11 Vibration Analysis

11.1 Introduction

A spring and a mass interact with one another to form a system that resonates at their characteristic natural frequency. If energy is applied to a spring-mass system, it will vibrate at its natural frequency. The level of a general vibration depends on the strength of the energy source as well as the damping inherent in the system. Consider the single degree of freedom (DOF) system in Figure 11-1 that is usually introduced in a first course in physics or ordinary differential equations. There, k is the spring constant, or stiffness, and m is the mass, and c is a viscous damper. If the system is subjected to a horizontal force, say $f(t)$, then Newton's law of motion leads to the differential equation of motion in terms of the displacement as a function of time, $x(t)$:

$$m\, d^2x / dt^2 + c\, dx / dt + k\, x(t) = f(t)$$

which requires the initial conditions on the displacement, $x(0)$, and velocity, $v(0) = dx / dt(0)$. When there is no external force and no damping, then it is called free, undamped motion, or simple harmonic motion (SHM):

$$m\, d^2x / dt^2 + k\, x(t) = 0.$$

The usual simple harmonic motion assumption is $x(t) = a \sin(\omega t)$ where a is the amplitude of motion and ω is the circular frequency of the motion. Then the motion is described by

$$[k - \omega^2 m]\, a \sin(\omega t) = 0, \text{ or } [k - \omega^2 m] = 0.$$

The above equation represents the simplest eigen-analysis problem. There you wish to solve for the eigenvalue, ω, and the eigenvector, a. Note that the amplitude, a, of the eigenvector is not known. It is common to scale the eigenvector to make the largest amplitude unity. The above scalar problem is easily solved for the circular frequency (eigenvalue),

$$\omega = 2\pi F_n = \sqrt{k/m},$$

which is related to the so called natural frequency, F_n, by $F_n = \omega / 2\pi$.

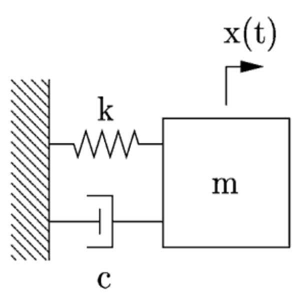


Figure 11-1 A spring-mass-damper single degree of freedom system

From this, it is seen that if the stiffness increases, the natural frequency also increases, and if the mass increases, the natural frequency decreases. If the system has damping, which all physical systems do, its frequency of response is a little lower, and depends on the amount of damping. Numerous tabulated solutions for natural frequencies and mode shape can be found in [3]. They can be useful in validating finite element calculations.

Note that the above simplification neglected the mass of both the spring and the dampener. Any physical structure vibration can be modeled by springs (stiffnesses), masses, and dampers. In elementary models you use line springs and dampers, and point masses. It is typical to refer to such a system as a "lumped mass system". For a continuous part, both its stiffness and mass are associated with the same volume. In other words, a given volume is going to have a strain energy associated with its stiffness and a kinetic energy associated with its mass. A continuous part has mass and stiffness matrices that are of the same size (have the same number of DOF). The mass contributions therefore interact and can not naturally be lumped to a single value at a point. There are numerical algorithms to accomplish such a lumped (or diagonal) mass matrix but it does not arise in the consistent finite element formulation.

11.2 Finite element vibration studies

In finite element models, the continuous nature of the stiffness and mass leads to the use of square matrices for stiffness, mass, and damping. They can still contain special cases of line element springs and dampers, as well as point masses. Dampers dissipate energy, but springs and masses do not.

If you have a finite element system with many DOF then the above single DOF system generalizes to a displacement vector, $\boldsymbol{X}(t)$ interacting with a square mass matrix, $\boldsymbol{M}$, stiffness matrix, $\boldsymbol{K}$, damping matrix $\boldsymbol{C}$, and externally applied force vector, $\boldsymbol{F}(t)$, but retains the same general form:

$$\boldsymbol{M}\, d^2\boldsymbol{X} / dt^2 + \boldsymbol{C}\, d\boldsymbol{X} / dt + \boldsymbol{K}\, \boldsymbol{X}(t) = \boldsymbol{F}(t)$$

plus the initial conditions on the displacement, $\boldsymbol{X}(0)$, and velocity, $\boldsymbol{v}(0) = d\boldsymbol{X} / dt(0)$. Integrating these equations in time gives a *time history solution*. The solution concepts are basically the same, they just have to be done using matrix algebra. The corresponding SHM, or free vibration mode ($\boldsymbol{C} = \boldsymbol{0}, \boldsymbol{F} = \boldsymbol{0}$) for a finite element system is

$$\boldsymbol{M}\, d^2\boldsymbol{X} / dt^2 + \boldsymbol{K}\, \boldsymbol{X}(t) = \boldsymbol{0}.$$

