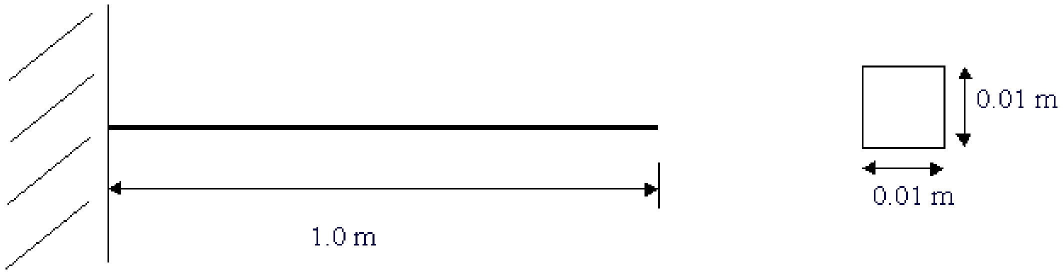
Modal Analysis of a Cantilever Beam

Introduction

This tutorial was created using ANSYS 7.0 The purpose of this tutorial is to outline the steps required to do a simple modal analysis of the cantilever beam shown below.



Modulus of Elasticity (E) = $206800(10^6)$ N/m^2

Density = 7830 kg/m^3

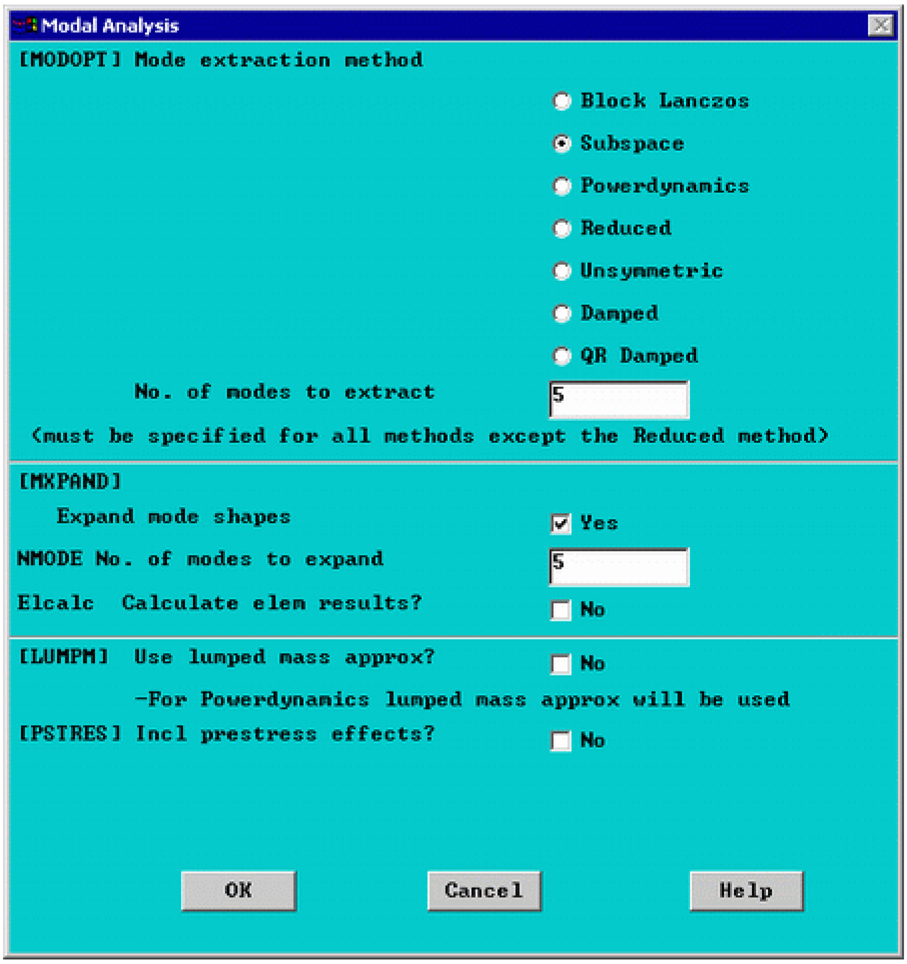
Preprocessing: Defining the Problem

The simple cantilever beam is used in all of the Dynamic Analysis Tutorials. If you haven't created the model in ANSYS, please use the links below. Both the command line codes and the GUI commands are shown in the respective links.

