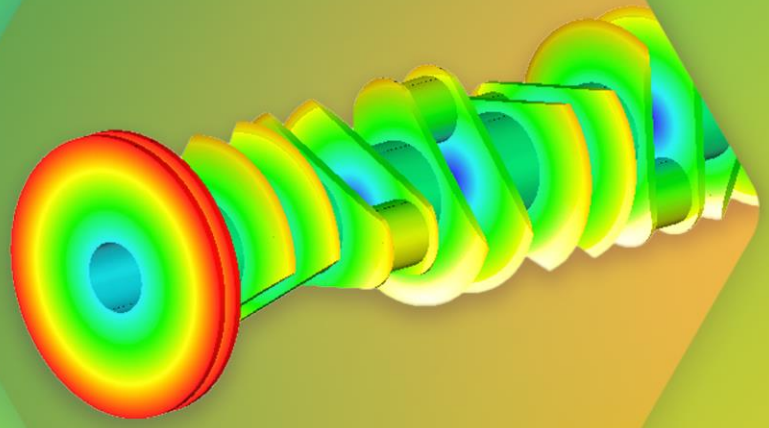
ZWSIM Primary Tutorial

Modal Analysis of Slender Rod



ZWSim Modal Analysis of Slender Rod

Printed in the P. R. China.

ZWSOFT CO., LTD.(GUANGZHOU)

Room 01-08, 32/F, No.15, Zhujiang West Road,

Tianhe District, Guangzhou 510623, China

(8620)38289780

Foreword

In this tutorial, we provide various case studies, which are from easy to difficult and combine theory with practice. We hope to improve users' CAE skills and techniques with ZWSim.

The tutorial bases on our technical engineers' years of experience in the industry and ZWSim, which is the fruit of a lot of efforts and wisdom. We sincerely hope that the tutorial will do help to you, and your precious advice on it is highly welcomed.

This tutorial is for users who have little or no prior CAE experience. If you are green hands of CAE software, or if you are a new user of ZWSim, we recommend that you get started with this tutorial. Here you can learn the basic knowledge and concepts of ZWSim, rapidly master the simple operations and workflows of ZWSim, and practice simple cases.

Thanks for being our user!

The ZWSim Team

Contents

1 Introduction

Modal analysis is a method to study the dynamic characteristics of structures, which is generally used in the field of engineering vibration. Among them, the modal refers to the natural vibration characteristics of the mechanical structure, and each mode has a specific natural frequency and mode shape. The process of analyzing these modal parameters is called modal analysis. In this case, through the simulation of a slender rod, to understand the steps of ZWSim modal analysis

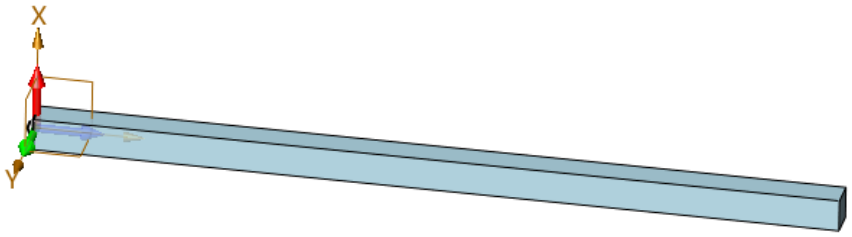


Figure 1 Geometry model

2 Modelling

Help -> More -> Structural -> model

STEP 01 Open the dialog box [Open File];

STEP 02 Select [All Files (*,*)] in [Files of type];

STEP 03 Select the model file [Casepole.stp] in [\Structural\model] and then click [Open] to complete.

