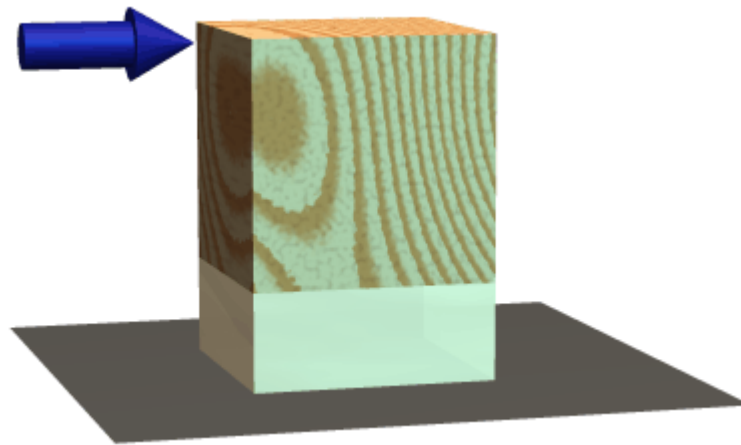


Substructuring

Introduction

This tutorial was completed using ANSYS 7.0. The purpose of the tutorial is to show how to use substructuring in ANSYS. Substructuring is a procedure that condenses a group of finite elements into one *super-element*. This reduces the required computation time and also allows the solution of very large problems.

A simple example will be demonstrated to explain the steps required, however, please note that this model is not one which requires the use of substructuring. The example involves a block of wood ($E = 10 \text{ GPa}$, $\nu = 0.29$) connected to a block of silicone ($E = 2.5 \text{ MPa}$, $\nu = 0.41$) which is rigidly attached to the ground. A force will be applied to the structure as shown in the following figure. For this example, substructuring will be used for the wood block.



The use of substructuring in ANSYS is a three stage process:

1. **Generation Pass**

Generate the super-element by condensing several elements together. Select the degrees of freedom to save (master DOFs) and to discard (slave DOFs). Apply loads to the super-element

2. **Use Pass**

Create the full model including the super-element created in the generation pass. Apply remaining loads to the model. The solution will consist of the reduced solution for the super-element and the complete solution for the non-superelements.

3. **Expansion Pass**

Expand the reduced solution to obtain the solution at all DOFs for the super-element.

Note that this method is a bottom-up substructuring (each super-element is created separately and then assembled in the Use Pass). Top-down substructuring is also possible in ANSYS (the entire model is built, then

super-element are created by selecting the appropriate elements). This method is suitable for smaller models and has the advantage that the results for multiple super-elements can be assembled in postprocessing.

Expansion Pass: Creating the Super-element

Preprocessing: Defining the Problem

1. Give Generation Pass a Jobname

Utility Menu > File > Change Jobname ...

Enter 'GEN' for the jobname

2. Open preprocessor menu

ANSYS Main Menu > Preprocessor
/PREP7

3. Create geometry of the super-element

Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners
BLC4, XCORNER, YCORNER, WIDTH, HEIGHT

Create a rectangle with the dimensions (all units in mm):

```
XCORNER (WP X) = 0
YCORNER (WP Y) = 40
Width = 100
Height = 100
```

4. Define the Type of Element

Preprocessor > Element Type > Add/Edit/Delete...

For this problem we will use PLANE42 (2D structural solid). This element has 4 nodes, each with 2 degrees of freedom (translation along the X and Y axes).

5. Define Element Material Properties

Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic

In the window that appears, enter the following geometric properties for wood:

- i. Young's modulus EX: 10000 (MPa)
- ii. Poisson's Ratio PRXY: 0.29

6. Define Mesh Size

Preprocessor > Meshing > Size Cntrls > Manual Size > Areas > All Areas ...

For this example we will use an element edge length of 10mm.