The SHM assumption generalizes to $\boldsymbol{X}(t) = \boldsymbol{A} \sin(\omega t)$ where the amplitude, $\boldsymbol{A}$, is usually called the mode shape vector at circular frequency ω. This leads to the general matrix *eigenvalue problem*

$$| \boldsymbol{K} - \omega^2 \boldsymbol{M} | = 0.$$

There is a frequency, say ω_k, and mode shape vector, $\boldsymbol{A}_k$, for each degree of freedom, k. A matrix eigenvalue-eigenvector solution is much more computationally expensive that a matrix time history solution. Therefore most finite element systems usually solve for the first few natural frequencies. Depending on the available computer power, that may mean 10 to 100 frequencies. SW Simulation includes natural frequency and mode shape calculations as well as time history solutions.

Usually you are interested only in the first few natural frequencies. In SW Simulation, the default number of frequencies to be determined is five (that number is controlled via **Study→Properties→Options→ Number of frequencies**). A zero natural (or slightly negative one) frequency corresponds to a rigid body motion. A part or assembly has at most six RBM of 'vibration', depending on how or if it is supported. If a shell model is used the rotational DOF exist and the mass matrix is generalized to include the mass moments of inertia. For every natural frequency there is a corresponding vibration mode shape. Most mode shapes can generally be described as being an axial mode, torsional mode, bending mode, or general mode

Like stress analysis models, probably the most challenging part of getting accurate finite element natural frequencies and mode shapes is to get the type and locations of the restraints correct. A crude mesh will give accurate frequency values, but not accurate stress values. The TK Solver case solver software contains equations for most known analytic solutions for the frequencies of mechanical systems. They can be quite useful in validating the finite element frequency results.

In section 3.3, the stiffness matrix for a linear axial bar was given. It is repeated here along with its consistent mass matrix:

$$[k] = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}, [m] = \frac{m}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}, \text{m} = \rho \text{AL}.$$

If you utilize a quadratic (three node) line element the corresponding element matrices are

$$[k] = \frac{EA}{3L}\begin{bmatrix} 7 & -8 & 1 \\ -8 & 16 & -8 \\ 1 & -8 & 7 \end{bmatrix},\ [m] = \frac{m}{30}\begin{bmatrix} 4 & 2 & -1 \\ 2 & 16 & 2 \\ -1 & 2 & 4 \end{bmatrix}.$$

11.3 Analytic solutions for frequencies

The analytic frequency and mode shape solutions for many common geometries are found In a course on the vibration of continuous media. The geometries include axial bars, axial shafts in torsion, beams with transverse motion vibration, flat plates of various shapes, and thin shells of various shapes. Several examples of them are given in the “validation problems” set of examples presented along side the software tutorials. of

Consider the longitudinal vibration of a bar. The results depend on which type of support is applied to each end of the bar. For one end restrained and the other end free the natural frequencies are

$$\omega_n = \frac{(2n-1)\pi c}{2L},\ c = \sqrt{\frac{E}{\rho}},\ n = 1, 2, 3, \ldots \infty.$$

However, if both ends are restrained they are

$$\omega_n = \frac{n\pi c}{L},\ c = \sqrt{\frac{E}{\rho}},\ n = 1, 2, 3, \ldots \infty.$$

This shows that for a continuous body there are, in theory, an infinite number of natural frequencies and mode shapes. Try a single quadratic element to model a fixed-fixed bar frequency. Restrain the two end DOF (the first and third row and column) of the above 3 by 3 matrices. Only a single DOF remains to approximate the first mode. Solve the restrained matrix eigen-problem: $[\mathrm{k}] - \omega^2[\mathrm{m}]| = 0$. The reduced terms in the matrices are

$$\frac{EA}{3L}[16] - \omega^2\frac{\rho\mathrm{AL}}{30}[16] = 0$$

so $\omega_1^2 = \frac{10E}{L^2\rho}$ and $\omega_1 = \sqrt{10}\frac{c}{L} = 3.16\frac{c}{L}$ which is less that 1% error compared to the exact result. Adding more elements increases the accuracy of each frequency estimate, and also yields estimates of the frequencies associated with the additional DOF. For example, adding a second quadratic bar element gives a total of three un-restrained DOF. So you could solve for the first three frequencies. The value for ω_1 would be more accurate and you would have the first estimates of $\omega_2$and ω_3.

