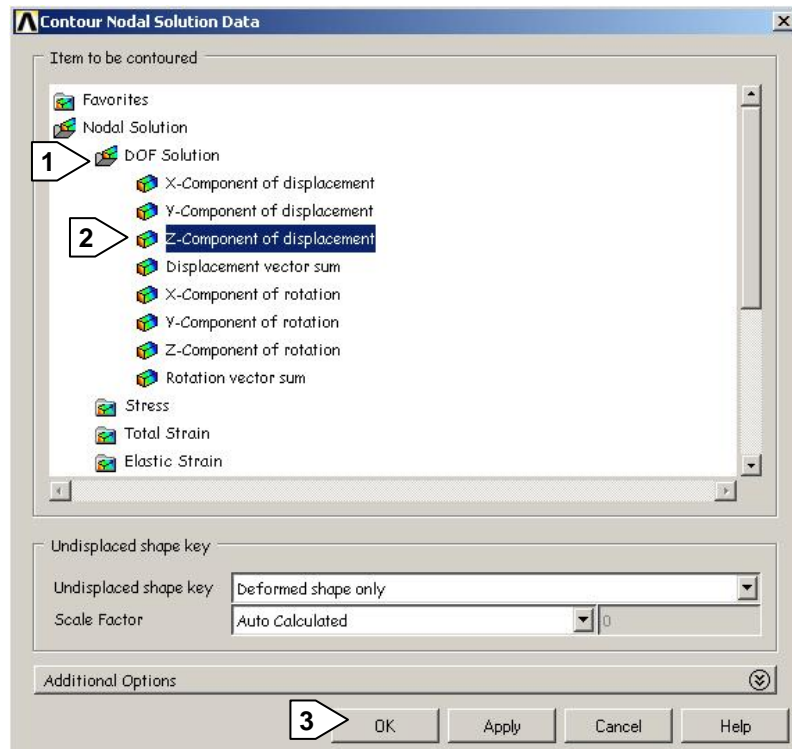
- 1 Enter viewpoint: 1,1,1
- 2 Select Z-axis up orientation
- 3 OK



In the following step we plot the maximum deflection in each point of the shell. You must take into account that this distribution does not correspond to any specific load state. Moreover, the maximum deflection for every point will come from a different load arrangement.

Main Menu: **General Postproc** → **Plot Results** → **-Contour Plot-Nodal Solu**

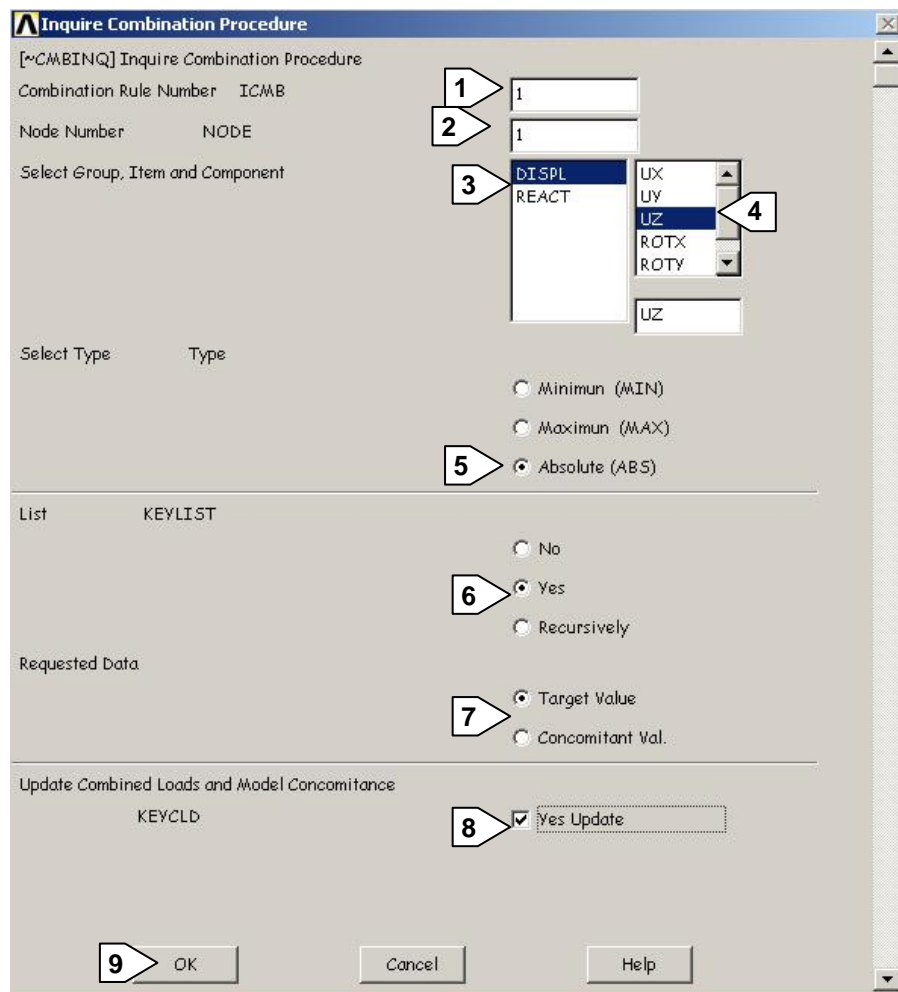
- 1 Select DOF solution
- 2 Select Z-Component of displacement
- 3 OK



