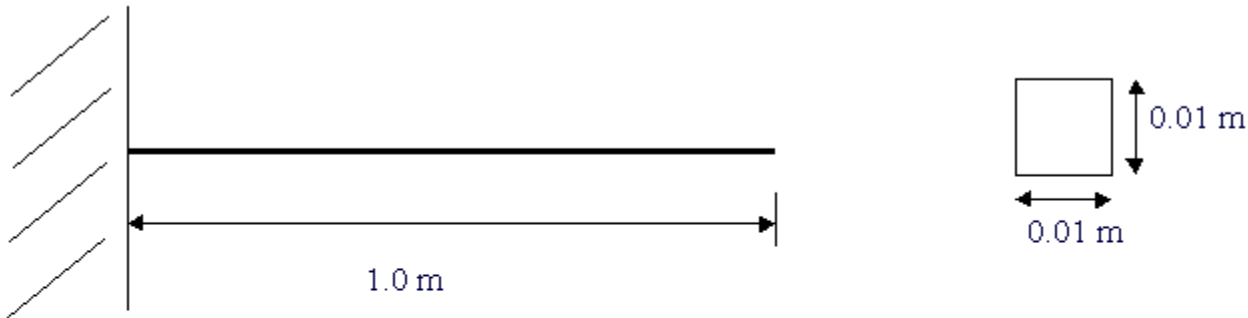


Modal Analysis of a Cantilever Beam

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/m²

Density = 7830 kg/m³

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

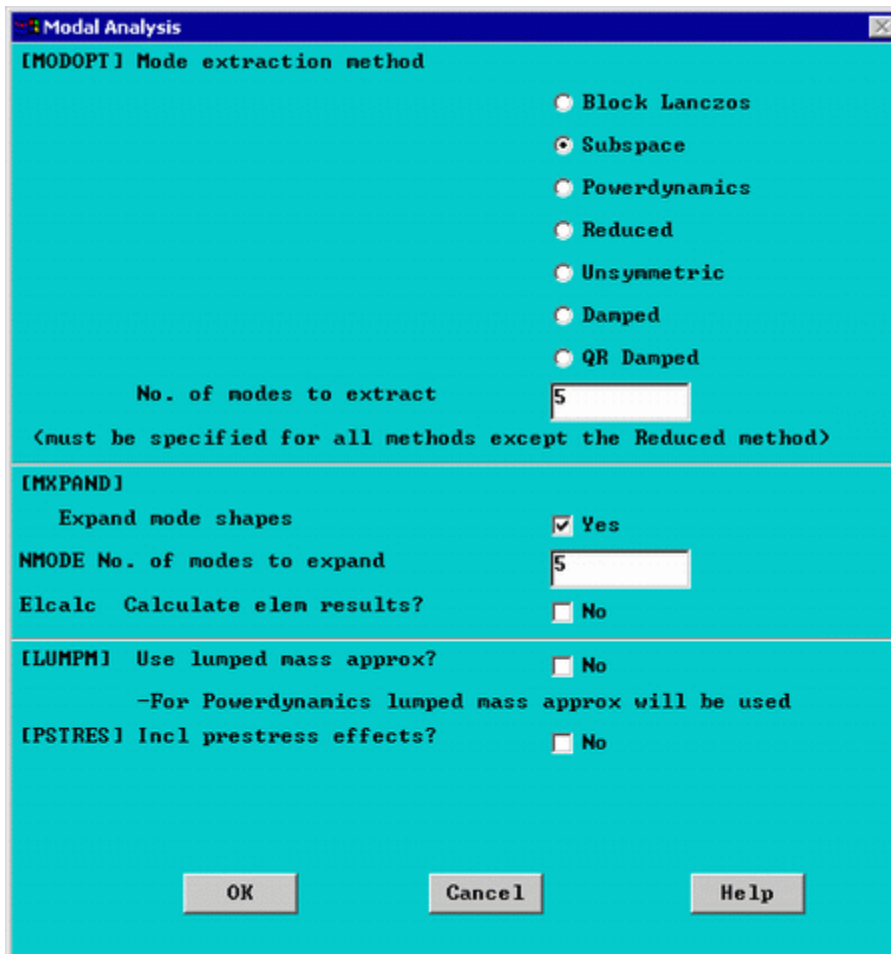
1. Define Analysis Type

Solution > Analysis Type > New Analysis > Modal
ANTYPE, 2

2. Set options for analysis type:

