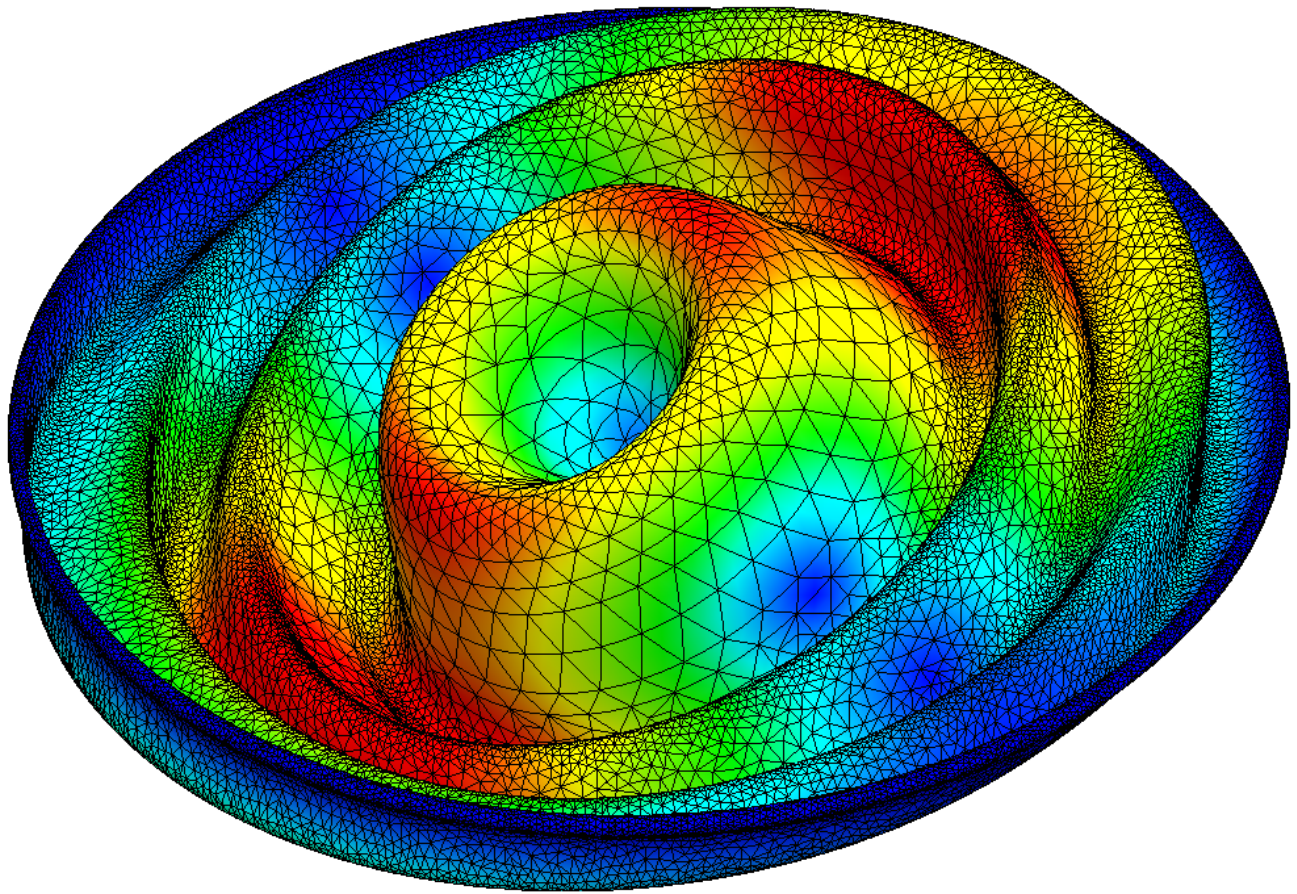# Vibration Analysis

with SOLIDWORKS® Simulation 2022

Paul M. Kurowski

Design Generator Inc.

CERTIFIED Solution Partner

3S SOLIDWORKS

Visit the following websites to learn more about this book:

SDC PUBLICATIONS

amazon.com

Google books

BARNES & NOBLE

4: Modal analysis with pre-stress

Topics covered

- Modal analysis with pre-stress
- Modal analysis and buckling analysis
- Artificial stiffness

If a structure is subjected to a load that produces significant tensile or compressive stresses, those stresses may significantly change the structure's stiffness and consequently, its modes of vibration. To account for that change, modal analysis needs to be completed on a structure with a modified stiffness caused by the existing state of stress. This type of modal analysis, called pre-load or pre-stress modal analysis, is conducted in two steps. First, a static analysis is run to find stresses that develop due to a load; these stresses are used to modify the structure's stiffness. Next, modal analysis is run on the structure with the stiffness modified by the previously found stresses.

Predominantly, tensile stresses will increase the natural frequencies as is illustrated by tuning a guitar string or stress-stiffening of a rotating component like a turbine blade. Rotating machinery typically requires that the effect of pre-stress be considered. Compressive stresses on the other hand will decrease natural frequencies. In this chapter we will illustrate the effect of tensile and compressive stresses on natural frequencies. Then we will demonstrate how a modal analysis with pre-stress relates to a buckling analysis.