Solution: Assigning Loads and Solving

1. **Define Analysis Type**
   Solution > Analysis Type > New Analysis > Modal
   ANTYPE,2
2. **Set options for analysis type:**
   - Select: **Solution > Analysis Type > Analysis Options..**

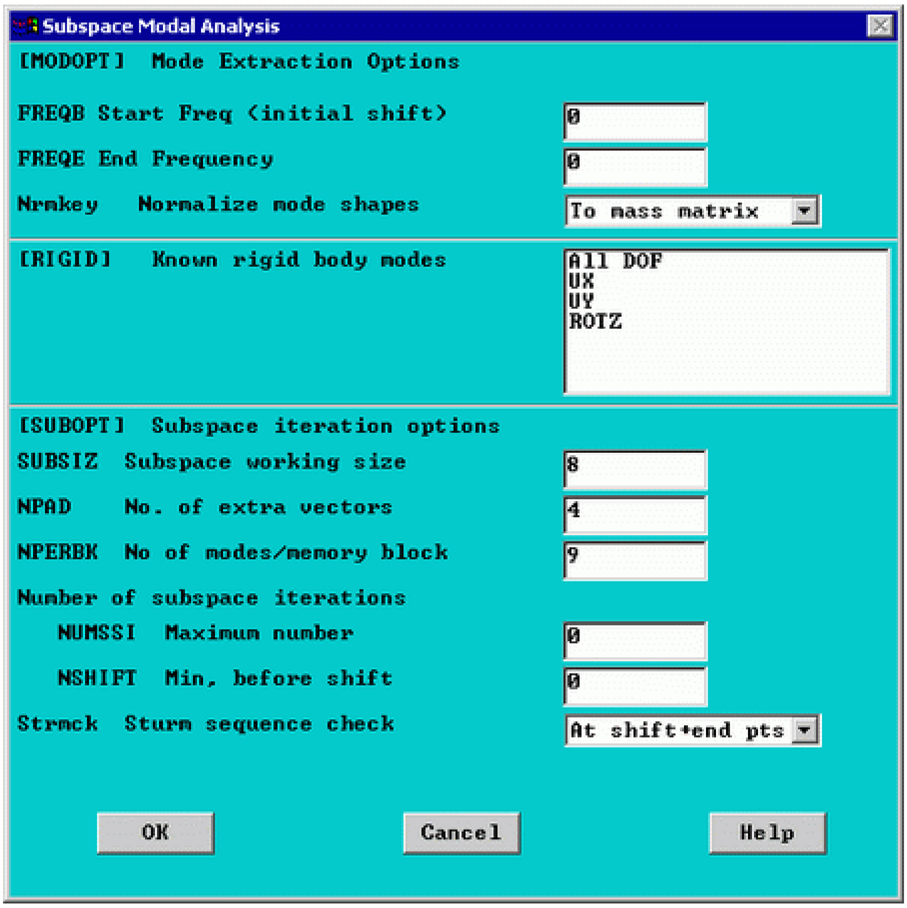
   The following window will appear



- As shown, select the **Subspace** method and enter 5 in the 'No. of modes to extract'
- Check the box beside 'Expand mode shapes' and enter 5 in the 'No. of modes to expand'
- Click 'OK'

Note that the default mode extraction method chosen is the **Reduced Method**. This is the fastest method as it reduces the system matrices to only consider the Master Degrees of Freedom (see below). The **Subspace Method** extracts modes for all DOF's. It is therefore more exact but, it also takes longer to compute (especially when the complex geometries).

- The following window will then appear



For a better understanding of these options see the *Commands* manual.