7. Mesh the block

Preprocessor > Meshing > Mesh > Areas > Free > click 'Pick All'
AMESH, 1

Solution Phase: Assigning Loads and Solving

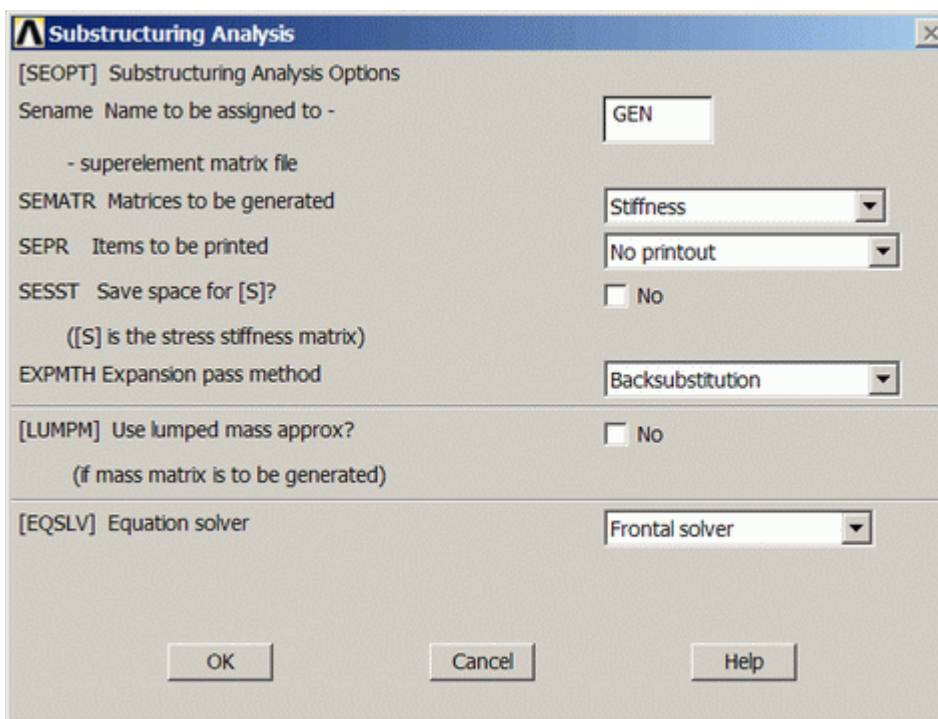
1. Define Analysis Type

Solution > Analysis Type > New Analysis > Substructuring
ANTYPE, SUBST

2. Select Substructuring Analysis Options

It is necessary to define the substructuring analysis options

- Select **Solution > Analysis Type > Analysis Options**
- The following window will appear. Ensure that the options are filled in as shown.

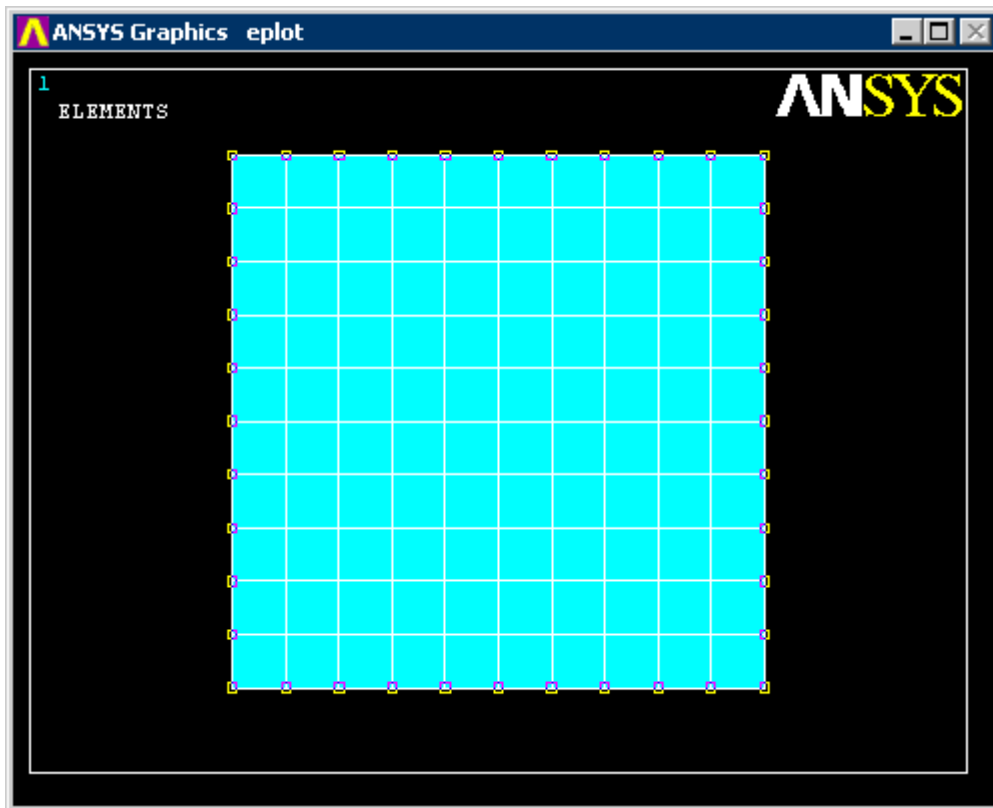


- *Sename* (the name of the super-element matrix file) will default to the jobname.
- In this case, the stiffness matrix is to be generated.
- With the option *SEPR*, the stiffness matrix or load matrix can be printed to the output window if desired.

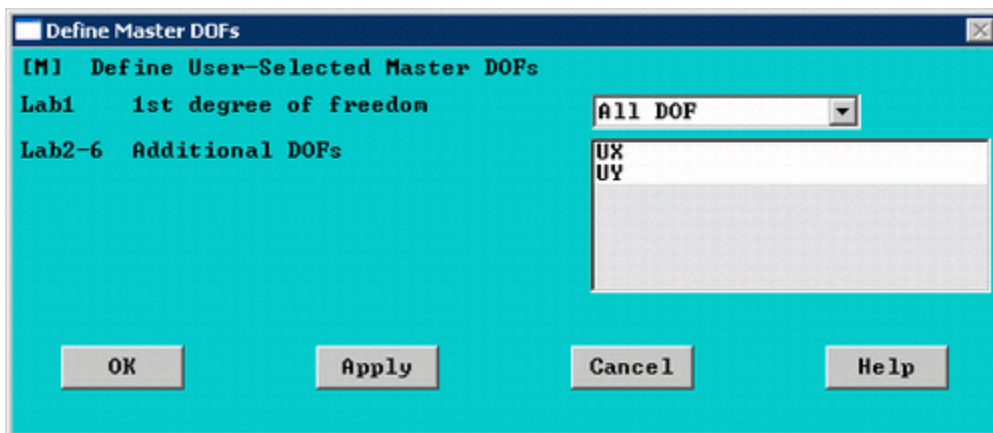
3. Select Master Degrees of Freedom

Master DOFs must be defined at the interface between the super-element and other elements in addition to points where loads/constraints are applied.

