

Modal Analysis

Introduction

A modal analysis determines the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic analysis, or a spectrum analysis. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. You can also perform a modal analysis on a pre-stressed structure, such as a spinning turbine blade.

If there is damping in the structure or machine component, the system becomes a damped modal analysis. For a damped modal system, the natural frequencies and mode shapes become complex.

For a rotating structure or machine component, the gyroscopic effects resulting from rotational velocities are introduced into the modal system. These effects change the system's damping. The damping can also be changed when a Bearing is present, which is a common support used for rotating structure or machine component. The evolution of the natural frequencies with the rotational velocity can be studied with the aid of Campbell Diagram Chart Results.

A Modal analysis can be performed using the ANSYS, Samcef, or ABAQUS solver. Any differences are noted in the sections below. Rotordynamic analysis is not available with the Samcef or ABAQUS solver.

Points to Remember

- The Rotational Velocity load is not available in Modal analysis when the analysis is linked to a Static Structural analysis.
- Pre-stressed Modal analysis requires performing a Static Structural analysis first. In the modal analysis you can use the Initial Condition object to point to the **Static Structural** analysis to include pre-stress effects.

Preparing the Analysis

Create Analysis System

General

Basic general information about this topic

Specifics

... for this analysis type:

From the **Toolbox**, drag a **Modal**, **Modal (Samcef)**, or **Modal (ABAQUS)** template to the **Project Schematic**.

Define Engineering Data

General

Basic general information about this topic

Specifics

... for this analysis type:

Due to the nature of modal analyses any nonlinearities in material behavior are ignored. Optionally, orthotropic and temperature-dependent material properties may be used. The critical requirement is to define stiffness as well as mass in some form. Stiffness may be specified using isotropic and orthotropic elastic material models (for example, Young's modulus and Poisson's ratio), using hyperelastic material models (they are linearized to an equivalent combination of initial bulk and shear moduli), or using spring constants, for example. Mass may be derived from material density or from remote masses.

Note: Hyperelastic materials are supported for pre-stress modal analyses. They are not supported for standalone modal analyses.

Attach Geometry

General *Basic general information about this topic*

Specifics *... for this analysis type:*

When 2D geometry is used, Generalized Plane Strain is not supported for the Samcef or ABAQUS solver.

When performing a Rotordynamic Analysis, the rotors can be easily generated using the Import Shaft Geometry feature of ANSYS DesignModeler. The feature uses a text file to generate a collection of line bodies with circular or circular tube cross sections.

Define Part Behavior

General *Basic general information about this topic*

Specifics *... for this analysis type:*

You can define a **Point Mass** for this analysis type.

Define Connections

General *Basic general information about this topic*

Specifics *... for this analysis type:*

- Joints are allowed in a modal analysis. They restrain degrees of freedom as defined by the joint definition.
- The stiffness of any spring is taken into account and if specified, damping is also considered.
- For the Samcef and ABAQUS solvers, only contacts, springs, and beams are supported. Joints are not supported.

Apply Mesh Controls/Preview Mesh

General

Basic general information about this topic

Specifics

... for this analysis type:

There are no special considerations for this analysis type.

Establish Analysis Settings

General

Basic general information about this topic

Specifics

... for this analysis type:

For a Modal analysis, the basic Analysis Settings include:

Options

Using the **Max Modes to Find** property, specify the number of frequencies of interest. The default is to extract the first **6** natural frequencies. The number of frequencies can be specified in two ways:

1. The first N frequencies ($N > 0$), or...
2. The first N frequencies in a selected range of frequencies.

Solver Controls

Two properties are available for this category:

- **Damped**: use this property to specify if the modal system is undamped or damped. Depending upon your selection, different solver options are provided. **Damped** by default, it is set **No** and assumes the modal system is an undamped system.
- **Solver Type**: it is generally recommended that you allow the program to select the type of solver appropriate for your model in both undamped and damped modal systems. When the Solver Type is set to **Reduced Damped**, the following additional properties become available:
 - **Store Complex Solution**: This property is only available when the **Solver Type** property is set to **Reduced Damped**. This property enables you to solve and store a damped modal system as an undamped modal system. By default, it is set to **Yes**.
 - **Mode Reuse**: This property allows the solver to compute complex eigensolutions efficiently during subsequent solve points by reusing the undamped eigensolution that is calculated at the first solve point. The default setting is **Program Controlled**. Set this property to **Yes** to enable it or **No** to disable it.