- Select: **Solution > Analysis Type > Analysis Options..**

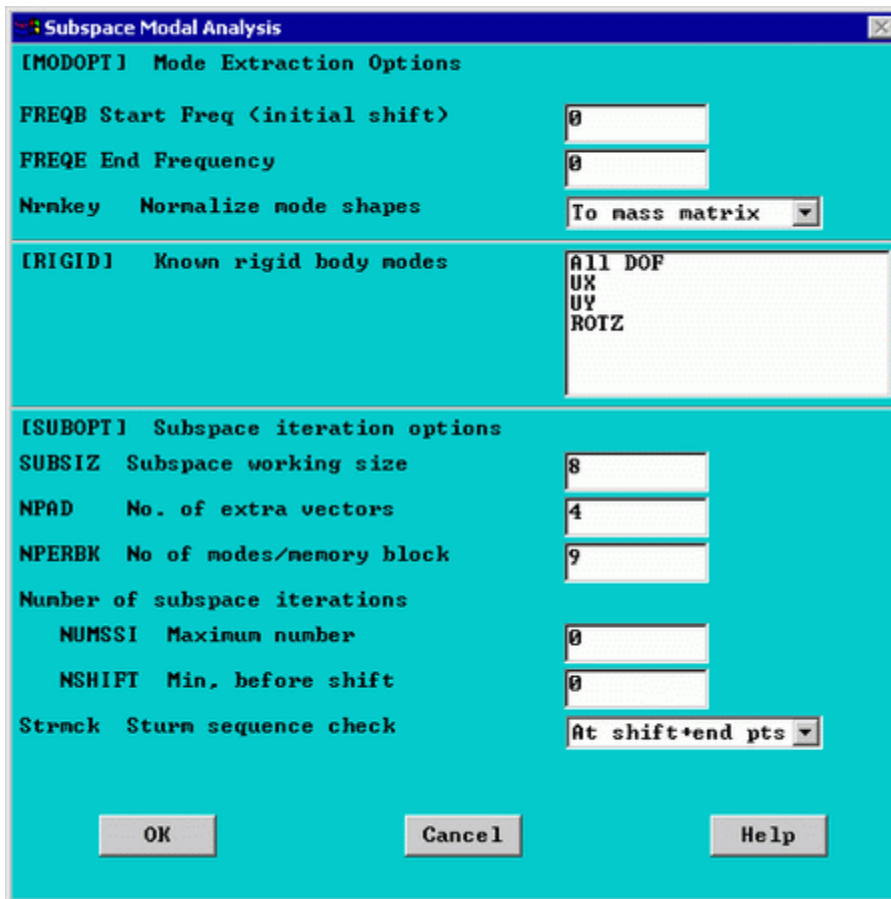
The following window will appear



- 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

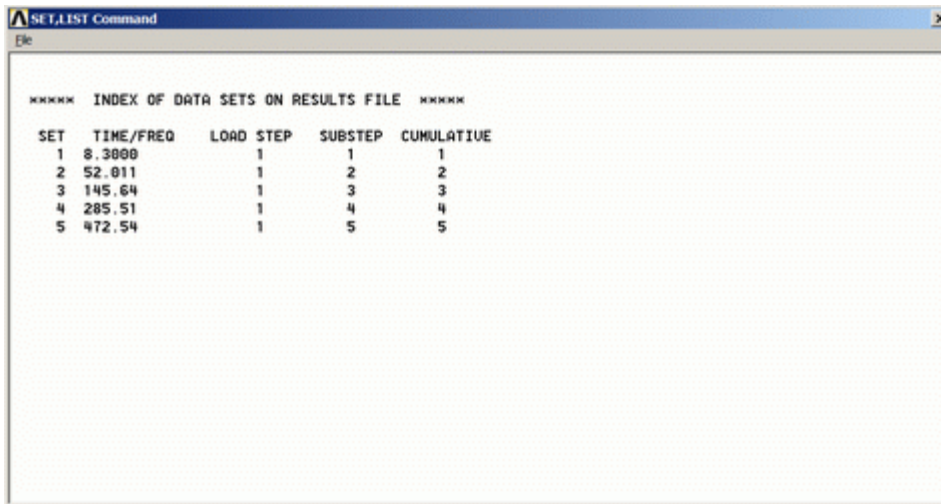
Solution > Solve > Current LS
SOLVE

Postprocessing: Viewing the Results

1. Verify extracted modes against theoretical predictions