- Select **Solution > Master DOFs > User Selected > Define**
- Select the Master DOF as shown in the following figure.



- In the window that appears, set the 1st degree of freedom to **All DOF**

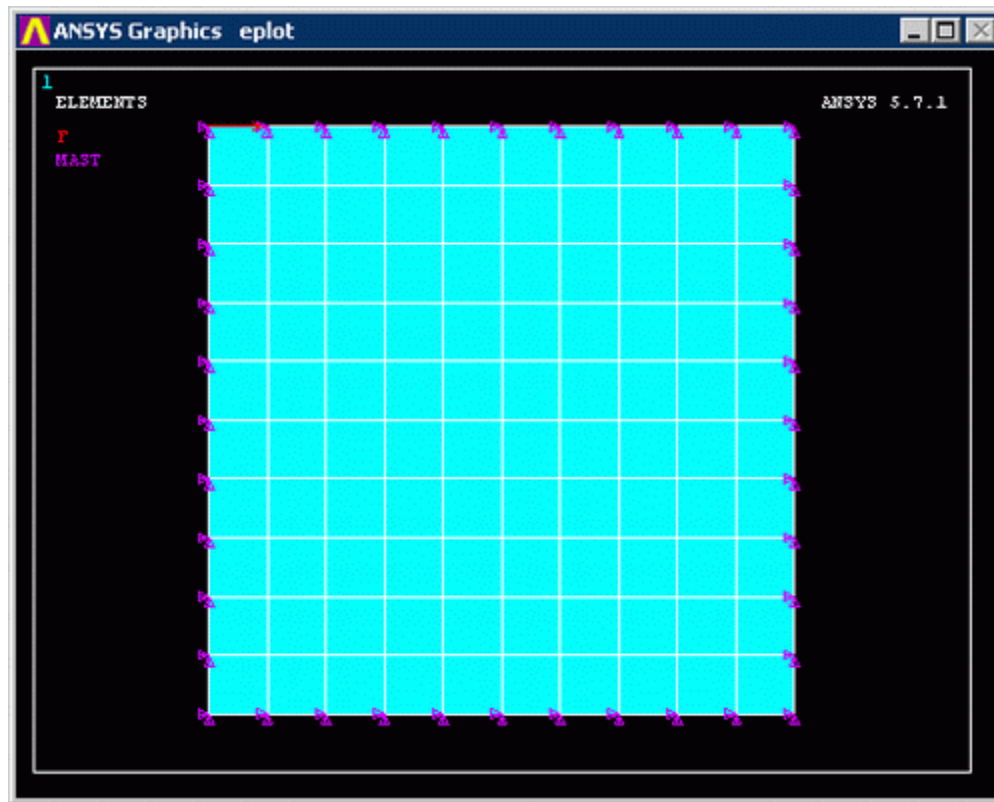


4. Apply Loads

Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

Place a load of 5N in the x direction on the top left hand node

The model should now appear as shown in the figure below.



5. Save the database

Utility Menu > File > Save as Jobname.db
SAVE

Save the database to be used again in the expansion pass

6. Solve the System

Solution > Solve > Current LS
SOLVE

Use Pass: Using the Super-element

The Use Pass is where we model the entire model, including the super-elements from the Generation Pass.

Preprocessing: Defining the Problem

1. Clear the existing database

Utility Menu > File > Clear & Start New

2. Give Use Pass a Jobname

Utility Menu > File > Change Jobname ...
FILNAME, USE

Enter 'USE' for the jobname

3. Open preprocessor menu

ANSYS Main Menu > Preprocessor
/PREP7

Now we need to bring the Super-element into the model

4. Define the Super-element Type

Preprocessor > Element Type > Add/Edit/Delete...

Select 'Super-element' (MATRIX50)

5. Create geometry of the non-superelement (Silicone)

Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners
BLC4, XCORNER, YCORNER, WIDTH, HEIGHT

Create a rectangle with the dimensions (all units in mm):

```
XCORNER (WP X) = 0
YCORNER (WP Y) = 0
Width = 100
Height = 40
```

6. Define the Non-Superelement Type

Preprocessor > Element Type > Add/Edit/Delete...

We will again use PLANE42 (2D structural solid).