Note:

- If a solver type of **Unsymmetric**, **Full Damped** or **Reduced Damped** is selected, the modal system cannot be followed by a Transient Structural, Harmonic Response, Random Vibration, or Response Spectrum system. However, for a MSUP Harmonic Analysis and a MSUP Transient Analysis, you can use the **Reduced Damped** solver with the **Store Complex Solution** property set to **No**. In this case, regular (non-complex) mode shapes are calculated and are used for Mode Superposition. Although complex frequencies are used for Mode Superposition, regular (non-complex) frequencies are reported in tabular data. In the presence of damping, the **Reduced Damped** solver with **Store Complex Solution** set to **No** is not equivalent to the **Undamped** solver.
- If an undamped Modal analysis has a pre-stressed environment from a Static Structural Analysis with the **Newton-Raphson Option** set to **Unsymmetric**, the **Program Controlled** option selects **Unsymmetric** as the **Solver Type** setting (the Mechanical APDL command **MODOPT,UNSYM** is issued).

Cyclic Controls

When running a cyclic symmetry analysis, set the **Harmonic Index Range** to **Program Controlled** to solve for all harmonic indices, or to **Manual** to solve for a specific range of harmonic indices.

Output Controls

By default, only mode shapes are calculated. You can request **Stress** and **Strain** results to be calculated but note that "stress" results only show the relative distribution of stress in the structure and are not real stress values. You can also choose whether or not to have these results stored for faster result calculations in linked systems.

Damping Controls

The options of the **Stiffness Coefficient Defined By** property, **Direct Input** or **Damping vs. Frequency**, enable you to define the method used to define the **Stiffness Coefficient**. If you select **Damping vs. Frequency**, the **Frequency** and **Damping Ratio** properties appear requiring you to enter values to calculate the **Stiffness Coefficient**. Otherwise, you specify the **Stiffness Coefficient** manually. The **Mass Coefficient** property requires a manual entry.

Rotordynamics Controls

Specify these properties as needed when setting up a Rotordynamic Analysis.

Analysis Data Management

This category is only applicable to **Modal** systems. These properties enable you to save specific solution files from the Modal analysis for use in other analyses. You can set the **Future Analysis** field to **MSUP Analyses** if you intend to use the modal results in a subsequent Transient Structural, Harmonic Response, Random Vibration (PSD), or Response Spectrum (RS) analysis. If you link a Modal system to another analysis type in advance, the **Future Analysis** property defaults to the setting, **MSUP Analyses**. When a PSD analysis is linked to a modal analysis, additional solver files must be saved to achieve the PSD solution. If the files were not saved, then the modal analysis has to be solved again and the files saved.

Note:

- **Solver Type**, **Damping Controls**, and **Rotordynamic Controls** are not available to the Samcef or ABAQUS solver.
- **Solver Type**, **Scratch Solver Files**, **Save ANSYS db**, **Solver Units**, and **Solver Unit System** are only applicable to **Modal** systems.

Define Initial Conditions

General

Basic general information about this topic

Specifics

... for this analysis type:

You can point to a **Static Structural** analysis in the **Initial Condition** environment field if you want to include pre-stress effects. A typical example is the large tensile stress induced in a turbine blade under centrifugal load that can be captured by a static structural analysis. This causes significant stiffening of the blade. Including this pre-stress effect will result in much higher, realistic natural frequencies in a modal analysis.

If the Modal analysis is linked to a Static Structural analysis for initial conditions and the parent static structural analysis has multiple result sets (multiple restart points at load steps/sub steps), you can start the Modal analysis from any restart point available in the Static Structural analysis. By default, the values from the last solve point are used as the basis for the modal analysis. See [Restarts from Multiple Result Sets](#) in the [Applying Pre-Stress Effects for Implicit Analysis](#) Help section for more information.