Open part model ROTOR which is a simplified representation of a helicopter rotor. We are interested in the effect of centrifugal forces that develop due to rotor rotation on the fundamental natural frequency of the rotor blade. The fundamental frequency is the lowest natural frequency. All four blades are identical and isolated from each other by a support at the hub. Therefore, we may simplify the analysis to only one blade (Figure 4-1).
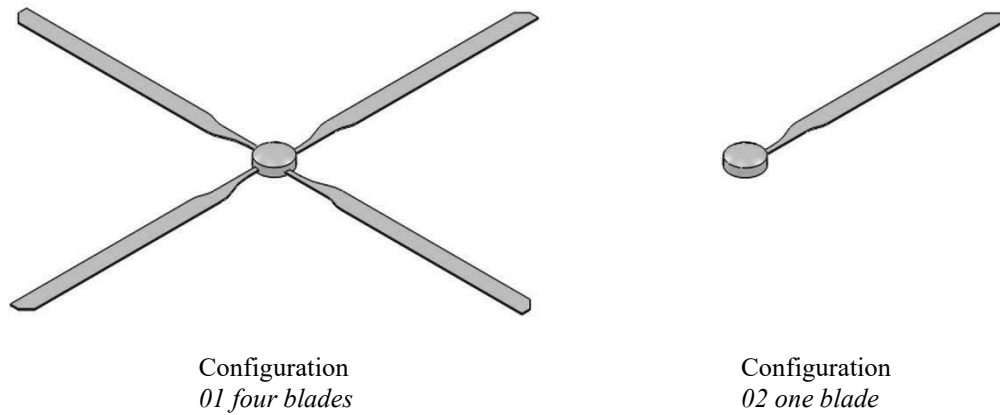
Configuration
01 four blades

Configuration
02 one blade

Figure 4-1: ROTOR model.

Modal analysis will be conducted on one blade.

Switch to *02 one blade* configuration and create a **Frequency** study *01 rotating*. Apply **Fixed** restraints as shown in Figure 4-2.
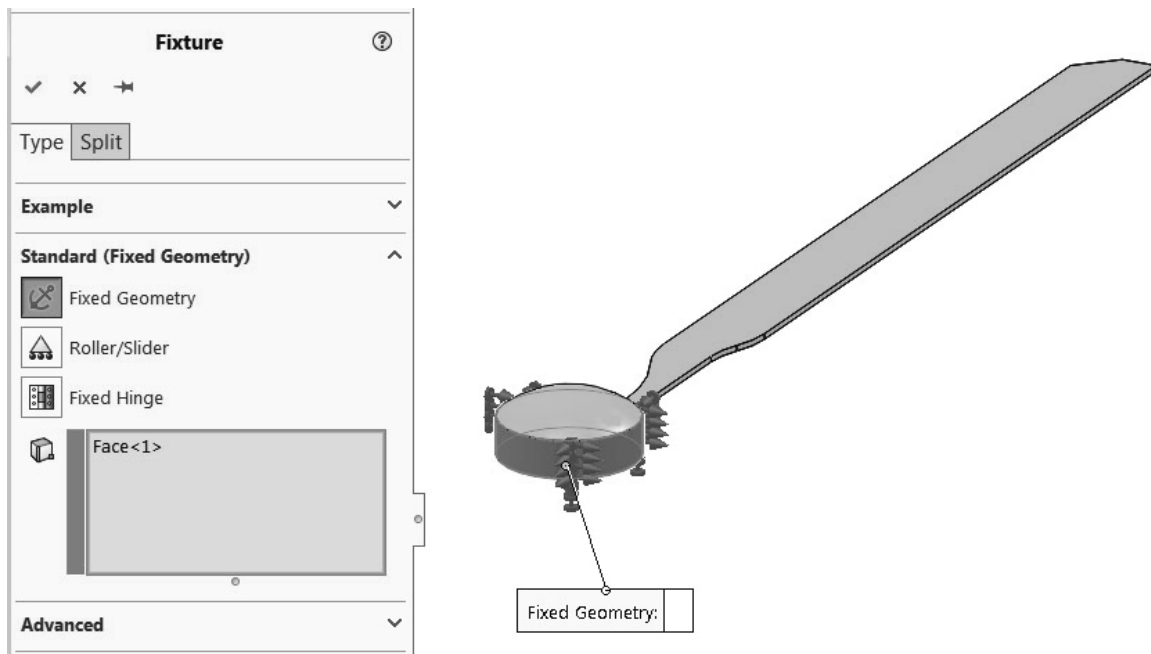

Figure 4-2: Definition of Restraints.

A fixed restraint is defined to the cylindrical surface. You may also apply it to the top and bottom face of the hub; it won't have any significant effect on the vibration of the blade.