- Select: **General Postproc > Results Summary...**

The following window will appear



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

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	8.3000	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

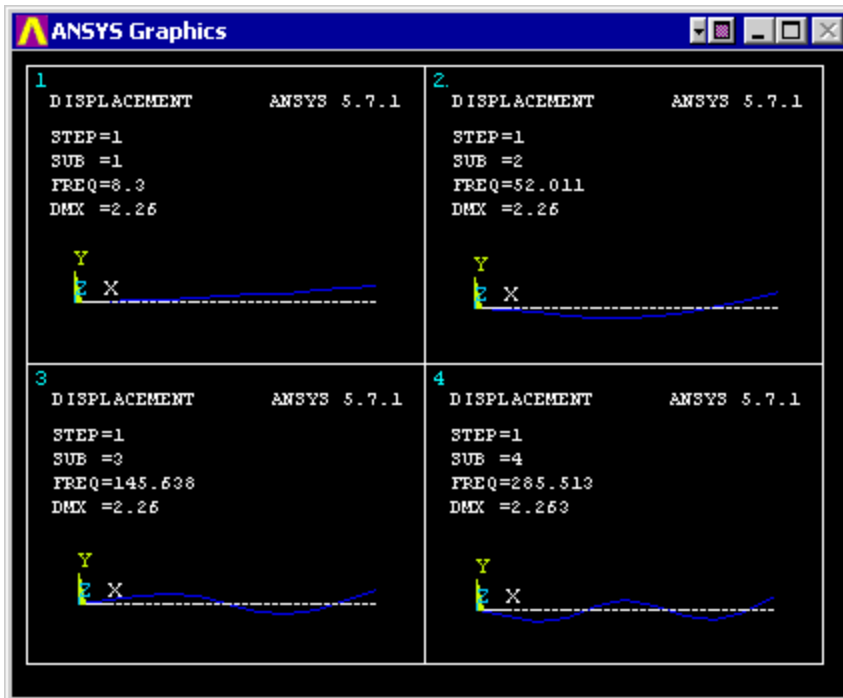
- Select: **General Postproc > Read Results > First Set**