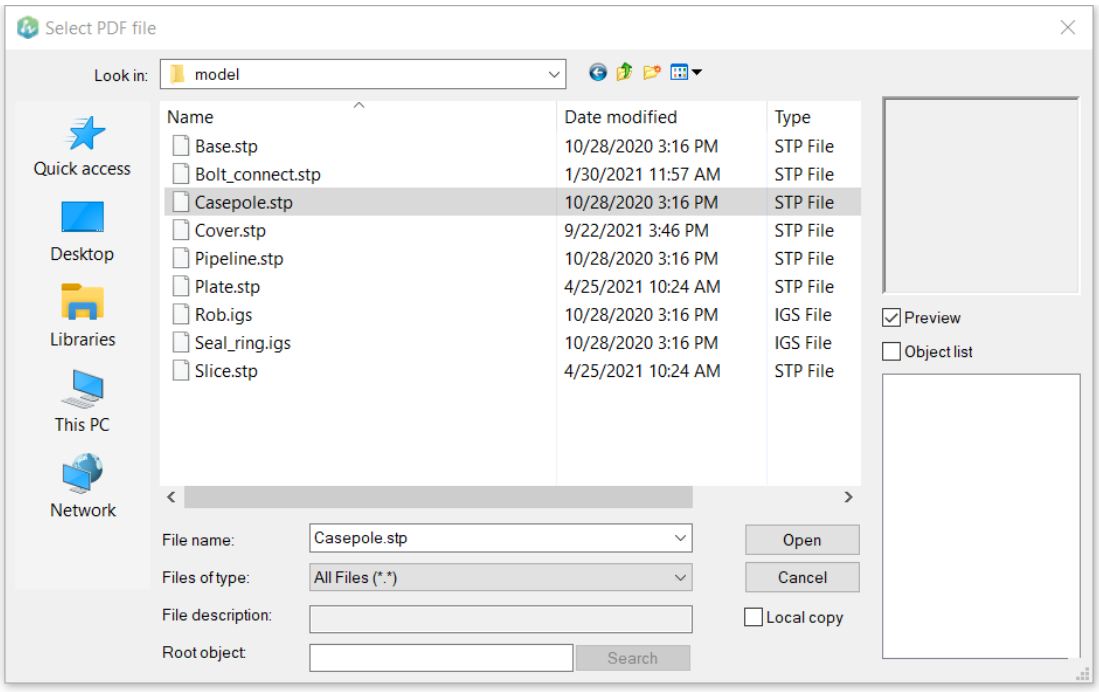


Figure 2 Import model

3 Unit setting

Simulation -> Pre-Processing -> Unit Manager

STEP 01 Open the dialog box [Unit Manager];

STEP 02 Select [MMKS] in [Unit System] and then click [OK] to complete.

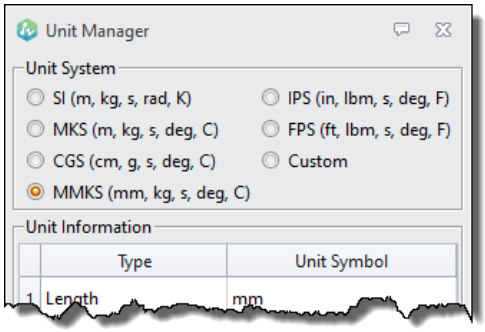


Figure 3 Unit manager

4 Simulation Task

Simulation -> Simulation Task -> New Structure

STEP 01 Open the dialog box [New Structure];

STEP 02 Select [Modal Analysis] in [Linear Modal] and then click [√] to complete.

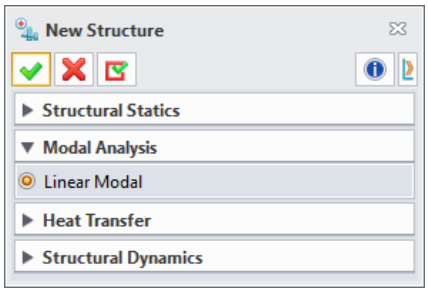


Figure 4 New structure

5 Task Option

ZWStructure -> Base Config -> Task Option

STEP 01 Open the dialog box [Frequency Analysis Options];

STEP 02 Set the number of modes to be solved by setting the [General Options]. This example solves the 6 modes, and the [Number of modes] option is set to 6.