Usually, the masses farthest from the supports have the most effects on the natural frequency calculations. If you only care about the frequencies you could use split lines to build larger elements near the supports. For beams and shells, the transverse displacements are more important than the tangential rotational DOF.

11.4 Frequencies of a curved solid

To illustrate a typical natural frequency problem consider a brass, *75 degree* segment of an annulus solid having a thickness of *0.3 m*, an average radius of *1.5 m*, and a width of *1 m*. The component is encastred (fixed) at one rectangular face. The thickness to width ratio is 0.3. That suggests that the study should be conducted with either a solid model or a thick shell model. Both types of elements will be used to indicate the range of uncertainty.

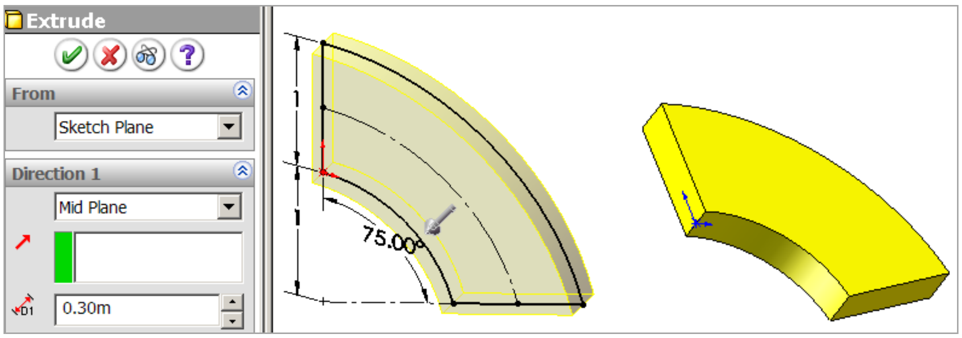
There is no simple analytic estimate to validate the study of a thick curved body. However, there is a simple cantilever beam frequency estimate that can give an estimate of the frequencies. The first frequency of such a thin beam is

$$\omega_1 = 1.732\sqrt{\frac{EI}{\rho A L^4}} = 1.732\sqrt{\frac{Eh^2}{12\rho L^4}}$$

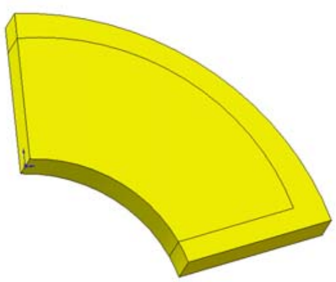
Here, the effect length, L, must be estimated. If you take the outer arc length as that length, the estimate is $\omega_1 = 48.4\ Hz$. Using the centerline arc gives *92.3 Hz*.

Generally, the displacement degrees of freedom are more important in getting natural frequencies and mode shapes than are rotational DOF. Therefore, the solid study is probably best here. In vibration problems, the material located farthest from the supports are more important. You should use mesh control to create small elements in such regions. The modeling process is:

1. Sketch and dimension the area. **Extrude** it to a thickness of *0.3 m*.



2. Click on a curved face, **Insert Sketch.**
3. Add a line and arc near the free edges farthest from the support, for later mesh control.
4. **Insert→Curve→Split Line**



11.4.1 SW Simulation frequency studies

11.4.1.1 SW Simulation Manager

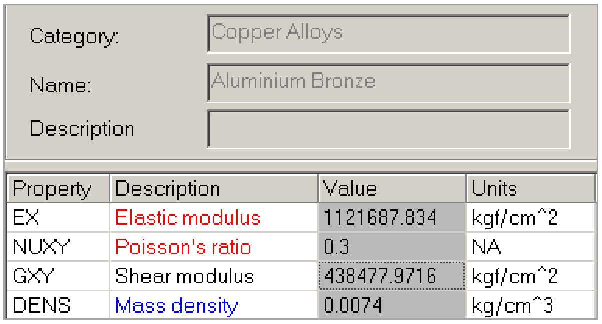
Selecting the **SW Simulation Manager** (CWManager) icon:

4. Right click on the top name to access **Study** which opens the **Study panel**.
5. Assign a **Study name**, choose **Frequency** for the **Analysis type.**
6. Define the **Mesh type** to be **solid,** click **OK**.