Note:

- When you perform a pre-stressed Modal analysis, the support conditions from the static analysis are used in the Modal analysis. You cannot apply any new supports in the Modal analysis portion of a pre-stressed modal analysis. When you link your Modal analysis to a Structural analysis, all structural loading conditions, including Inertial loads, such as Acceleration and Rotational Velocity, are deleted from the Modal portion of the simulation once the loads are applied as initial conditions (via the **Pre-Stress** object). Refer to the Mechanical APDL command **PERTURB,HARM,,,DZEROKEEP** for more details.
- To account for the Coriolis Effect of rotational velocity applied in the Static analysis, you need to re-apply the rotational velocity in the Modal analysis.
- For **Pressure** boundary conditions in the Static Structural analysis: if you define the load with the **Normal To** option for faces (3D) or edges (2-D), you could experience an additional stiffness contribution called the "pressure load stiffness" effect. The **Normal To** option causes the pressure acts as a follower load, which means that it continues to act in a direction normal to the scoped entity even as the structure deforms. **Pressure** loads defined with the **Components** or **Vector** options act in a constant direction even as the structure deforms. For a same magnitude, the "normal to" pressure and the component/vector pressure can result in significantly different modal results in the follow-on Modal Analysis. See the [Pressure Load Stiffness](#) topic in the [Applying Pre-Stress Effects for Implicit Analysis](#) Help Section for more information about using a pre-stressed environment.
- If displacement loading is defined with **Displacement**, **Remote Displacement**, **Nodal Displacement** or **Bolt Pretension** (specified as **Lock**, **Adjustment**, or **Increment**) loads in the Static Structural analysis, these loads become fixed boundary conditions for the Modal solution. If the Modal solution is followed by a Harmonic solution, these displacement loads become fixed boundary conditions for the Harmonic solution as well. This prevents the displacement loads from becoming a sinusoidal load during the Harmonic solution.

Apply Loads and Supports

General

[Basic general information about this topic](#)

Specifics

... for this analysis type:

Only the Rotational Velocity and Thermal Condition boundary conditions are supported for a stand-alone modal analysis. All structural supports can be applied except a non-zero **Displacement**, a **Remote Displacement**, and the **Velocity** support. Due to its nonlinear nature, a **Compression Only Support** is not recommended for a modal analysis. Use of compression only supports may result in extraneous or missed natural frequencies.

For the Samcef and ABAQUS solvers, the following supports are not available: Compression Only Support, Elastic Support. When using line bodies, the following Pipe Pressure and Pipe Temperature loads are not available to the Samcef solver. Additionally, the Pipe Idealization object is also unavailable for the Samcef or ABAQUS solver.

Note: In a pre-stressed modal analysis:

- Any structural supports used in the static analysis persist. Therefore, you are not allowed to add new supports in the pre-stressed modal analysis.
- When creating a Campbell diagram, the Rotational Velocity in the Static Structural Analysis is used to create normal stress stiffening effects in the Modal Analysis. It is not used to create centrifugal force effects for generating the Campbell diagram.

Solve

General

Basic general information about this topic

Specifics

... for this analysis type:

Solution Information continuously updates any listing output from the solver and provides valuable information on the behavior of the structure during the analysis.

Important: If you specify the **Distribute Solution** setting (the default setting on the **Advanced Properties** dialog of the Solve Process Settings), the files file.full, file.esav and file.emat may not be combined at the end of the Modal analysis solution. As a result, any downstream system, including a Response Spectrum, Mode Superposition Harmonic Response, Mode Superposition Transient, or Random Vibration analysis, or a follow on Mechanical APDL (turn on the **Distributed** property in Project Schematic), must also use a **Distributed Solution** setting as opposed to a shared memory solution, when the setting is turned off.

Review Results

General

Basic general information about this topic

Specifics

... for this analysis type:

Highlight the **Solution** object in the tree to view a bar chart of the frequencies obtained in the modal analysis. A tabular data grid is also displayed that shows the list of frequencies, stabilities, modal damping ratios and logarithm decrements of each mode.