Define a centrifugal load of 300RPM by selecting the cylindrical surface where the **Fixed** restraint has been defined. This cylindrical face uniquely defines the axis which is taken as the axis of rotation as shown in Figure 4-3.
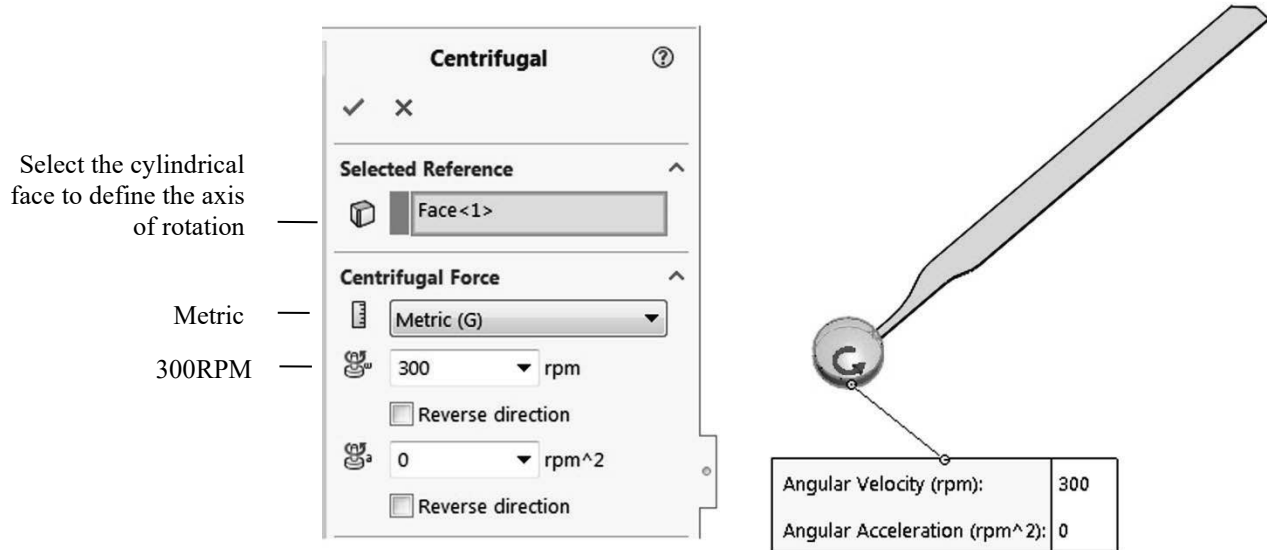

Figure 4-3: Centrifugal load definition.

A centrifugal load is defined by an angular rotation about the axis. Use metric units to define it in revolutions per minute (RPM). Do not define any angular acceleration.

In the study properties, specify one mode and use the **Direct sparse** solver (Figure 4-4).
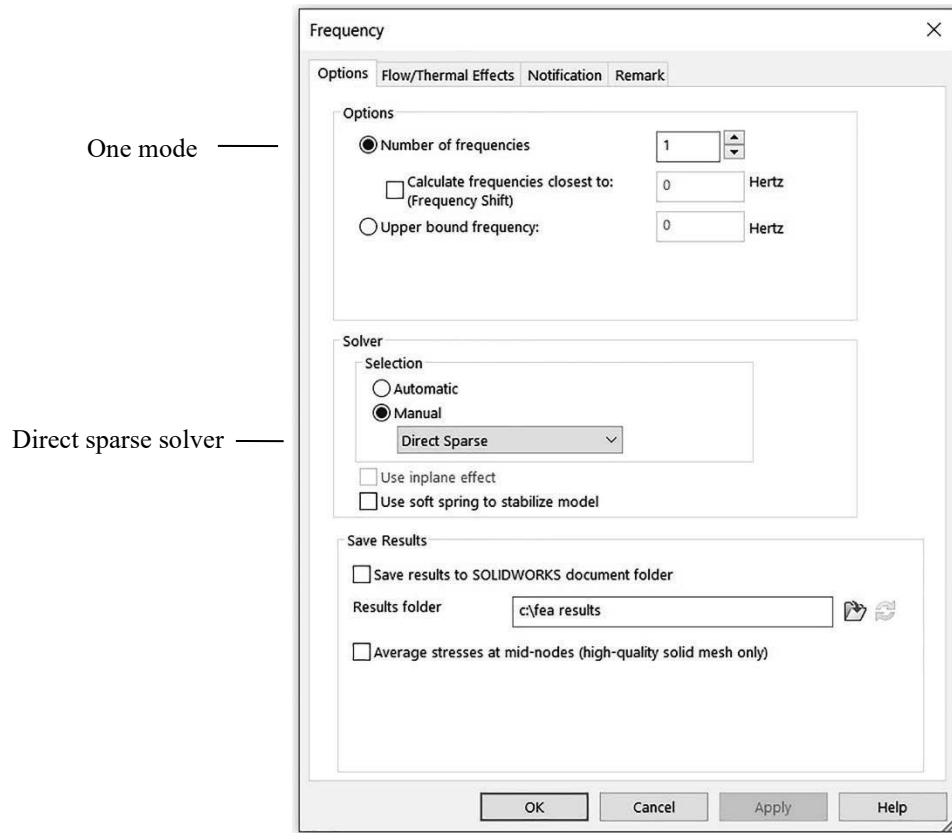

Figure 4-4: Properties of the frequency study.