7. Define Element Material Properties

Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic

In the window that appears, enter the following geometric properties for silicone:

- i. Young's modulus EX: 2.5 (MPa)
- ii. Poisson's Ratio PRXY: 0.41

8. Define Mesh Size

Preprocessor > Meshing > Size Cntrls > Manual Size > Areas > All Areas ...

For this block we will again use an element edge length of 10mm. Note that it is imperative that the nodes of the non-superelement match up with the super-element MDOFs.

9. Mesh the block

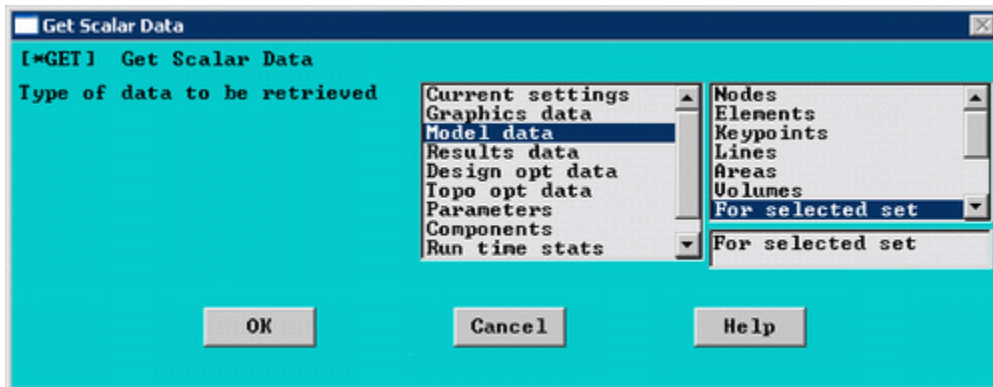
Preprocessor > Meshing > Mesh > Areas > Free > click 'Pick All'
AMESH, 1

10. Offset Node Numbering

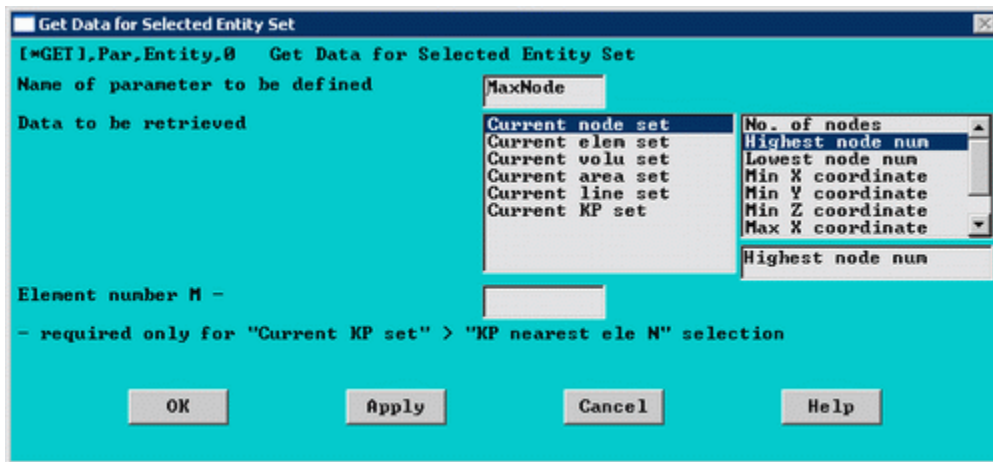
Since both the super-element and the non-superelement were created independently, they contain similarly numbered nodes (ie both objects will have node #1 etc.). If we bring in the super-element with similar node numbers, the nodes will overwrite existing nodes from the non-superelements. Therefore, we need to offset the super-element nodes

Determine the number of nodes in the existing model

- Select **Utility Menu > Parameters > Get Scalar Data ...**
- The following window will appear. Select Model Data, For Selected set as shown.

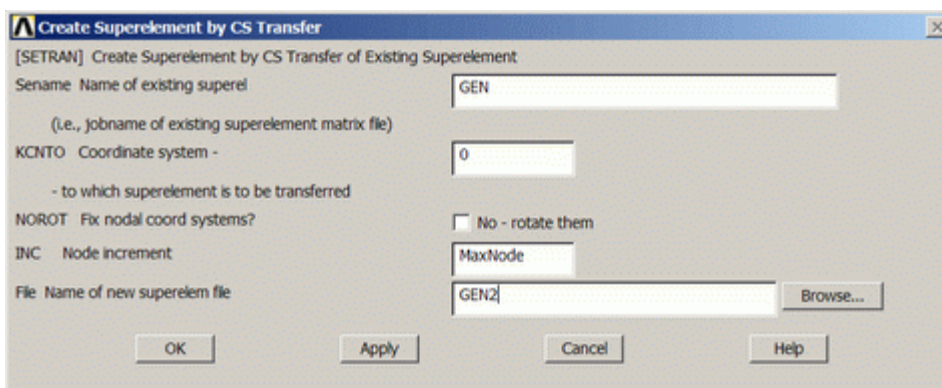


- Fill in the following window as shown to set **MaxNode = the highest node number**