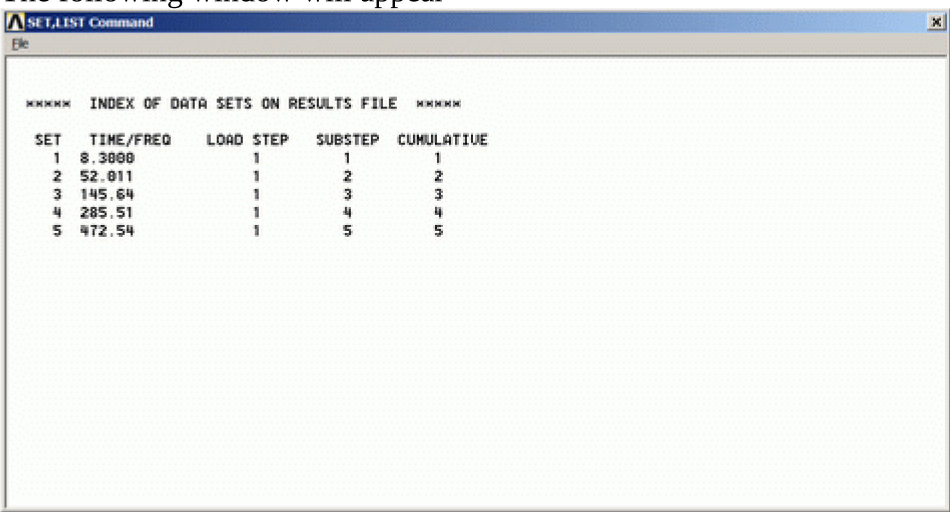
- For this problem, we will use the default options so click on OK.

3. **Apply Constraints**
   Solution > Define Loads > Apply > Structural > Displacement > On Keypoints
   Fix Keypoint 1 (ie all DOFs constrained).
4. **Solve the System**
   Solution > Solve > Current LS
   SOLVE

Postprocessing: Viewing the Results

1. **Verify extracted modes against theoretical predictions**
   - Select: **General Postproc > Results Summary...**

   The following window will appear


SET,LIST Command
File

***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	8.3088	1	1	1
2	52.011	1	2	2
3	145.64	1	3	3
4	285.51	1	4	4
5	472.54	1	5	5

The following table compares the mode frequencies in Hz predicted by theory and ANSYS.

Mode	Theory	ANSYS	Percent Error
1	8.311	8.300	0.1
2	51.94	52.01	0.2
3	145.68	145.64	0.0
4	285.69	285.51	0.0
5	472.22	472.54	0.1

Note: To obtain accurate higher mode frequencies, this mesh would have to be refined even more (i.e. instead of 10 elements, we would have to model the cantilever using 15 or more elements depending upon the highest mode frequency of interest).

2. **View Mode Shapes**

o Select: **General Postproc > Read Results > First Set**

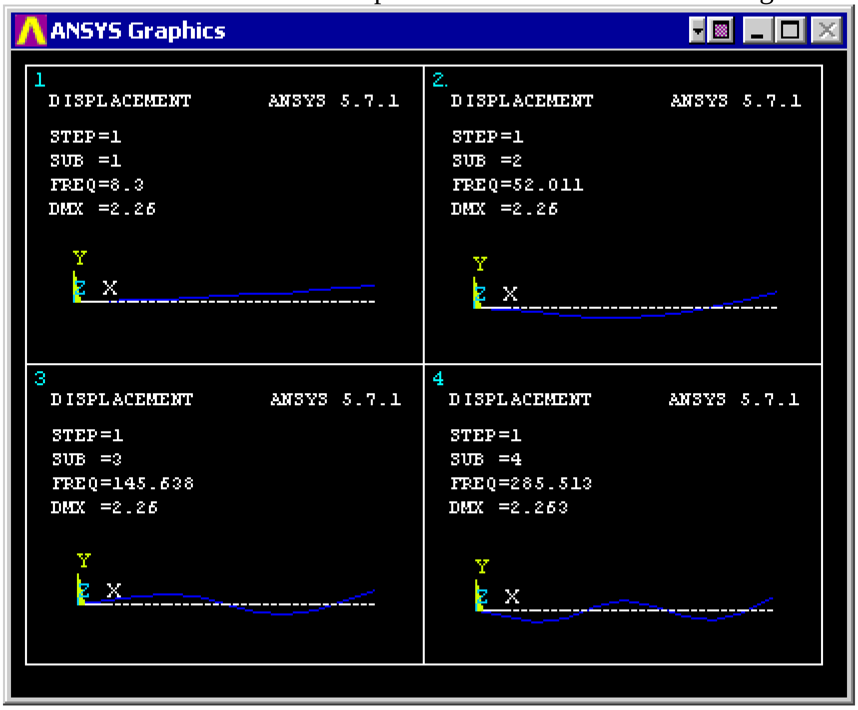
This selects the results for the first mode shape