11.4.1.2 Define the material

At this point **Solids** will appear in the CWManager menu:

1. Right click on it to apply material data. The component is to be made of brass.
2. Pick **Apply Material to All→ Material panel→From library files button→Copper Alloys** and select **brass**, set the **Units** to MKS.



Category: Copper Alloys

Name: Aluminium Bronze

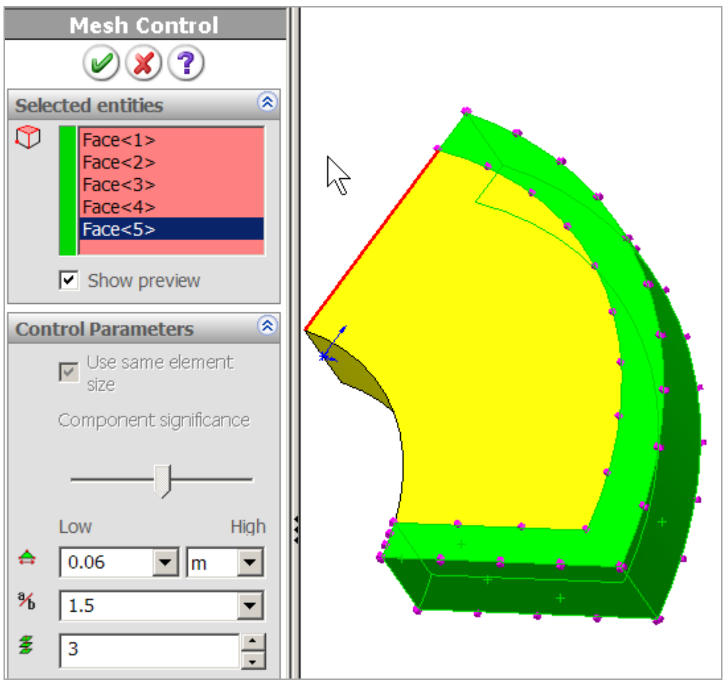
Description

Property	Description	Value	Units
EX	Elastic modulus	1121687.834	kgf/cm^2
NUXY	Poisson's ratio	0.3	NA
GXY	Shear modulus	438477.9716	kgf/cm^2
DENS	Mass density	0.0074	kg/cm^3

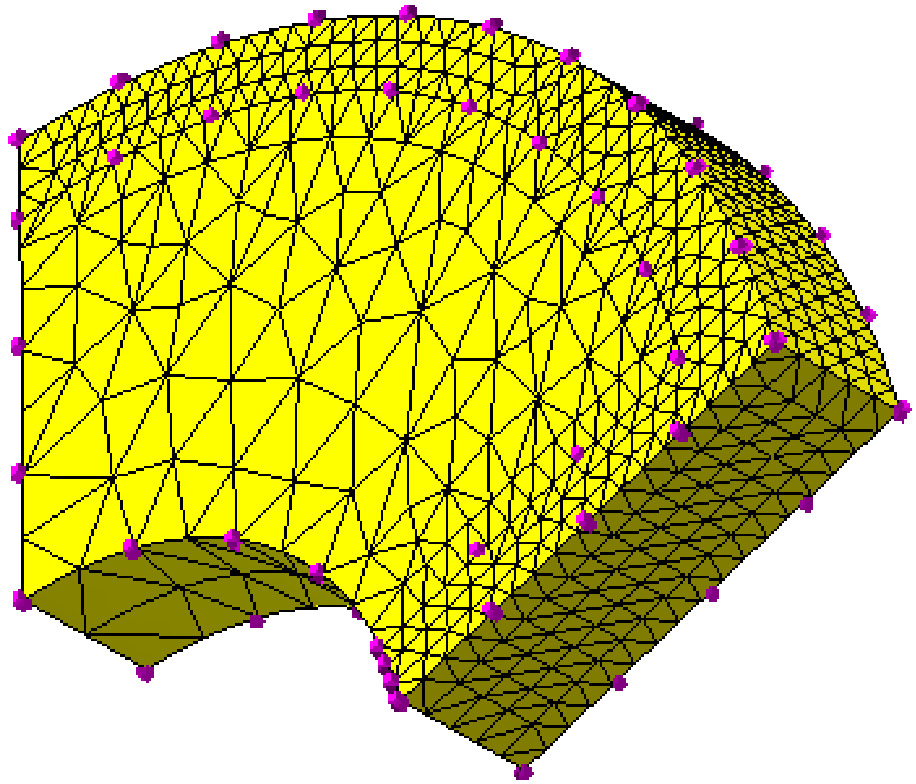
11.4.1.3 Meshing

Specify a finer mesh away from the support, and a crude mesh near the support:

1. **Mesh→Mesh Control**, select small outer faces, set size to 0.06 m.