Direct Sparse solver must be used for modal analysis with pre-stress. FFEPlus solver is not available for modal analysis with pre-stress.

You may also use Automatic solver selection.

Mesh the model with the default mesh size and solve the study. Copy study *01 rotating* into *02 stopped*, suppress the centrifugal load and solve the study without pre-stress.

Fundamental frequency
of rotating blade: 5.35Hz

Fundamental frequency
of stopped blade: 0.54Hz
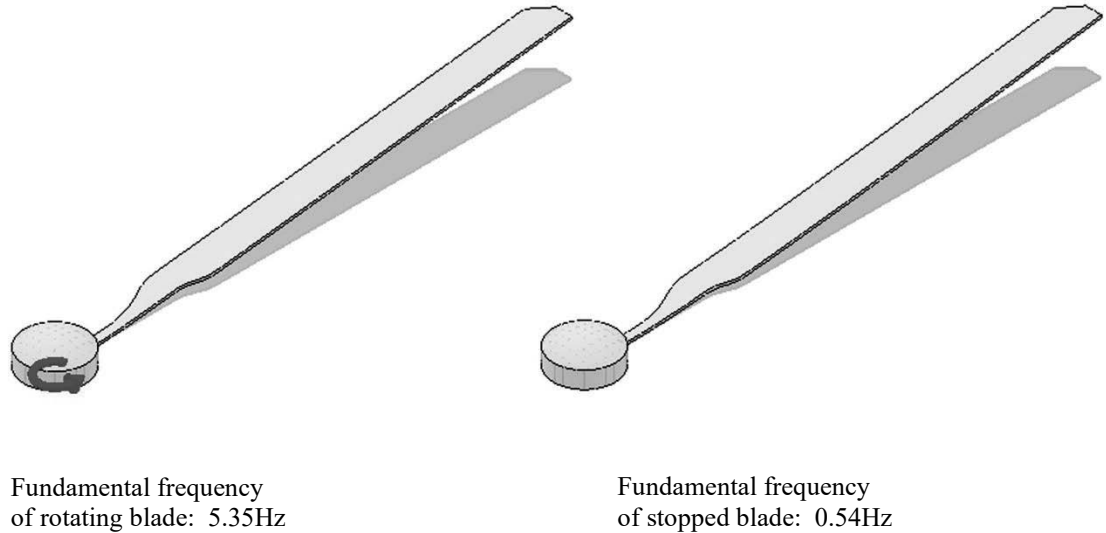
Figure 4-5: The first mode of vibration of rotating and stopped blade.

The mode shape is the same for the rotating and stopped blade, but the modal frequency is very different. Both plots show the undeformed shape superimposed on the modal plot.

A comparison of results between rotating and stopped blade shows a very strong effect of rotation; the first natural frequency is ten times as high for the rotating blade as it is for the stopped blade. The centrifugal force produces tensile stresses which cause stress stiffening of the blade. The resultant stiffness of the rotating blade is the sum of the elastic stiffness and stress stiffness. Stress stiffness due to tensile stress is positive, and the resultant stiffness of the rotating blade is higher than the stiffness of the stationary blade. Higher stiffness, in turn, produces a higher natural frequency.

When a load is present in a **Frequency** study, this load is used to modify the stiffness. The analysis is then called modal analysis with pre-load. The term modal analysis with pre-stress is also used. The solution progresses in two steps. First, a static analysis is run to find stresses. These stresses are used to modify the model stiffness. Then modal analysis is run using the combined elastic and stress stiffness.

Remember that it is a serious error to confuse pre-load with excitation load!

The effect of tensile stress on the natural frequency may be easily demonstrated by tuning a guitar string, where increasing string tension increases the natural frequency.

The effect of compressive stresses on the natural frequency is the opposite; compressive stresses produce negative stress stiffness which decreases the resultant stiffness and the natural frequency decreases.

We will demonstrate this with the COLUMN model subjected to compression by two wedges as shown in Figure 4-6. This illustration schematically shows an apparatus used to demonstrate buckling of a rectangular column.
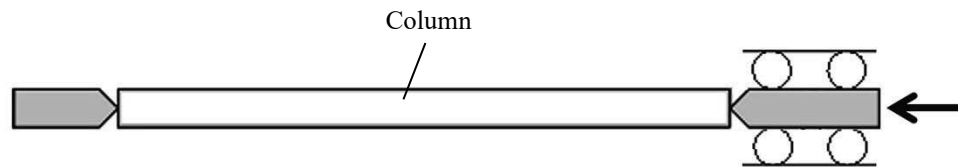
Column

Figure 4-6: A prismatic column compressed by two wedges.

The left wedge is fixed. The right wedge is held in-between rollers. It can only translate in the direction of the load. Compressive load applied to the moving wedge.

The column is shown shorter than in COLUMN assembly model.

The column is "squeezed" between two wedges: one wedge is fixed and provides support, and the other one is guided by rollers and can move only in the direction of the load. We want to simulate this test rig to find a relationship between the applied load and the first natural frequency of the loaded column in the range of the loads changing from tensile load to compressive, to the point of buckling. Using the test rig as shown in Figure 4-6 we would not be able to apply a tensile load, but using SOLIDWORKS Simulation, this is very easy. The analysis does not require analysis of an assembly; it can be completed on one part.