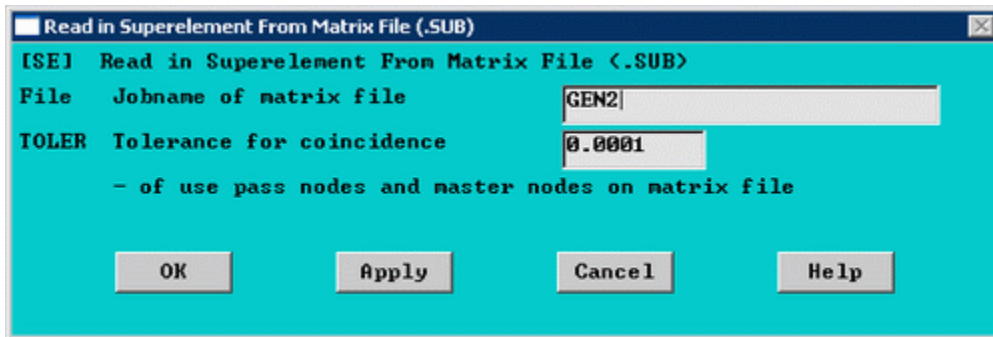
Offset the node numbering

- Select **Preprocessor > Modeling > Create > Elements > Super-elements > BY CS Transfer**
- Fill in the following window as shown to offset the node numbers and save the file as GEN2



Read in the super-element matrix

- Select **Preprocessor > Modeling > Create > Elements > Super-elements > From .SUB File...**
- Enter 'GEN2' as the Jobname of the matrix file in the window (shown below)

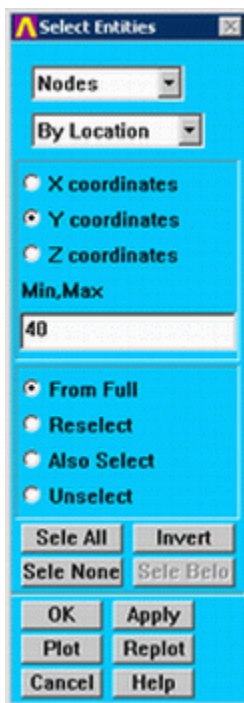


- Utility Menu > Plot > Replot

11. Couple Node Pairs at Interface of Super-element and Non-Superelements

Select the nodes at the interface

- Select **Utility Menu > Select > Entities ...**
- The following window will appear. Select Nodes, By Location, Y coordinates, 40 as shown.



Couple the pair nodes at the interface

- Select **Preprocessor > Coupling / Ceqn > Coincident Nodes**

Re-select all of the nodes

- Select **Utility Menu > Select > Entities ...**
- In the window that appears, click '**Nodes > By Num/Pick > From Full > Sele All**'

Solution Phase: Assigning Loads and Solving

1. Define Analysis Type

Solution > New Analysis > Static
ANTYPE, 0

2. Apply Constraints

Solution > Define Loads > Apply > Structural > Displacement > On Lines

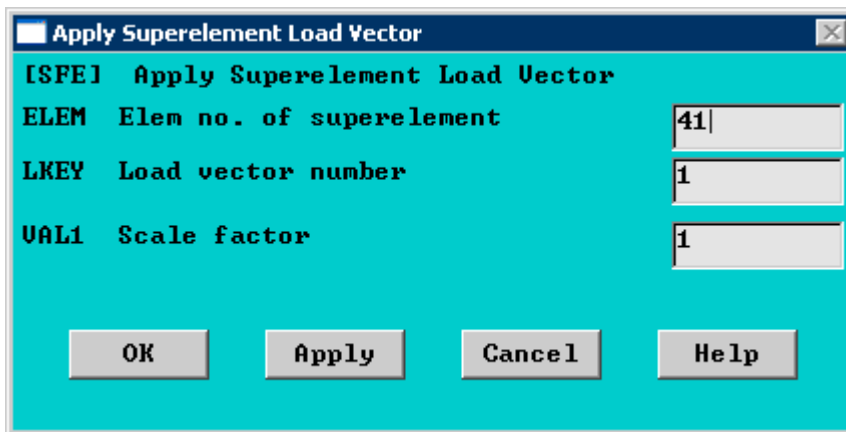
Fix the bottom line (ie all DOF constrained)

3. Apply super-element load vectors

- Determine the element number of the super-element (Select **Utility Menu > PlotCtrls > Numbering...**)

You should find that the super-element is element 41

- Select **Solution > Define Loads > Apply > Load Vector > For Super-element**
- The following window will appear. Fill it in as shown to apply the super-element load vector.