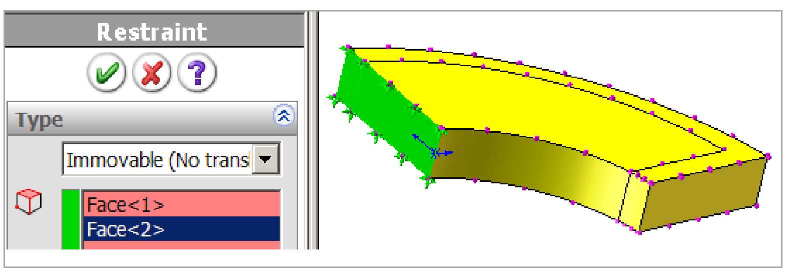


2. **Mesh→Mesh Control**, select other faces, set size to 0.3 m.
3. **Mesh→Create Mesh**



11.4.1.4 Restrain the system

4. Select **Fixtures→Immovable** and pick the support rectangles. Click **OK**. **Run**.



11.4.1.5 Post-process the frequencies and mode shapes

The **Run→Properties** were set to compute five modes and frequencies, but only the first three are summarized here. Select Results and display each mode in turn. Change views for better understanding as in Figure 11-2. Mode one is like that of a cantilever beam, with the outer edge moving perpendicular to the original plane. Mode two is a vibration in the original plane. Mode three seems to be mainly a twisting vibration. The frequencies are shown in the figure text. You can also have SW Simulation list them. The first three modes are also given in Table 11-1, along with the corresponding values from a thick shell model presented below. There is about a 10% difference in the computed frequencies.