16. Calculation of the load distribution to obtain the maximum deflection at a vertex

Main Menu: – CivilFEM – **Civil Postprocess** → **Combine Results** → INQUIRE: **Nodal Results**

- 1 Enter 1 to select combination rule 1
- 2 Enter 1 for node
- 3 Select group DISPL
- 4 Select item and component UZ
- 5 Pick Absolute
- 6 Pick Yes to list result
- 7 Pick Target Value
- 8 Select Yes Update
- 9 OK



In the following graphs we can read a list of starting states that satisfy the defined target while observing its distribution in the model.

CMB_COEF.TXT

File

LIST OF COEFFICIENTS AND SELECTED START STATES THAT MATCH WITH:

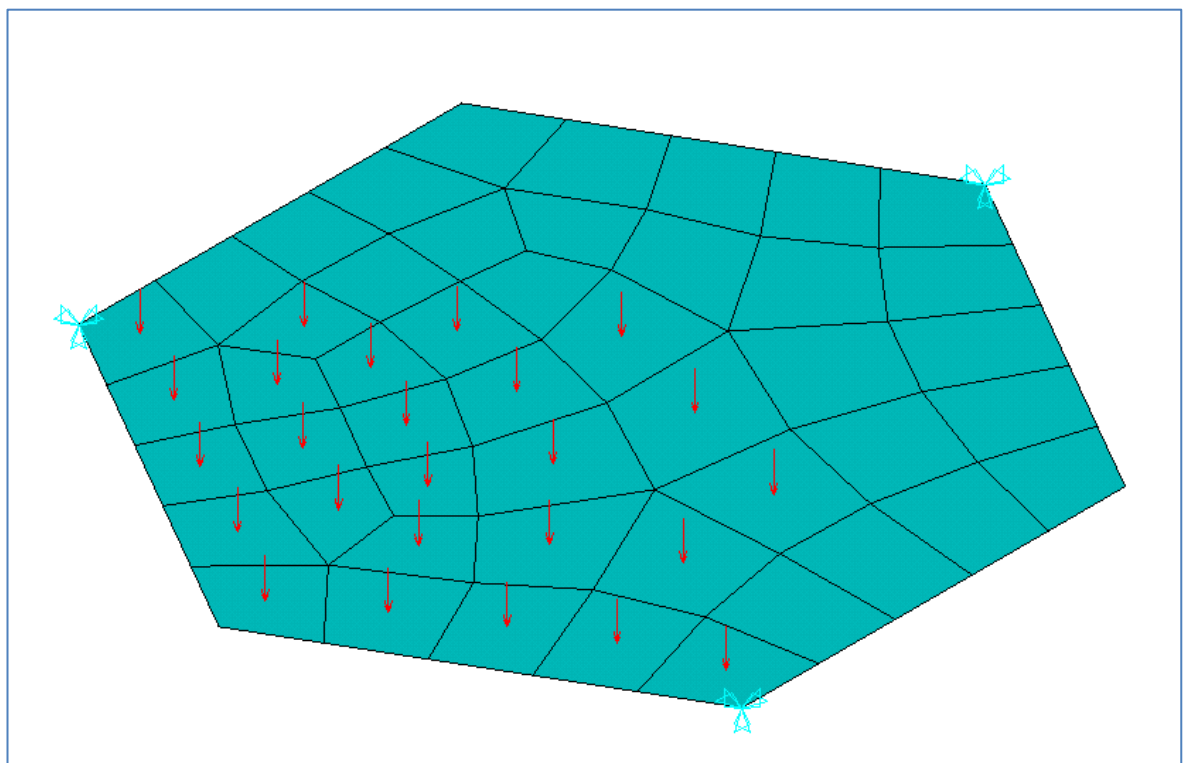
- Target : DISPL ,UZ*
 - At Node: 1
 - Requested Value : U Z (Target Value)