4. Save the database

Utility Menu > File > Save as Jobname.db
SAVE

Save the database to be used again in the expansion pass

5. Solve the System

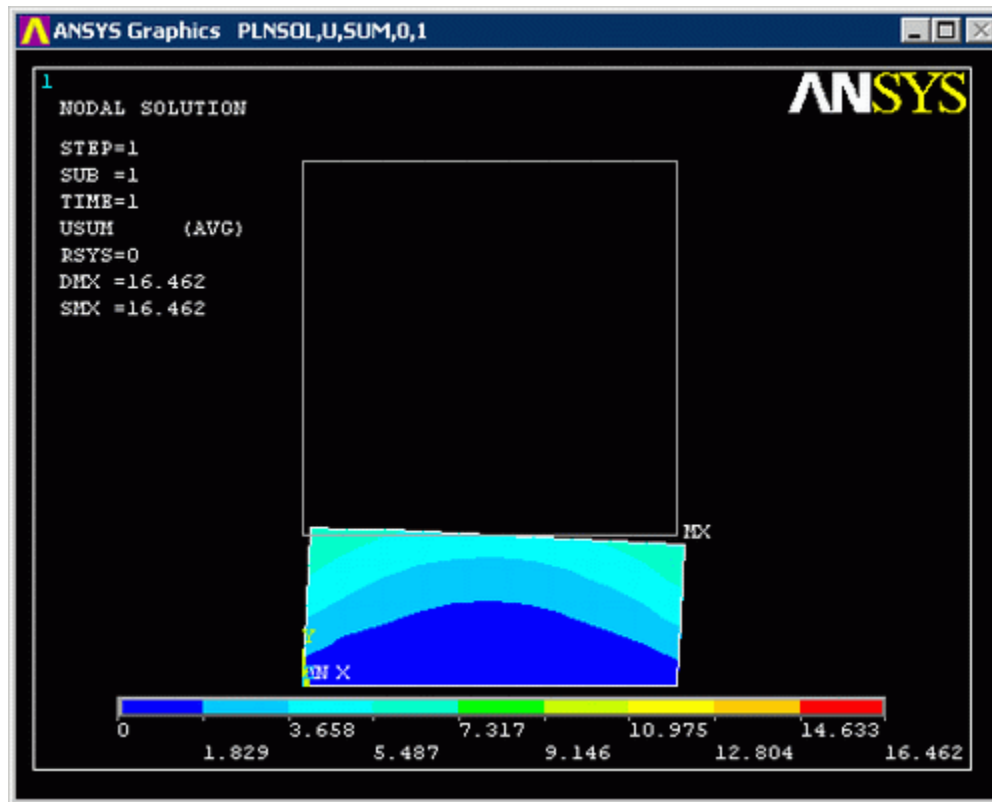
Solution > Solve > Current LS
SOLVE

General Postprocessing: Viewing the Results

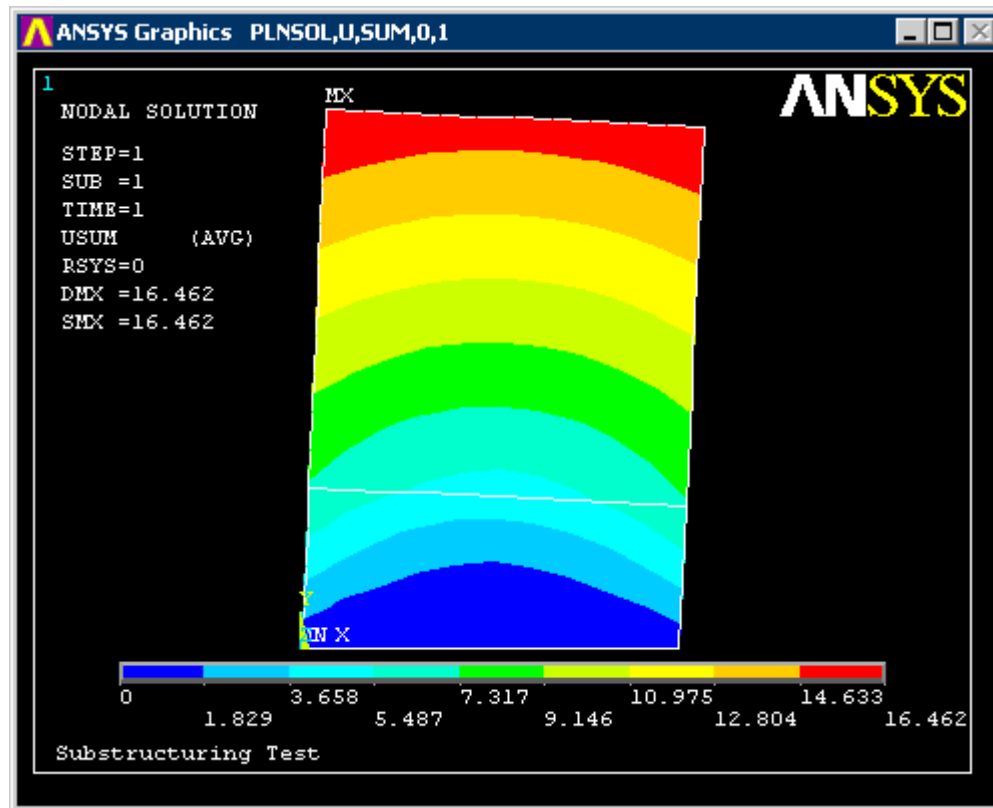
1. Show the Displacement Contour Plot

General Postproc > Plot Results > Contour Plot > Nodal Solution ... > DOF solution, Translation
USUM

PLNSOL,U,SUM,0,1



Note that only the deformation for the non-superelements is plotted. This results agree with what was found without using substructuring (see figure below).