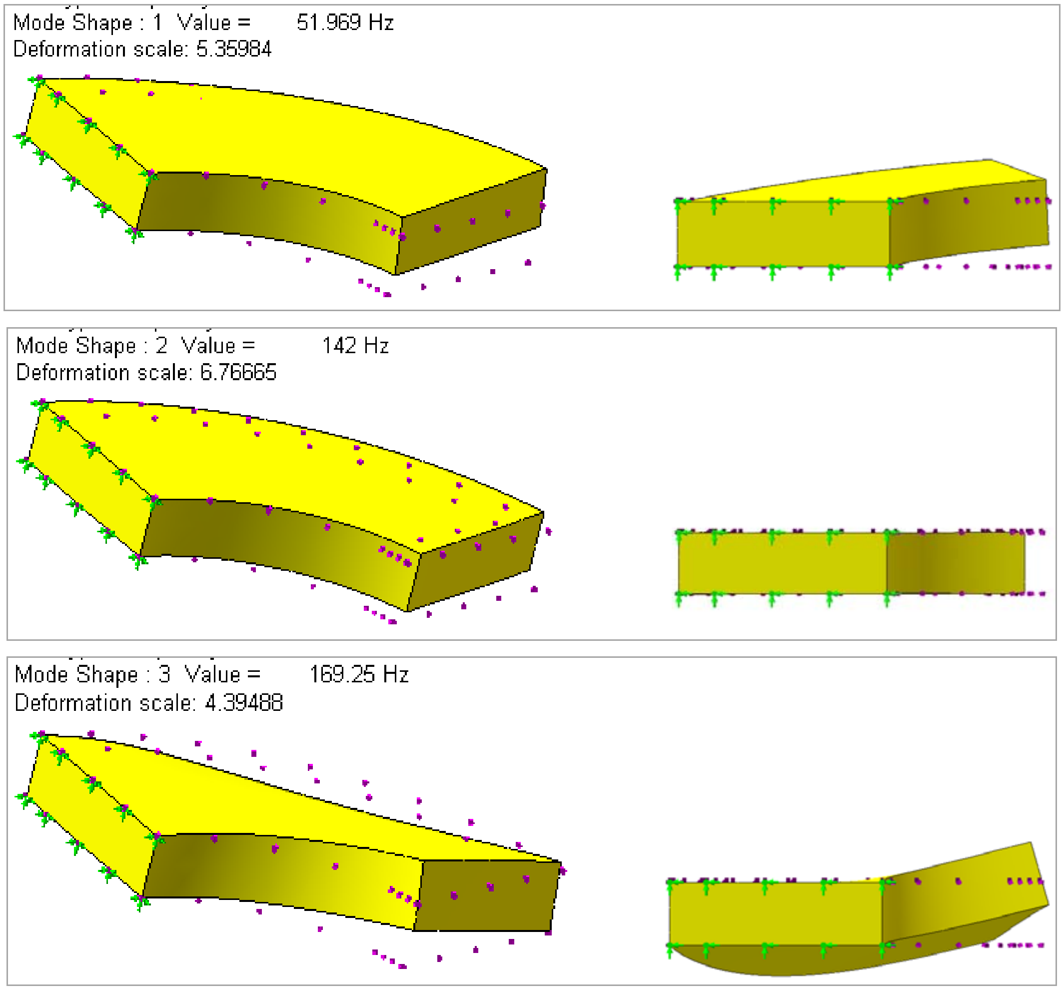


Figure 11-2 First three solid studies modes and frequencies

Table 11-1 Natural frequencies (Hz) from solids and thick shells

Model	Mode 1	Mode 2	Mode 3
Solid	52	142	169
Thick shell	46	126	155
Thin beam	48.4	-	-

11.4.2 Thick shell version

The above study was repeated with a thick shell and the same mesh controls. Some results are in Figure 11-3

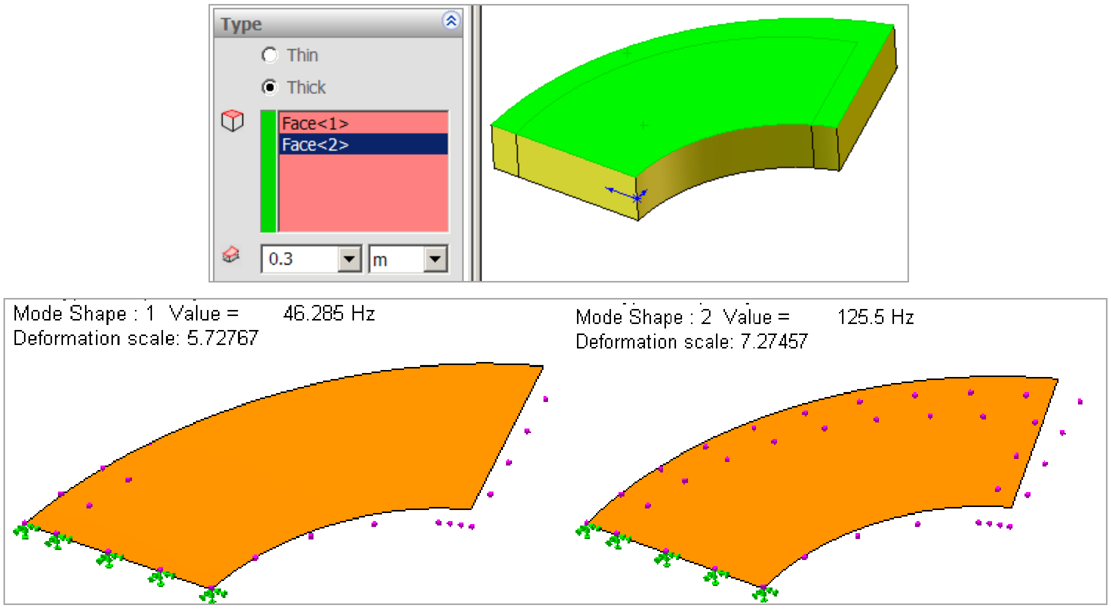


Figure 11-3 First two thick shell frequencies

11.5 Influencing the natural frequency

If you wish to influence the natural frequency you can automate the process by employing the SW Simulation optimization ability to vary the part geometric design parameters. You can also get a feel for the controlling factors by noting the fact that the natural frequencies (in Hz) of plates can generally be expressed as

$$f_j = \frac{\lambda_j}{2\pi\, L^2}\left[\frac{E\, h^3}{12\, \rho(1-\nu^2)}\right]^{1/2}$$

where f_j is the natural frequency in cycles per second, E is the elastic modulus, ν is Poisson's ratio, ρ is the mass density, *h* is the (thin) plate thickness, L is a characteristic length of the plate. The remaining factor, λ_j, is dependent on the support conditions and geometric shape of the plate. It is often a tabulated feature in standard handbooks like [3, 11]. Usually, the thickness and length are the easiest features to change. The quantity in square brackets reduces to *[E I/ρ]* for a straight beam.