o Select **General Postproc > Plot Results > Deformed shape** . Select 'Def + undef edge'

The first mode shape will now appear in the graphics window.

o To view the next mode shape, select **General Postproc > Read Results > Next Set** . As above choose **General Postproc > Plot Results > Deformed shape** . Select 'Def + undef edge'.

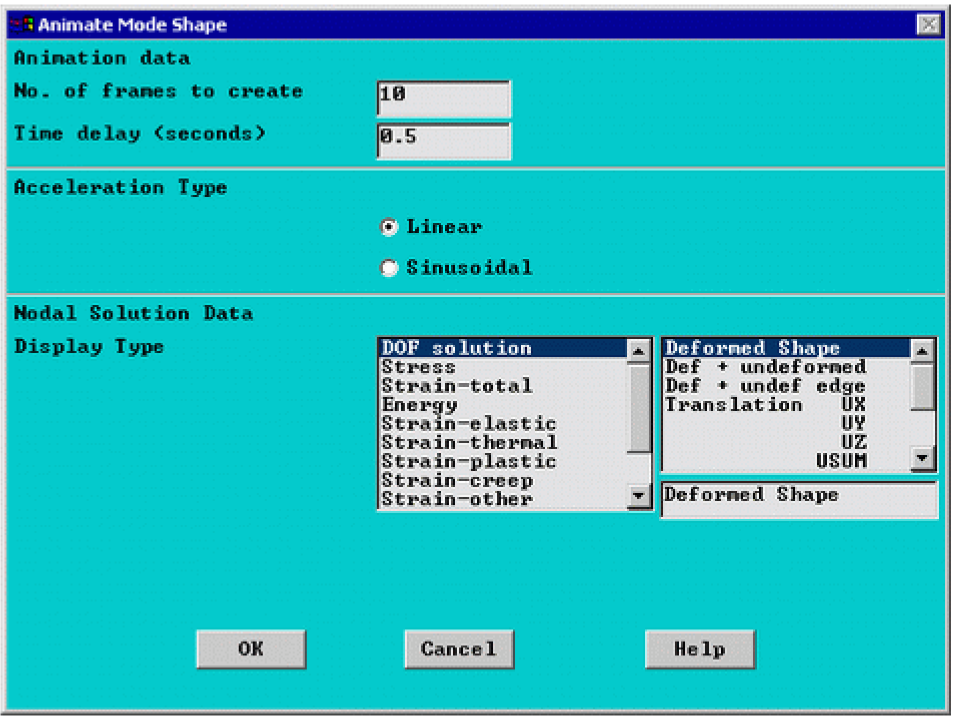
o The first four mode shapes should look like the following:



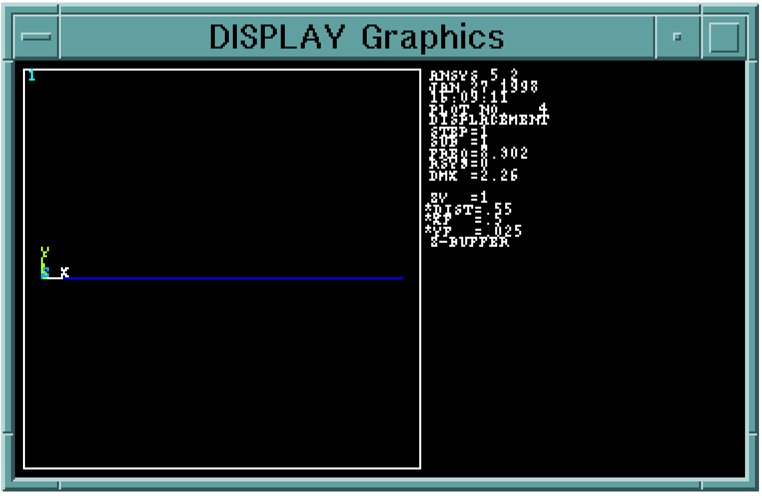
3. **Animate Mode Shapes**

o Select **Utility Menu (Menu at the top) > Plot Ctrls > Animate > Mode Shape**

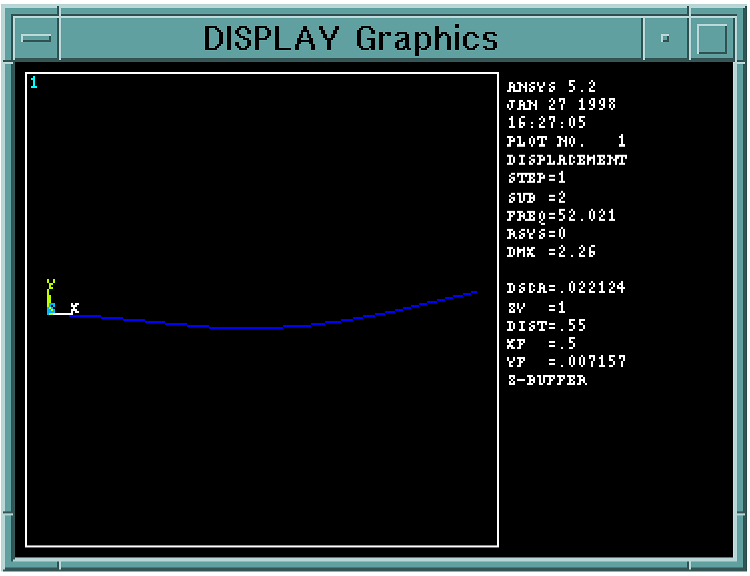
The following window will appear



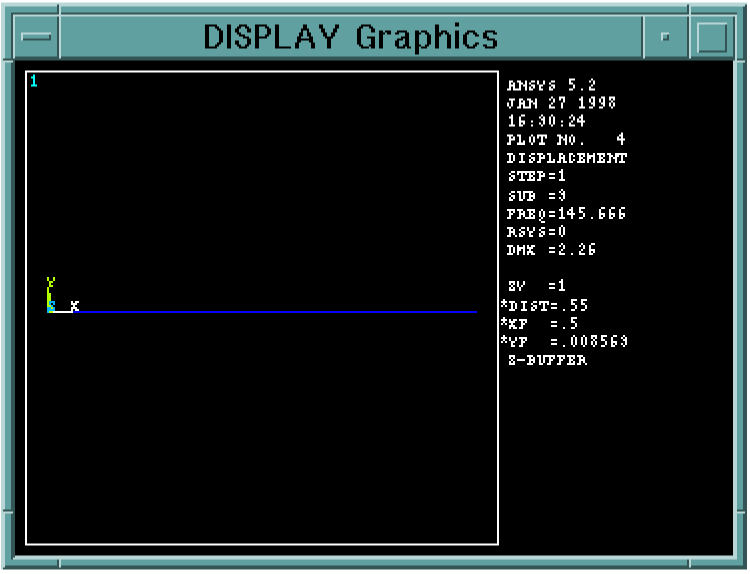
- Keep the default setting and click 'OK'
- The animated mode shapes are shown below.
  - Mode 1