Expansion Pass: Expanding the Results within the Super-element

To obtain the solution for all elements within the super-element you will need to perform an expansion pass.

Preprocessing: Defining the Problem

1. **Clear the existing database**
Utility Menu > File > Clear & Start New
2. **Change the Jobname back to Generation pass Jobname**
Utility Menu > File > Change Jobname ...
FILENAME, GEN

Enter 'GEN' for the jobname
3. **Resume Generation Pass Database**
Utility Menu > File > Resume Jobname.db ...
RESUME

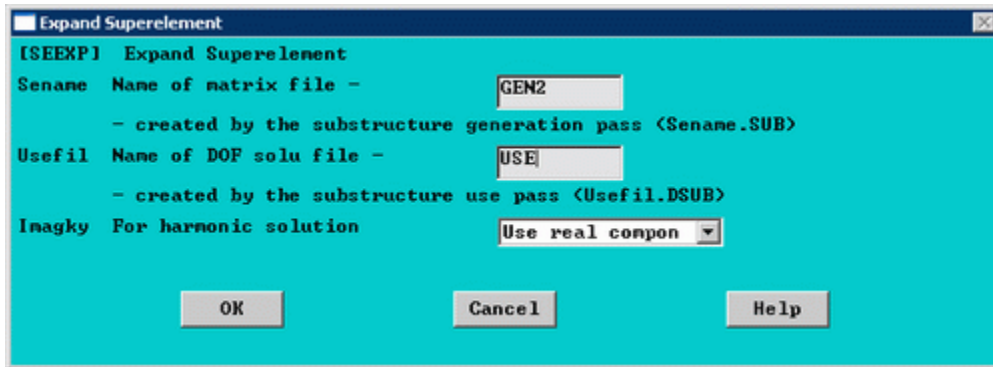
Solution Phase: Assigning Loads and Solving

1. **Activate Expansion Pass**

- Enter the Solution mode by selecting **Main Menu > Solution** or by typing /SOLU into the command line.
- Type 'EXPASS,ON' into the command line to initiate the expansion pass.

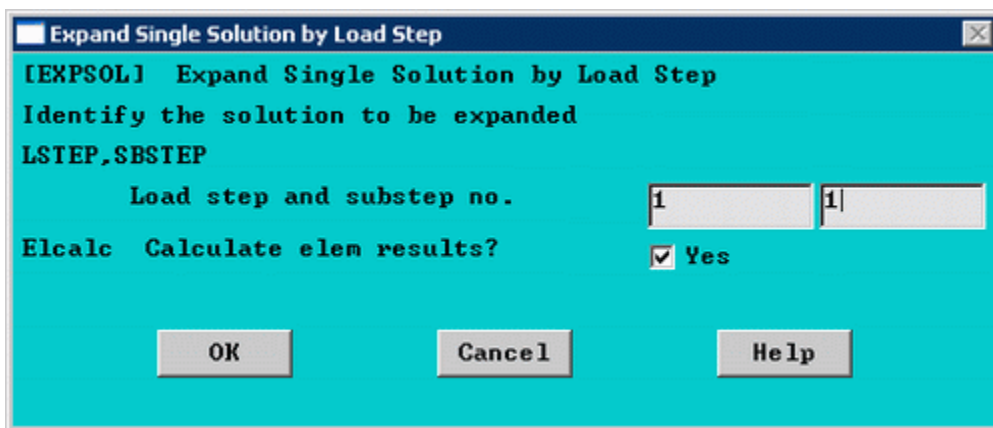
2. Enter the Super-element name to be Expanded

- Select **Solution > Load STEP OPTS > ExpansionPass > Single Expand > Expand Superelem ...**
- The following window will appear. Fill it in as shown to select the super-element.



3. Enter the Super-element name to be Expanded

- Select **Solution > Load Step Opts > ExpansionPass > Single Expand > By Load Step...**
- The following window will appear. Fill it in as shown to expand the solution.



