Here is a tutorial to implement a simple Modal Analysis of a Cantilever Cylinder

1. Open the ANSYS workbench

2. Drop and drag the "Modal" analysis system into the project schematic

3. Right click on "Engineering Data" and edit the material. There are predetermined properties found in the "Engineering Data Sources" (Right click on the description or press the books in the top right hand corner). For this tutorial, fused silica was created with these parameters:

* Young's Modulus E: 7.2E10 Nm^-2 (Isotropic Elasticity)
* Density p: 2200 kgm^-3 (Density)
* Specific Heat c: 770 J/kgK (Specific Heat)
* Isotropic Thermal Conductivity k: 1.38 W/mk (Isotropic Thermal Conductivity)
* Coefficient of Thermal Expansion: 3.9E-7 (Isotropic Secant Coefficient of Thermal Expansion)
* Reference Temperature: 21 C (Isotropic Secant Coefficient of Thermal Expansion)
* Poisson's Ratio: 0.17 (Isotropic Elasticity)

In order to put in the parameters, just drag and drop from the toolbox. Also make sure the filter is turned off (that's the filter symbol next to the engineering data sources/books in the top right hand corner) this will allow a user to few all the parameters.

4. Return to the project, and open/edit the geometry. The cantilever that was used for this tutorial is a cylinder with a diameter of 2 mm with a length of 200 mm, shown in image 1.

* How to Create a geometry:
  + Open up the geometry
  + Decide which plane you want to start drawing in: XY Plane, ZX Plane, YZ Plane
  + Click on "new sketch" in the 4th row from the top after "Sketch1" it's blue sketch
  + Whatever plane you decide to choose, press the "Look at Face/Plane/Sketch" icon in the top right hand corner (The person looking at a face)
  + This will allow the view to go to the view where you want to sketch on
  + Click on "Sketching" which is next to the "Modeling" right above the "Details View"
  + Here will be Several Options
    - Draw: This is where you can decide what shape you want to use
    - Modify: This is the extras such as fillets, chamfers and corners
    - Dimensions: This is where you can select what type of dimensions you will be using
    - Constraints: This is where you can constrain lines or connect lines
      * When you want to connect lines together, click on each end and make them "Coincident" this way the shape will connect when you change dimensions
    - Settings: This is where you can show a grid or snap grid
  + Sketch Color:
    - Teal: Underdefined: The sketch does not have enough dimensions to constrain and tell the program what exactly the shape is
    - Blue: Defined: The sketch is fully defined and does not need any more dimensions
    - Red: Overdefined: The sketch has repeating dimensions that causes errors
  + For this tutorial, draw a circle. I made mine coincident to the middle by pressing the vertex in the middle and one of the axis and then repeat the process with the second axis to center the circle
  + Dimensions: The diameter is 2 mm, under dimensions to display the number, click "Display" and then "Value" instead of "name"
  + To change the dimensions, the "Details View" in the bottom left hand corner will say D1 and from there you can change the diameter
  + From there in order to create the length of the cylinder, press "Extrude" at the top (4th row) towards the right. From there it will ask what geometry, click on your sketch and click "Apply". Make sure the "Operation" is "Add Material" to Extrude.
  + Then choose a "Depth" in this case it's 200 mm. Then click "Generate" the lightning bolt next to the "New Sketch" symbol.
  + Your cylinder should look like image 1.

5. Return to the project and open the Model tab. With that open, go to the subtab of "Geometry", named "Solid". Under Solid go to "Assignment" under "Material" to assign Fused Silica or whatever material you decide to choose.

6. Go to the "Mesh" tab. In this case the "Element Size" under the "Sizing" tab is 0.04 and the "Relevance Center" as "Fine". Right click on the mesh and press "Generate mesh".

7. Under the Modal tab, go to the Analysis Settings and change the "Max Modes to Find" for ANSYS to calculate. In this tutorial, the amount of modes that were used was 17.

8. Right Click the Modal and press on "Fixed Support". This will make the bar a cantilever bar, once the geometry is set to fix one face of the bar. Press on one of the faces and click on "apply" on the Geometry.

9. Right click on the solution and press "Solve". Let ANSYS run the modes through.

10. On the Solution tab, the "Tabular Data" is listed but the Total Deformation has not been listed. In order to do so, select all the frequency of the modes in the column and right click and press "Create Mode Shape Results".

11. Once loaded, the total deformation will have a lightning bolt next to each entry, right click and press "Solve" or "Evaluate All Results". This will make all the entries have a green check mark.

12. This will give the user the ability to animate each entry. This is down by clicking the play (sideways triangle button). ANSYS will run through the simulation for that mode that is selected, shown in image 2.

13. Using Mathematica (or another computational system) input the analytical solution for a Cantilever bar fixed to one end.

* wn=(knL)2Sqrt(EI/mL4)
  + Where:
  + wn is the frequency measured in radians/second
  + E = Young's Modulus
  + I = moment of cross section
    - Irectangle = ba3/12 (b and a are the sides)
    - Icircle = (Pi/64)(d4) (d is the diameter)
  + m = mass per unit length
  + L = length of the bar
  + kn relates to the amount of nodes
    - k1L = 1.875, k2L = 4.694, k3L = 7.855, k4L = 10.996, k5L = 14.137
    - n greater than 5: knL = (2n -1)(Pi/2)
* Convert wn to f measured in Hz

14. Compare only to the bar actually bending, not twisting or contracting. There are modes that are the same due to the symmetry of the bar. In image 3 the underlined frequencies compare to the analytical calculations (Mathematica) and the computational calculations (ANSYS).