 Combination Rule 1

 Title: Compatible Load on Shell
 Type : COMPATIB (Compatible)

The selected Start States are:

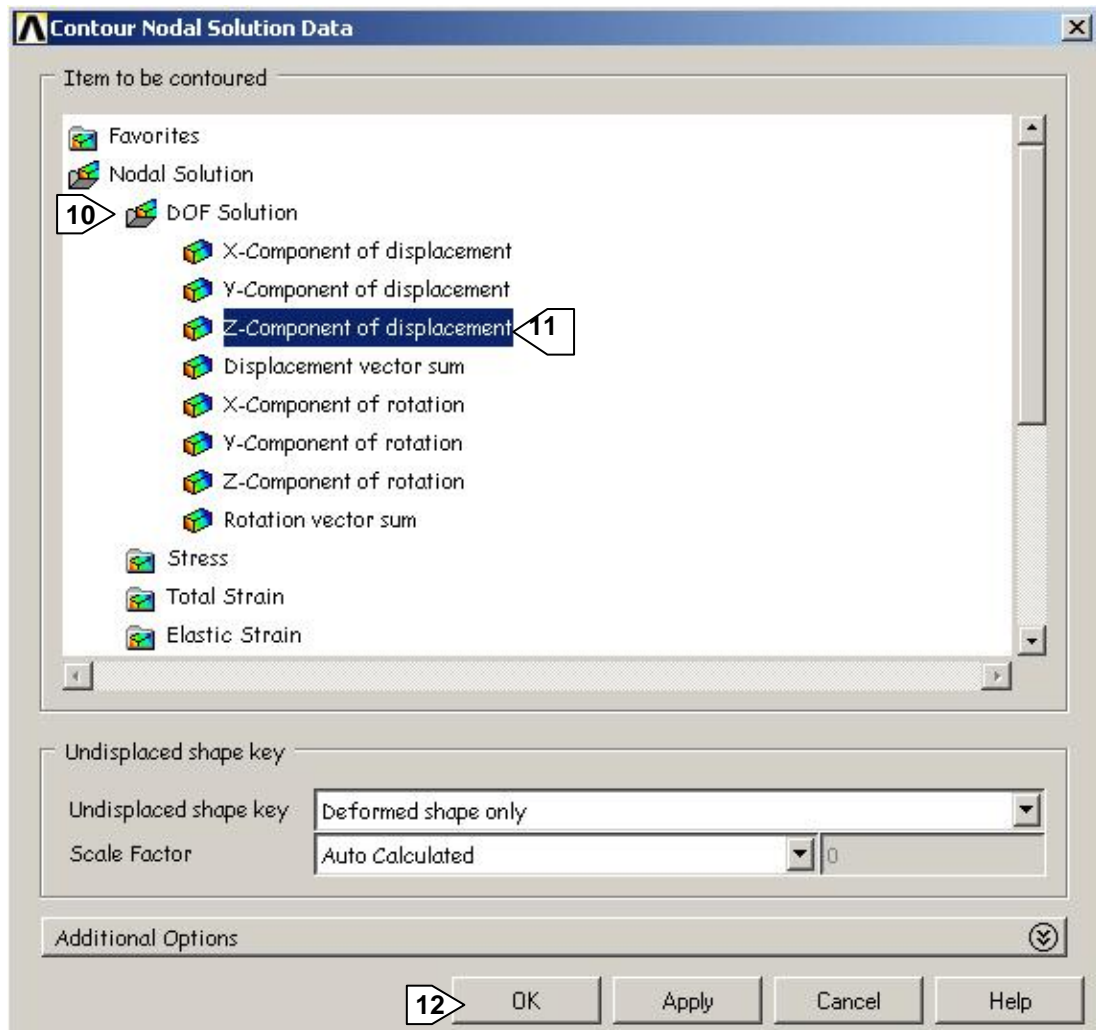
S.State	Description	Value
1	Load Step 1	-2.357E-03
2	Load Step 2	-3.859E-03
3	Load Step 3	-5.196E-03
10	Load Step 10	-490.310E-06
11	Load Step 11	-650.757E-06
19	Load Step 19	-3.797E-03
20	Load Step 20	-750.804E-06
21	Load Step 21	-8.582E-03
22	Load Step 22	-5.037E-03
23	Load Step 23	-10.498E-03
24	Load Step 24	-1.234E-03
25	Load Step 25	-537.301E-06
26	Load Step 26	-1.819E-03
27	Load Step 27	-2.987E-03
28	Load Step 28	-2.654E-03
29	Load Step 29	-1.210E-03
30	Load Step 30	-620.905E-06
32	Load Step 32	-364.289E-06
33	Load Step 33	-5.049E-03
34	Load Step 34	-3.781E-03
35	Load Step 35	-1.938E-03
41	Load Step 41	-6.541E-03
42	Load Step 42	-9.835E-03
43	Load Step 43	-5.086E-03
44	Load Step 44	-3.165E-03
	Result	-88.036E-03