Review the COLUMN assembly model, and then open the COLUMN part model which will be used for analysis.

To find the magnitude of the buckling load we need to run a buckling analysis. Using the COLUMN part model, create a **Buckling** study titled *00 buckling* and define restraints as shown in Figure 4-7.

A fixed restraint to split line on the end face (line not visible).

Restraint to the split line at the end face in the direction normal to the Top reference plane.
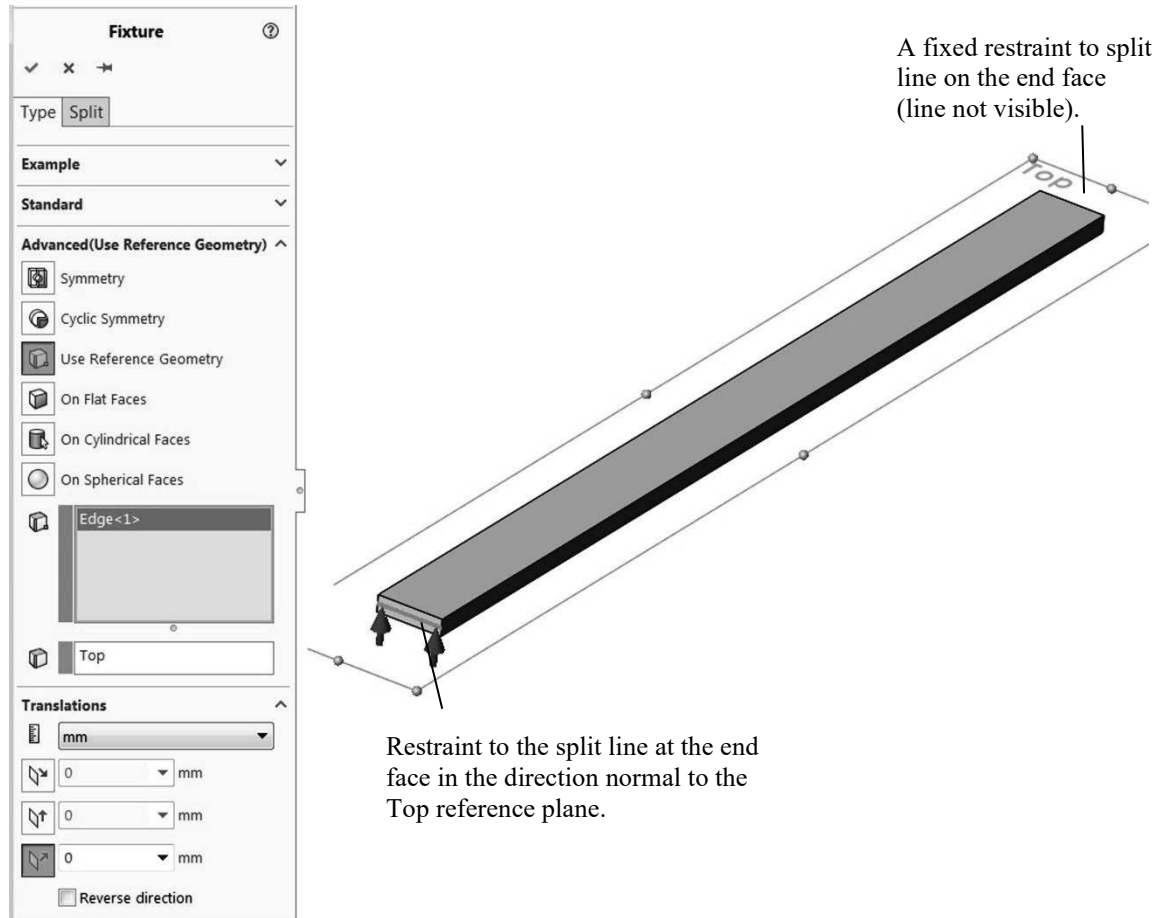
Figure 4-7: Restraints applied to the COLUMN model.

The fixture window shows restraint on the side of the moving wedge. Restraint on the side of the fixed wedge is a fixed restraint. The fixed restraint is not shown.