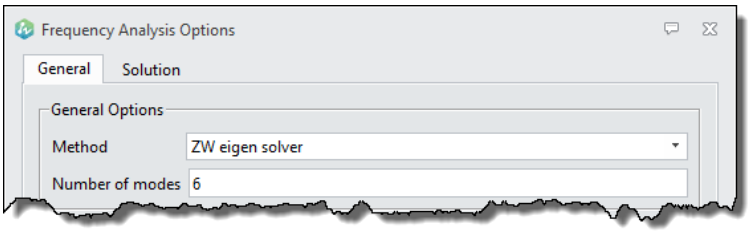


Figure 5 Frequency analysis option settings

6 Material

Simulation Tree -> Frequency and mode shapes1 -> Part -> Geometry Part -> S1(Solid)

STEP 01 Right-click [S1(Solid)] on the tree and then click [Edit Material] to open the dialog box [Material Library];

STEP 03 Right-click [Local] and then click [Create Material] to open the dialog box [Create Material];

STEP 03 For modal analysis, three parameters of elastic modulus, poisson's ratio, and material density must be set. Set the material parameters as shown in the figure below and name it "Alloy Steel";

STEP 04 Click [OK] in dialog box [Create Material] and then click [OK] in dialog box [Material Library] to complete.

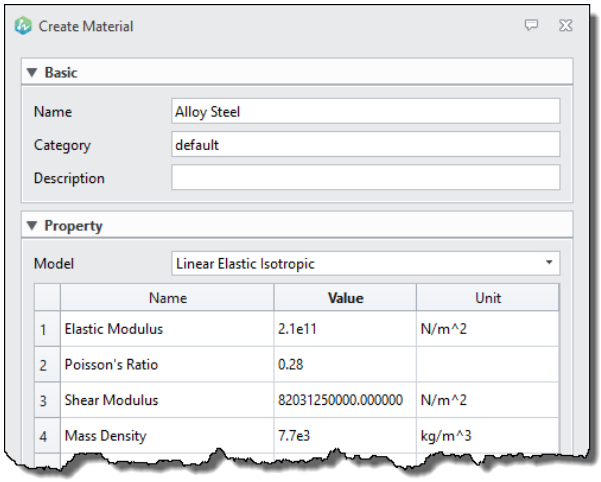


Figure 6 Create material “Alloy Steel”

7 Constraint

Simulation Tree -> Frequency and mode shapes1 -> Constraint

STEP 01 Right-click [Constraint] on the tree and then click [Fixed Geometry] to open the dialog box [Fixed Geometry];

STEP 02 Select [Geometry] in [Object type], and select the square surface F1 at one end of the slender rod in [Entities], as shown in the figure below;

STEP 03 Click [√] to complete.