This selects the results for the first mode shape

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

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

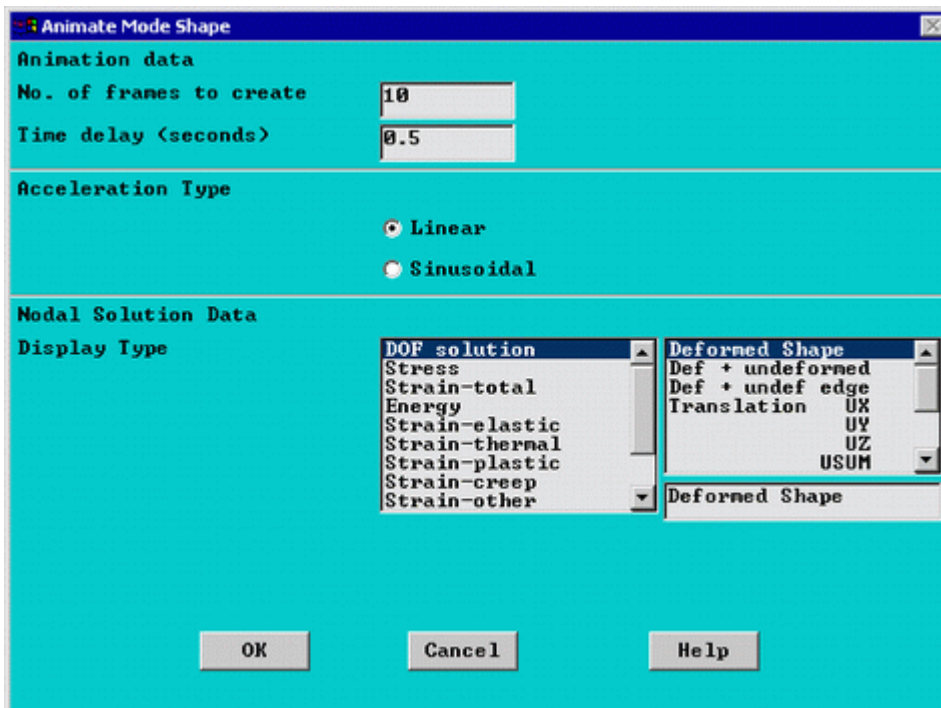
- To view the next mode shape, select **General Postproc > Read Results > Next Set** . As above choose **General Postproc > Plot Results > Deformed shape** . Select 'Def + undef edge'.
- The first four mode shapes should look like the following:



3. Animate Mode Shapes

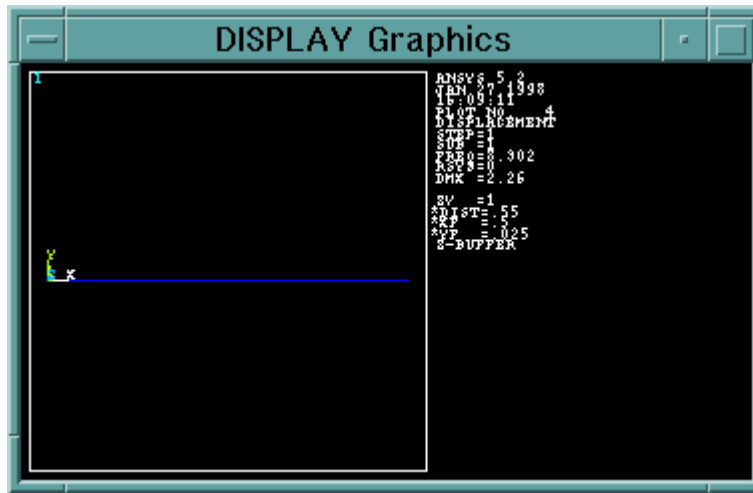
- o Select **Utility Menu (Menu at the top) > PlotCtrls > Animate > Mode Shape**

The following window will appear

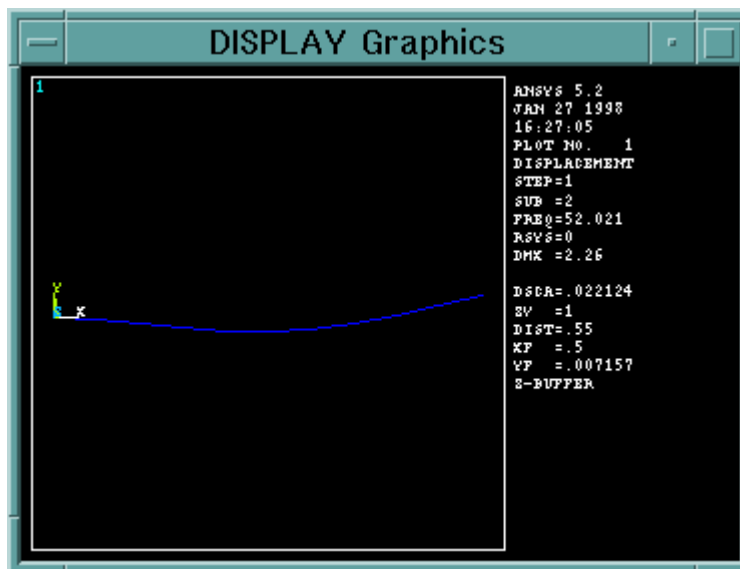


- o Keep the default setting and click 'OK'
- o The animated mode shapes are shown below.