Notice that the **Fixed** restraint, as defined on the split line on the side of the fixed wedge, will be transferred to nodes of solid elements which have three degrees of freedom. Therefore, this **Fixed** restraint will effectively produce a hinge support.
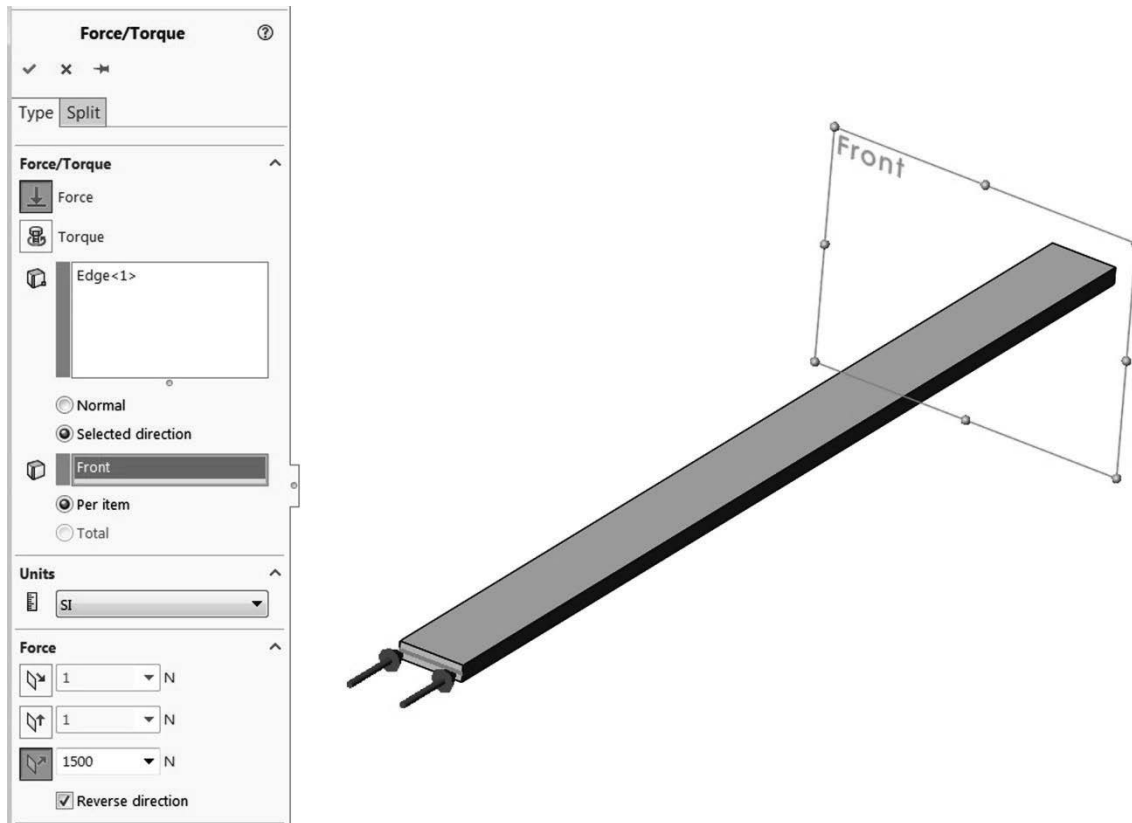
Next, define a load as shown in Figure 4-8.

Figure 4-8: Load applied to the COLUMN model.

The load is applied to the same split line where the restraint on the side of the moving wedge has been applied.

Notice that on the loaded end, the restraint and load are both applied to the same entity (the split line), but in different directions. If the load was applied in the direction of this restraint, it would be ineffective.

Mesh the model with the default element size and run the solution of the buckling study. The first buckling mode is shown in Figure 4-9 and the **Buckling Load Factor** (BLF) is 1.5755, meaning that according to the linear buckling model, buckling will happen at a load of 1575.5N.
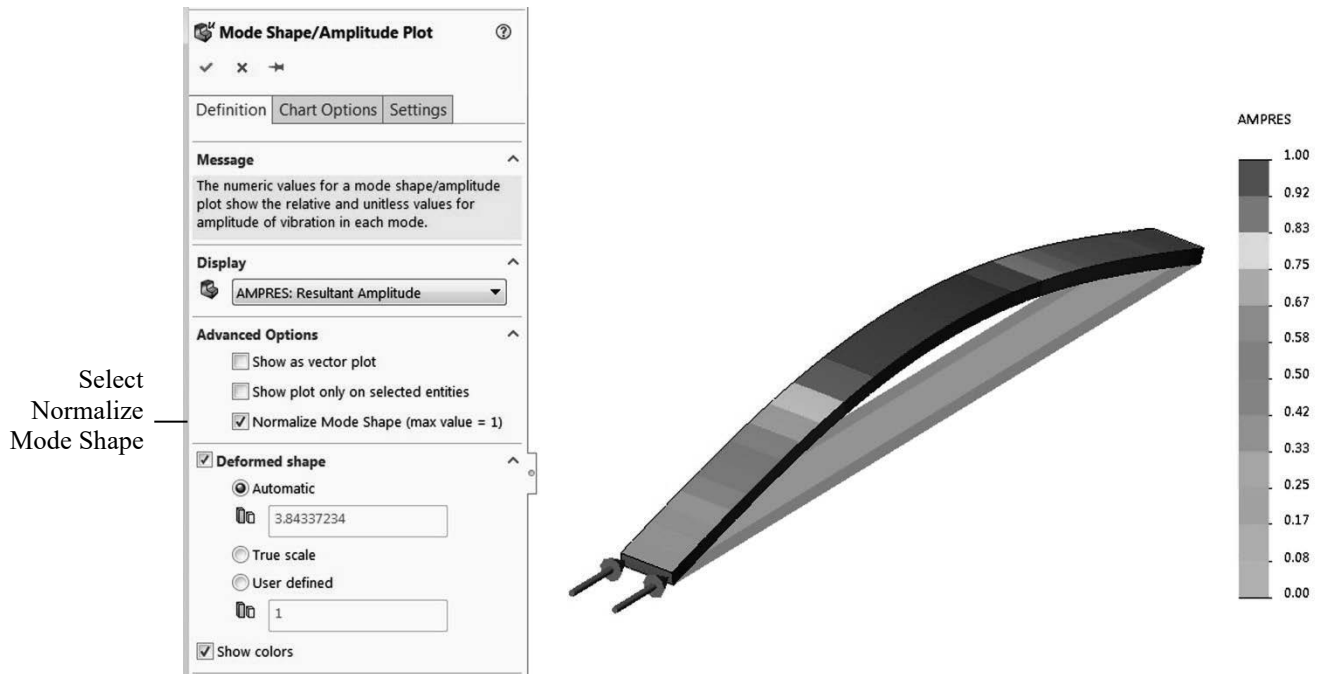
Figure 4-9: Buckled shape shown using displacement plot; undeformed model is superimposed on the deformed plot.