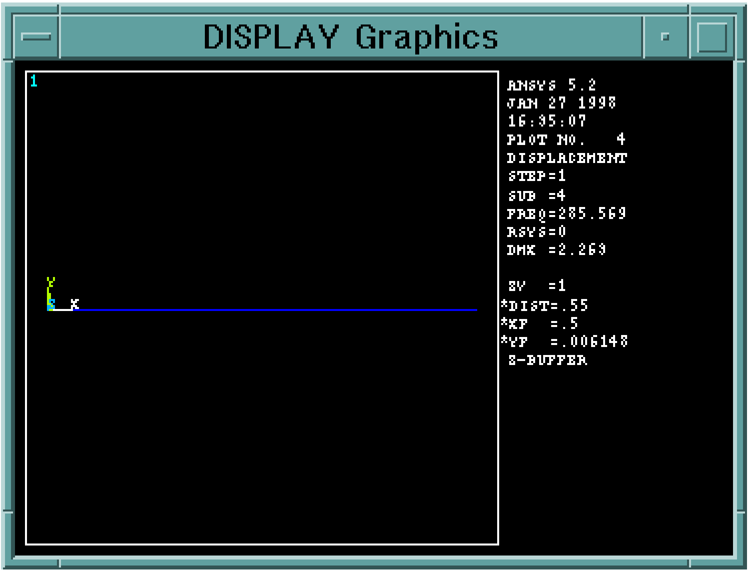
  - Mode 2



- Mode 3

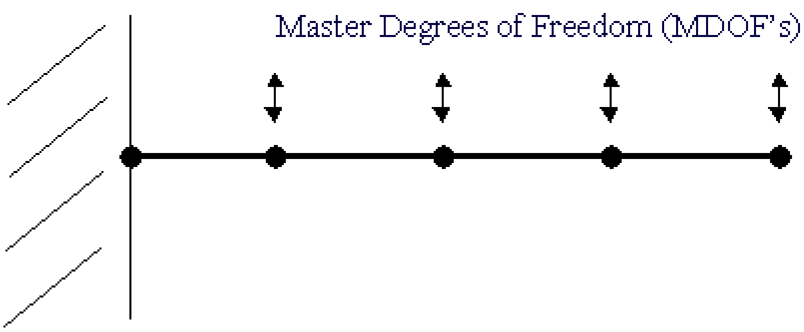


- Mode 4



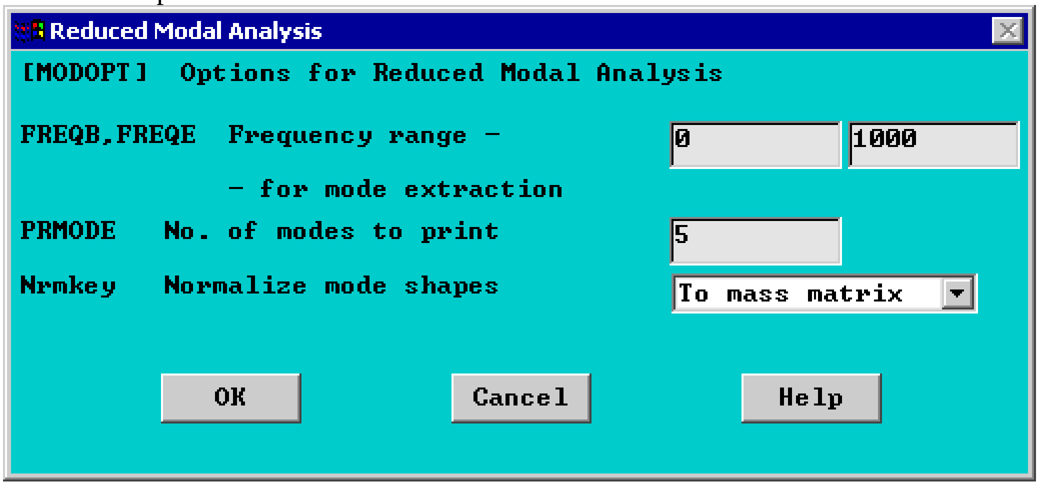
Using the Reduced Method for Modal Analysis

This method employs the use of Master Degrees of Freedom. These are degrees of freedom that govern the dynamic characteristics of a structure. For example, the Master Degrees of Freedom for the bending modes of cantilever beam are



For this option, a detailed understanding of the dynamic behavior of a structure is required. However, going this route means a smaller (reduced) stiffness matrix, and thus faster calculations.
The steps for using this option are quite simple.