4. Solve the System

Solution > Solve > Current LS
SOLVE

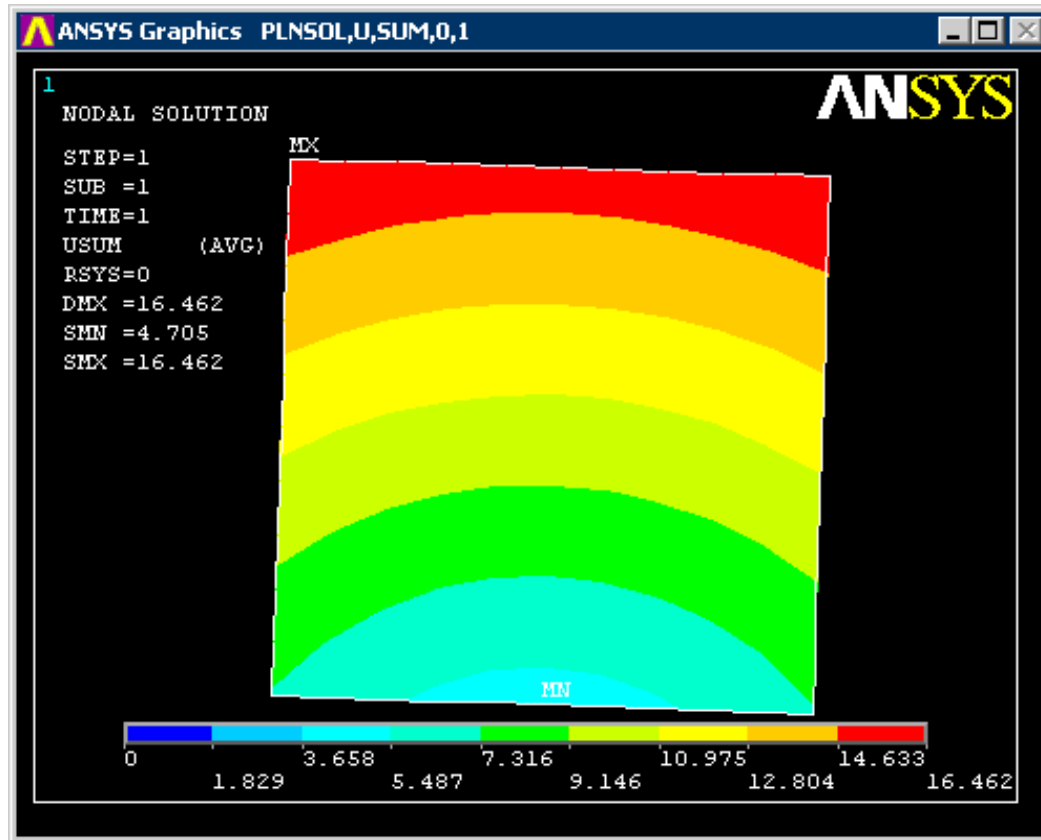
General Postprocessing: Viewing the Results

1. Show the Displacement Contour Plot

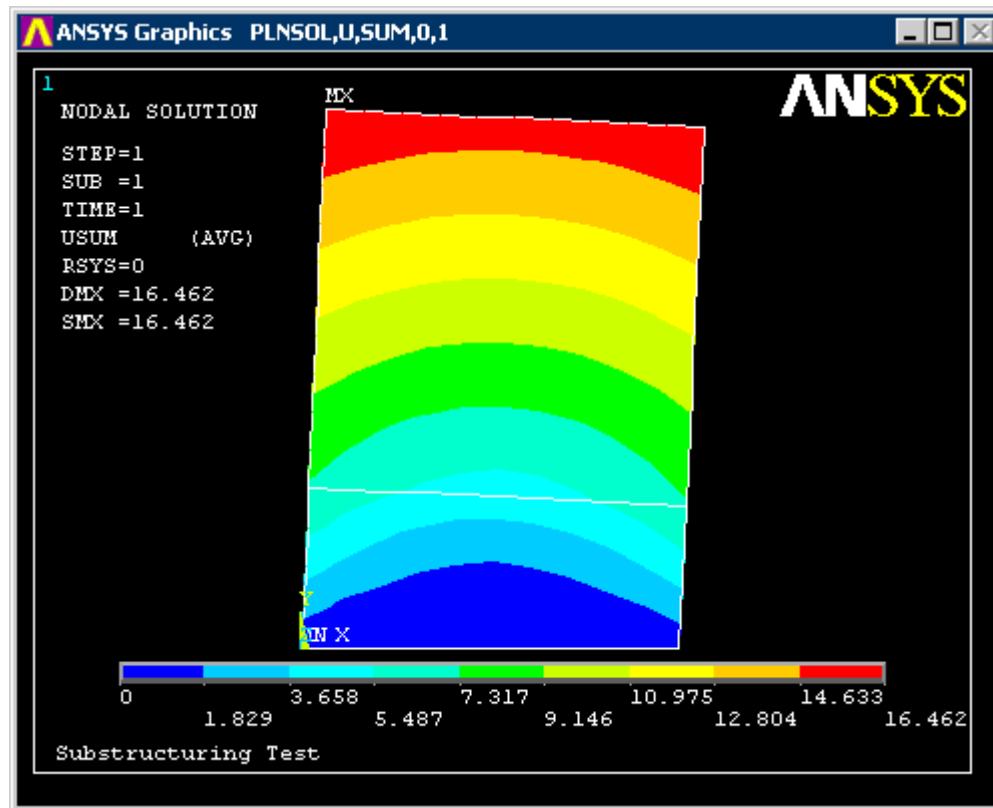
General Postproc > Plot Results > (-Contour Plot-) Nodal Solution ... > DOF solution, Translation

USUM

PLNSOL,U,SUM,0,1



Note that only the deformation for the super-elements is plotted (and that the contour intervals have been modified to begin at 0). This results agree with what was found without using substructuring (see figure below).



Command File Mode of Solution

The above example was solved using the Graphical User Interface (or GUI) of ANSYS. This problem has also been solved using the [ANSYS command language interface](#) that you may want to browse. Open the file and save it to your computer. Now go to '**File > Read input from...**' and select the file.