Even though numerical values are shown, their values may be used only to find a displacement ratio rather than absolute displacement. This is in close analogy to modal analysis. In this plot, the results have been normalized to 1.

The linear buckling analysis was necessary to establish the range of loads in the modal analysis with pre-stress. We will now conduct a numerical experiment subjecting the model to different loads ranging from tensile to compressive to study the effect of pre-load on the fundamental natural frequency. The experiment will be conducted in twelve **Simulation** studies while the load is changed from 1500N tensile load to a compressive load causing buckling. A tensile load is denoted as positive, compressive as negative.

We should point out that numerical results presented in this experiment may differ slightly depending on the service pack used. In all cases the **Direct sparse solver** is used.

Create a **Frequency** study titled *01*; you may copy loads, restraints and the mesh from *00 buckling*. Remember to apply tensile load and make it 1500N. Obtain the solution and record the first natural frequency. Next, proceed in load steps shown in Figure 4-10.

Study number	Preload [N]	Frequency [Hz]
01	+1500	177.88
02	+1000	162.78
03	+500	146.12
04	0	127.31
05	-500	105.19
06	-1000	76.95
07	-1200	62.15
08	-1400	42.49
09	-1500	27.89
10	-1550	16.20
11	-1570	7.52
12	-1575.49	0.29

Figure 4-10: A summary of the 12 studies illustrates the change of the first natural frequency with the applied pre-load. All runs are solved with the default element size of 3.11mm.

+ sign indicates tensile load; - sign indicates compressive load.

Review the results in Figure 4-10 and notice that the compressive pre-load in study 12 is just 0.01N less than the bucking load.

Results summarized in Figure 4-10 are presented as a graph in Figure 4-11. Steep curve near the buckling load is the reason why the last four steps were conducted with small load increments.
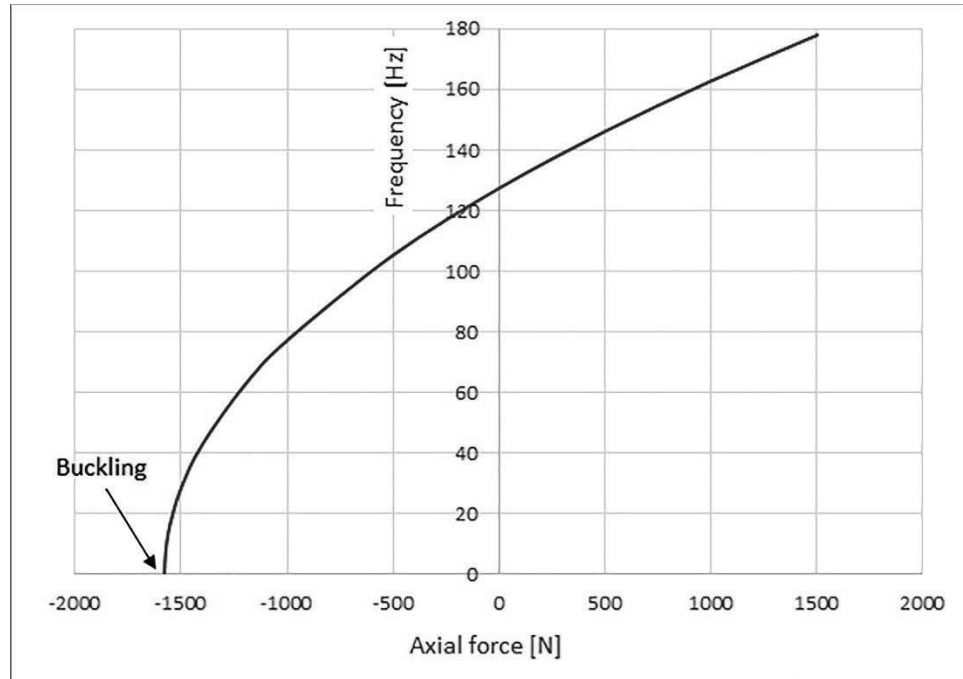

Figure 4-11: Frequency of the first mode of vibration as a function of pre-load.