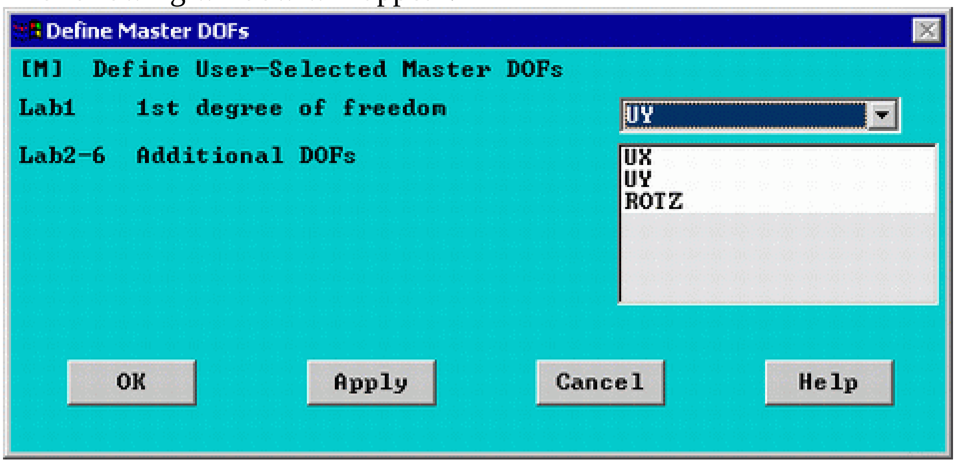
- Instead of specifying the **Subspace** method, select the **Reduced** method and specify 5 modes for extraction.
- Complete the window as shown below



Note: For this example both the number of modes and frequency range was specified. ANSYS then extracts the minimum number of modes between the two.

- Select **Solution** > **Master DOF** > **User Selected** > **Define**
- When prompted, select all nodes except the left most node (fixed).

The following window will appear:



- Select UY as the 1st degree of freedom (shown above).

The same constraints are used as above.

The following table compares the mode frequencies in Hz predicted by theory and ANSYS (Reduced).

Mode	Theory	ANSYS	Percent Error
1	8.311	8.300	0.1
2	51.94	52.01	0.1
3	145.68	145.66	0.0
4	285.69	285.71	0.0
5	472.22	473.66	0.3

As you can see, the error does not change significantly. However, for more complex structures, larger errors would be expected using the reduced method.

Command File Mode of Solution

The above example was solved using a mixture of the Graphical User Interface (or GUI) and the command language interface of ANSYS. This problem has also been solved using the ANSYS command language interface that you may want to browse. Copy and paste following code into Notepad or a similar text editor and save it to your computer. Now go to **'File > Read input from...'** and select the file.

```
FINISH
/CLEAR

/TITLE, Dynamic Analysis
/PREP7

K,1,0,0                         ! Enter keypoints
K,2,1,0

L,1,2                           ! Create line

ET,1,BEAM3                      ! Element type

R,1,0.0001,8.33e-10,0.01        ! Real Const: area,I,height

MP,EX,1,2.068e11                ! Young's modulus
MP,PRXY,1,0.33                  ! Poisson's ratio
MP,DENS,1,7830                  ! Density

LESIZE,ALL,,,10                 ! Element size
LMESH,1                         ! Mesh line

FINISH
/SOLU

ANTYPE,2                        ! Modal analysis
MODOPT,SUBSP,5                  ! Subspace, 5 modes
EQSLV,FRONT                     ! Frontal solver
MXPAND,5                        ! Expand 5 modes

DK,1,ALL                        ! Constrain keypoint one

SOLVE
FINISH

/POST1                          ! List solutions
SET,LIST

SET,FIRST
PLDISP                          ! Display first mode shape

ANMODE,10,0.5, ,0               ! Animate mode shape
```

Reference

http://www.mece.ualberta.ca/tutorials/ansys