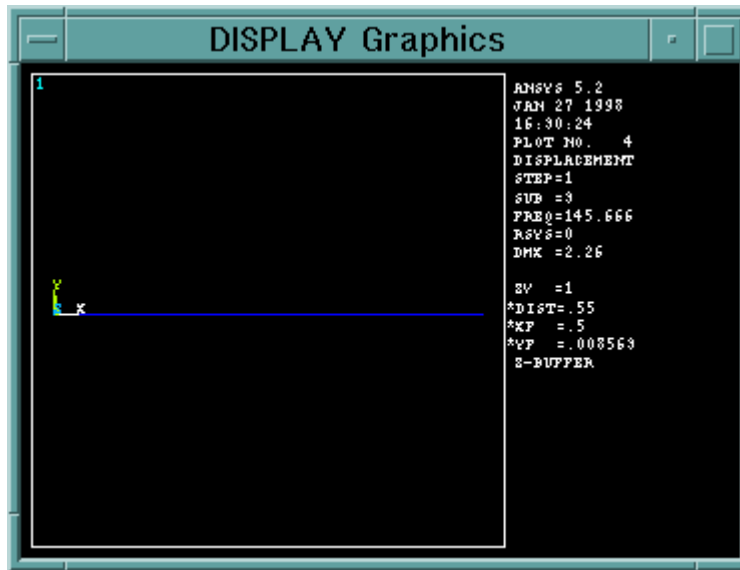
- Mode 1



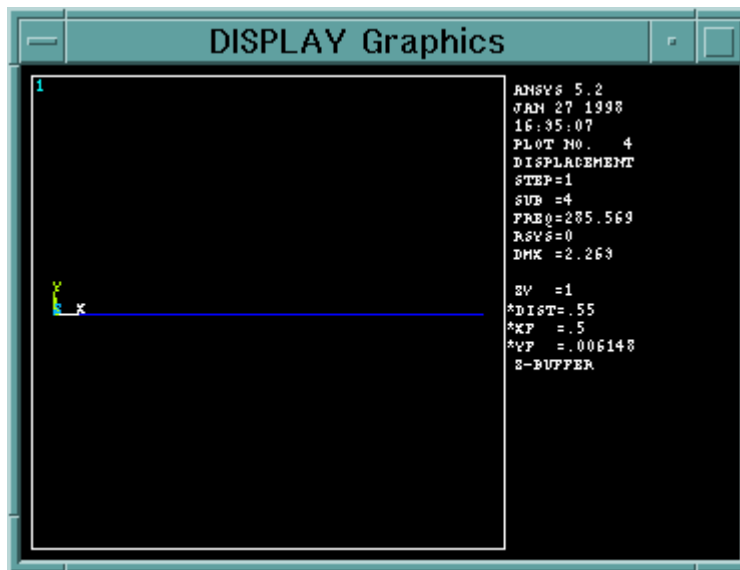
- Mode 2



- Mode 3

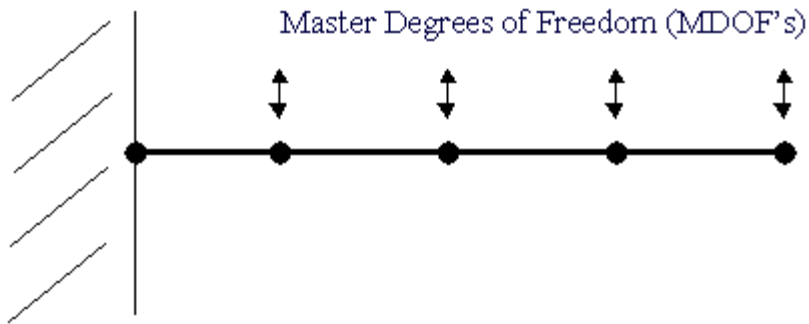


- Mode 4



Using the Reduced Method for Modal Analysis

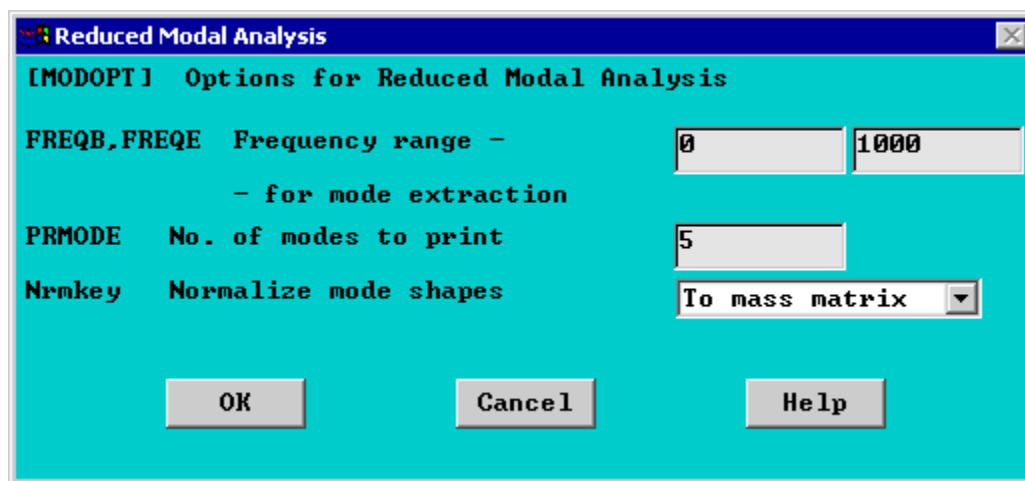
This method employs the use of Master Degrees of Freedom. These are degrees of freedom that govern the dynamic characteristics of a structure. For