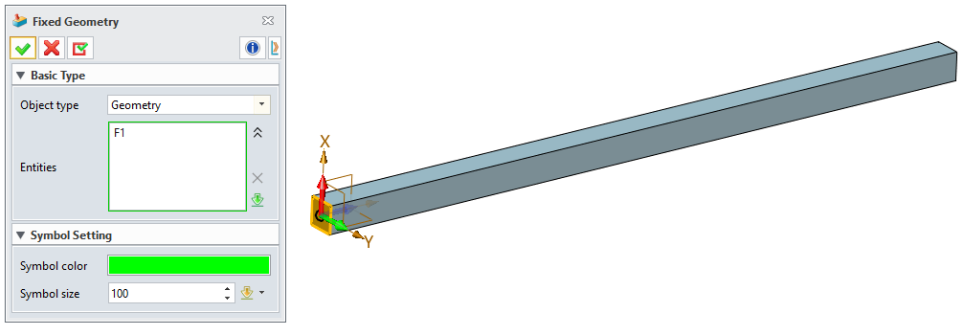


Figure 7 Fixed Geometry

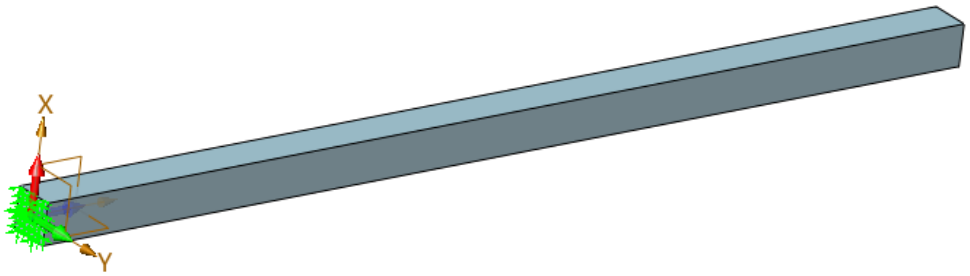


Figure 8 Constraint setting

8 Mesh

Simulation Tree -> Frequency and mode shapes1 -> Mesh

STEP 01 Right-click [Mesh] on the tree and then click [Create Mesh] to open the dialog box [Create Mesh];

STEP 02 Mesh a second-order tetrahedral using the [standard] meshing method. The default element size is 5.8504 mm. The parameter settings are shown in the figure below;

STEP 03 Click [√] to complete.

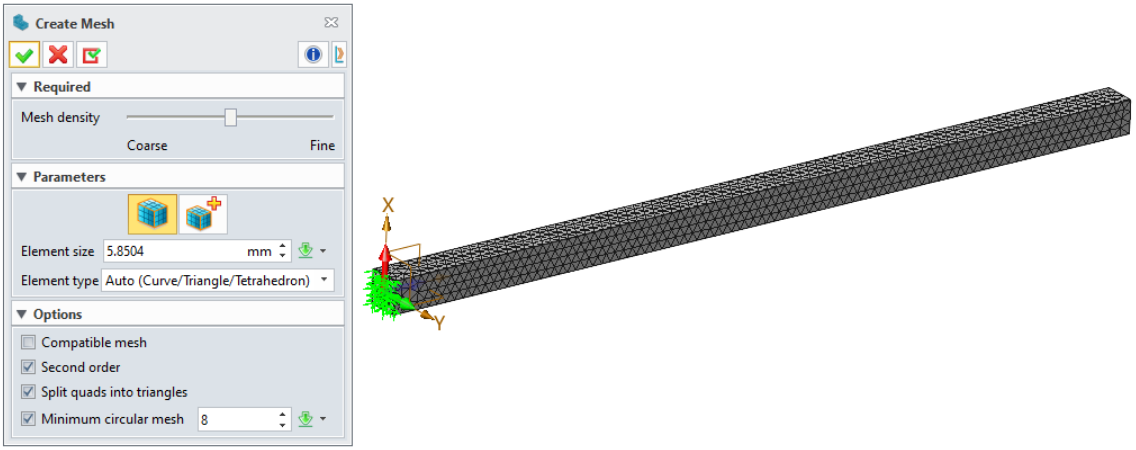


Figure 9 Create mesh

9 Calculation

Simulation Tree -> Frequency and mode shapes1 -> Result

STEP 01 Right-click [Result] on the tree and then then click [Calculate]. If the file has been saved, click [Calculate] and the program will directly perform the solution calculation. If the file is not saved, a [Save As] dialog box will pop up, prompting you to save file.

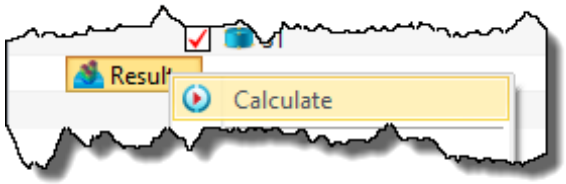


Figure 10 Calculation

10 Post-processing

Under the results node of the simulation tree, you can see the results of the 6 modes generated by default. The natural frequency and mode shape results of the first mode are shown in the figure below.