The frequency reaches zero when the compressive load equals the buckling load.

As Figure 4-11 shows, a pre-load causing the drop of the first natural frequency to zero is equal to the buckling load.

Tensile stresses develop positive stress stiffness that adds up to the elastic stiffness; therefore, the resultant stiffness increases, which is reflected by a higher natural frequency as compared to the unloaded column. Compressive stresses develop negative stress stiffness that is subtracted from the elastic stiffness, and this decreases the resultant stiffness, consequently the natural frequency decreases. When the pre-load approaches the magnitude of buckling load, the resultant stiffness is very low producing a very low natural frequency. When the pre-load reaches the buckling load, the stiffness drops down to zero and that leads to buckling.

When the magnitude of a compressive load approaches the buckling load, the effective stiffness is very close to zero; this is why it has a near-zero natural frequency. Therefore, we may use the model from study *12*, the one pre-

loaded with a 1575.49N compressive load, as a tool to investigate the effect of discretization on the model stiffness. As you know, meshing adds some stiffness to a model; let's call it artificial stiffness. The larger the elements are, the more artificial stiffness is added and vice versa. The artificial stiffness does not change with pre-load; therefore, in study *12*, the artificial stiffness is a major contributor to the model stiffness. The natural frequency of 0.31Hz is not a real frequency under that load. Elastic stiffness and stress stiffness have almost canceled themselves out and this 0.31Hz frequency is an artifact produced by the artificial stiffness that is there due to discretization error. The numerical value of that frequency completely depends on the choice of mesh size.

We will now conduct a numerical experiment to demonstrate that the remaining model stiffness in study *12* is a product of discretization, and not a real stiffness.

Keeping in mind that all twelve studies were run with default element size 3.1mm, we will now use 2mm element size. Copy study *12* into *12 fine* and re-mesh it with a 2mm element size. With a smaller element size, the artificial stiffness is reduced, and the effective stiffness becomes negative. An attempt to run the solution produces an error message shown in Figure 4-12.
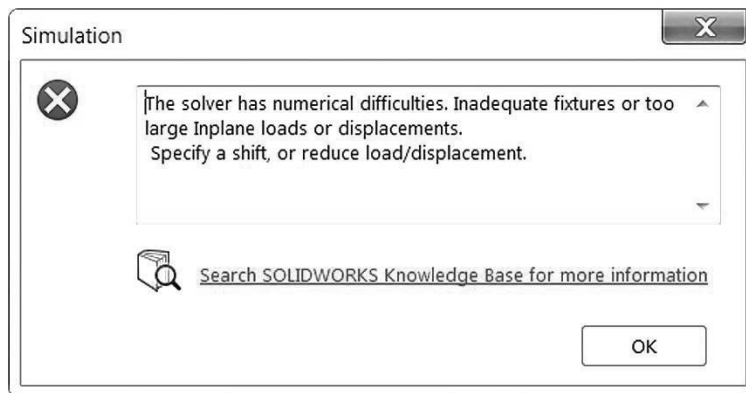

Figure 4-12: Error message caused by a negative effective stiffness.

The explanation offered by the error message is generic; it does not apply to our problem.

We now realize that all that was "holding" the model stable in study *12* was an artificial stiffness produced by the process of discretization. The contribution of artificial stiffness to total stiffness is significant only for loads very close to buckling because the real stiffness is then very low.

As an additional exercise, you may want to analyze the effect of element size on the natural frequency in the model subjected to a compressive load that is very close to the buckling load. You'll find that the natural frequency strongly depends on the element size. Numerical results are summarized in Figure 4-13 and are graphed as shown in Figure 4-14.

Element size mm	Natural frequency Hz
4	1.024
5	1.229
6	1.583
7	2.103
8	2.647

Figure 4-13: Natural frequency at the compressive load of 1575.49N as a function of the element size.

A strong dependence of frequency on the element size makes these results useless.

Figure 4-14: Natural frequency at the compressive load of 1575.49N as a function of the element size.

Increasing element size adds stiffness to the model and that increases the natural frequency. These results are useless.

The dependence of frequency on element size is clearly visible in this example because stiffness added by meshing is higher than the stiffness of the model that is very close to buckling.

The fundamental rule of using FEA results states that before we use results to make a design decision, we must prove that these results are not significantly dependent on the choice of discretization (element size). The graph in Figure 4-14 proves the opposite: frequency results for load magnitudes close to buckling are strongly dependent on the mesh size rendering results presented in Figure 4-14 useless.

Summary of studies completed

Model	Configuration	Study Name	Study Type
ROTOR.sldprt	*02 one blade*	*01 rotating*	Frequency
		02 stopped	Frequency
COLUMN.sldprt	*Default*	*00 buckling*	Buckling
		01	Frequency
		02	Frequency
		03	Frequency
		04	Frequency
		05	Frequency
		06	Frequency
		07	Frequency
		08	Frequency
		09	Frequency
		10	Frequency
		11	Frequency
		12	Frequency
		12 fine	Frequency

Figure 4-15: Models in Chapter 4