example, the Master Degrees of Freedom for the bending modes of cantilever beam are



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.

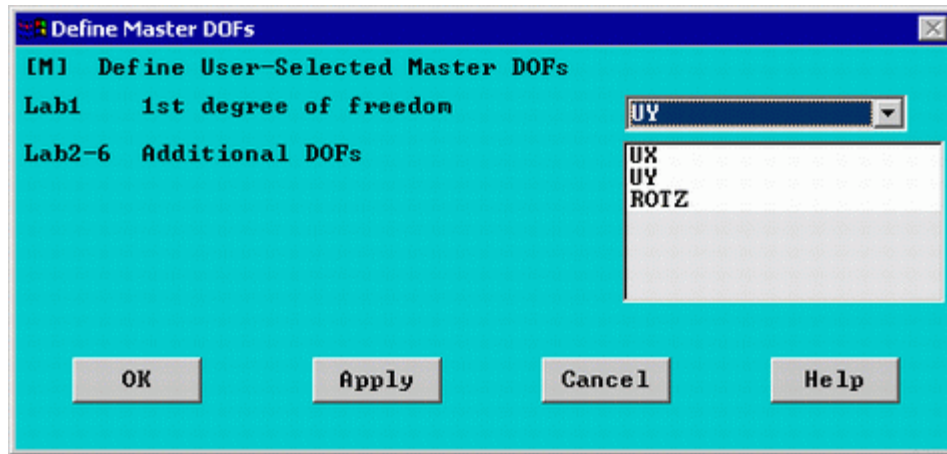
- Instead of specifying the **Subspace** method, select the **Reduced** method and specify 5 modes for extraction.
- Complete the window as shown below



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.