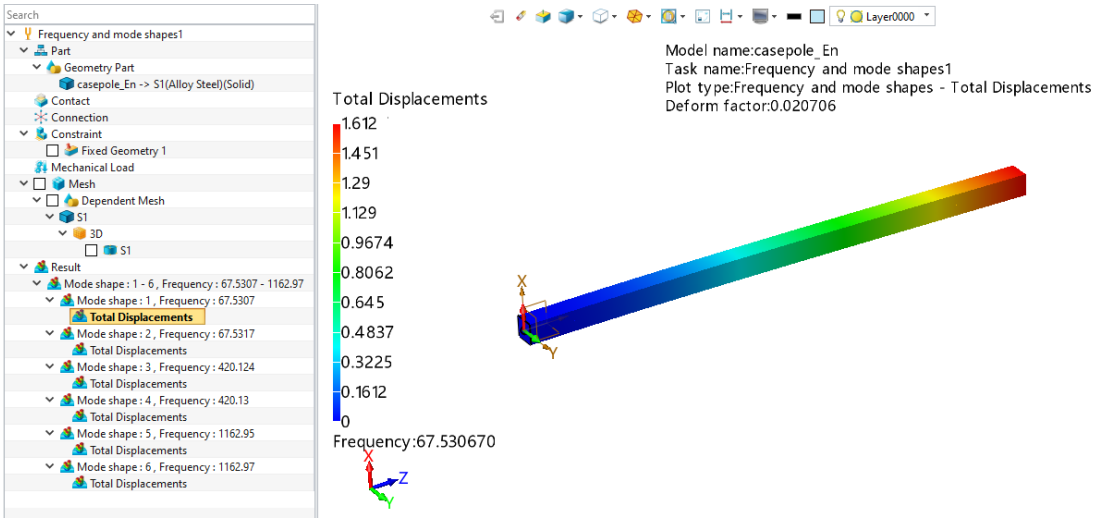


Figure 11 Result

To view all natural frequency calculation results, you can use [List Mode Result].

STEP 01 Right-click [Result] on the simulation tree, click [List Mode Result] to open the [Result Table] to display the natural frequency calculation results corresponding to all modes.

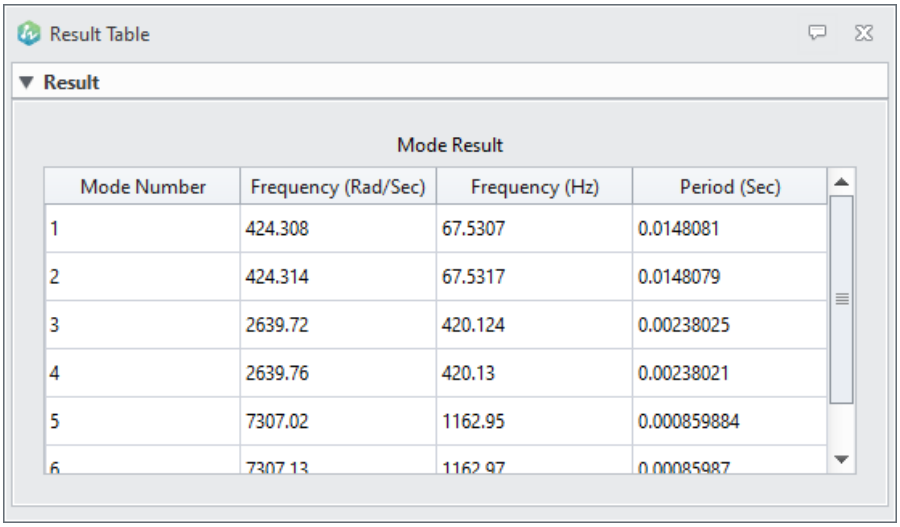


Figure 12 Result table