Note: In a Modal Analysis (and other eigenvalue-based analyses such as buckling), the solution consists of a deformed shape scaled by an arbitrary factor. The actual magnitudes of the deformations and any derived quantities, such as strains and stresses, are therefore meaningless. Only the relative values of such quantities throughout the model should be considered meaningful. The arbitrary scaling factor is numerically sensitive to slight perturbations in the analysis; choosing a different unit system, for example, can cause a significantly different scaling factor to be calculated.

For an undamped modal analysis, only frequencies are available in the **Tabular Data** window. For a damped modal analysis, real and imaginary parts of the eigenvalues of each mode are listed as **Stability** and **Damped Frequency**, respectively, in the **Tabular Data** window. If the real/stability value is negative, the eigenmode is considered to be stable. For the damped modal analysis, **Modal Damping Ratio** and **Logarithmic Decrement** are also included in the **Tabular Data** window. Like the stability value, these values are an indicator of eigenmode stability commonly used in rotordynamics.

If you select the **Reduced Damped** solver and set the **Store Complex Solution** property to **No**, then the application solves and stores the damped modal system as an undamped modal system. In addition to the undamped **Frequency**, the **Damped Frequency**, **Stability**, **Modal Damping Ratio** and **Logarithmic Decrement** result values are available in the **Tabular Data** window.

Note: For the **Reduced Damped** solver with the **Store Complex Solution** property set to **No**, the Mechanical APDL Solver only writes undamped frequencies into result file. The solver retrieves the **Damped Frequency**, **Stability**, **Modal Damping Ratio** and **Logarithmic Decrement** from the ANSYS database on the fly during the solution process. Use extra caution when using the **/POST1** in a **Command** object and make sure that your command entries and syntax are correct (especially if using the ***GET** command). Incorrect command entries can cause zero values for the **Damped Frequency** and **Stability**. Check the **Solution Information** and error/warning messages to troubleshooting issues.

If **Campbell Diagram** is set to On, a Campbell diagram chart result is available for insert under Solution. A Campbell diagram chart result conveys information as to how damped frequencies and stabilities of a rotating structural component evolve/change in response to increased rotational velocities. More detailed information about the result can be found in [Campbell Diagram Chart Results](#). The **Campbell Diagram** function is not available to the Samcef or ABAQUS solver.

Note: The Campbell diagram result chart is only appropriate for a rotating structural component that is axis-symmetrical. It is supported for all body types: solid, shell, and line bodies, but limited to single spool systems. For a single spool system, all bodies in the modal system are subjected to one and only single rotational velocity.

The contour and probe results are post-processed using set number, instead of mode number. The total set number is equal to number of modes requested multiplied by number of rotational velocity solve points. You can use the **Set**, **Solve Point** and **Mode** columns in the table to navigate between the set number and mode, and rotational velocity solve point and mode.

The ABAQUS solver does not allow modal expansion when post-processing mode shapes.

You can choose to review the mode shapes corresponding to any of these natural frequencies by selecting the frequency from the bar chart or tabular data and using the context sensitive menu (right-click) to choose **Create Mode Shape Results**. You can also view a range of mode shapes.

"Stresses" from a Modal analysis do not represent actual stresses in the structure, but they give you an idea of the relative stress distributions for each mode. Stress and Strain results are available only if requested before solution using **Output Controls**.

You can view the mode shape associated with a particular frequency as a [contour plot](#). You can also [animate](#) the deformed shape including, for a damped analysis, the option to allow or ignore the time decay animation for complex modes. The contours represent relative displacement of the part as it vibrates.

For complex modes, the Phase Angle associated with a particular frequency represents the specified angle in time domain and is equivalent to the product of frequency and time. Since the frequency is already specified in the results details view for a specific mode, the phase angle variation produces the relative variation of contour results over time.

When running a [cyclic symmetry](#) analysis, additional result object settings in the Details view are available, as well as enhanced animations and graph displays. See [Cyclic Symmetry in a Modal Analysis](#) for more information.

Note: The use of construction geometry is not supported for the postprocessing of cyclic symmetry results.

Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2020 R2 - © ANSYS, Inc. All rights reserved.