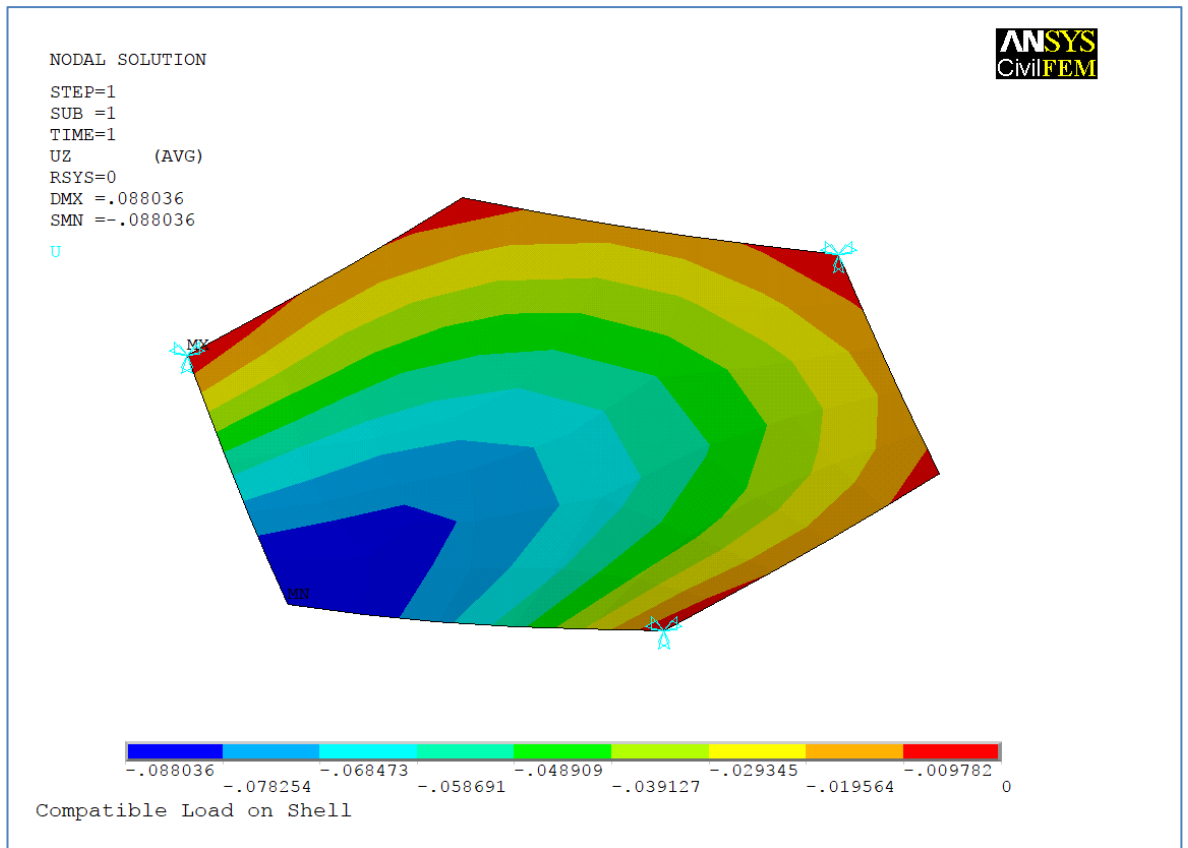


To obtain the deflections distribution for the start states combination that satisfy the requested targets we must proceed as follows:

Main Menu: **General Postproc** → **Plot Results** → -Contour Plot-**Nodal Solu**

- 10 Select DOF solution
- 11 Select Z-Component of displacement
- 12 OK





17. Exit the ANSYS program

We save everything before exiting the ANSYS program.

Utility Menu: **File** → **Exit**

- 1 Pick Save Everything
- 2 OK

