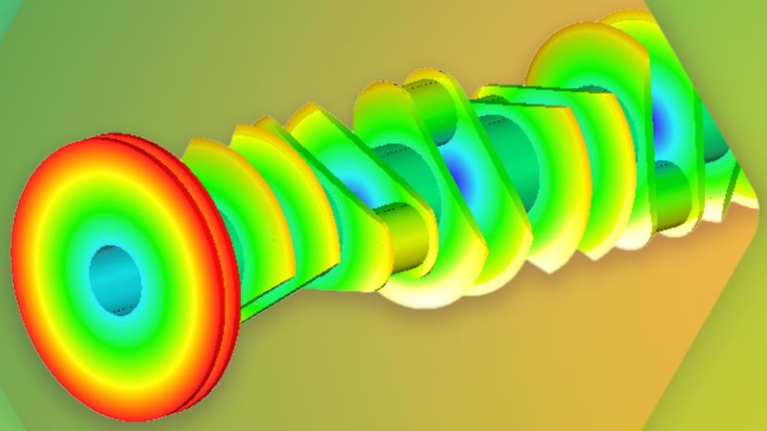
ZWSIM Primary Tutorial

Dynamic Analysis of a Slab Beam (Mode Superposition)



ZWSim Dynamic Analysis of a slab beam (Mode superposition)

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 1

2 Modelling 1

3 Unit setting 2

4 Simulation Task 2

5 Analysis Option 2

6 Material 3

7 Property 4

8 Constraint 4

9 Load 5

10 Mesh 7

11 Calculation 7

12 Post-processing 8

1 Introduction

Kinetics analysis is one of the main contents of CAE analysis, which is used to study the change of structure response over time. It is often necessary to consider the influence of damping and inertial effects.

The model shown in Figure 1 is a slab beam with a thickness of 1 mm. One end of the beam is hinged and one end is simply supported. The upper surface of the beam is subjected to a simple harmonic force with a frequency of 50 Hz and a magnitude of 10N, and the beam begins to vibrate from rest. Considering Rayleigh damping in the vibration process, the damping coefficient is α =0.02, β =0.0001. The calculation adopts the mode superposition method, and takes the first 5 modes to describe the motion of the object.

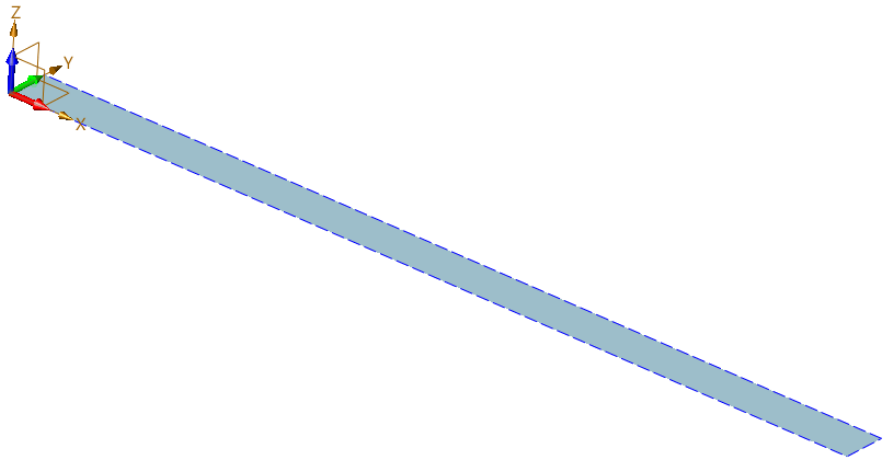


Figure 1 Geometry model

2 Modelling

Help -> More -> Structural -> model

STEP 01 Open the dialog box [Select PDF file];

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

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

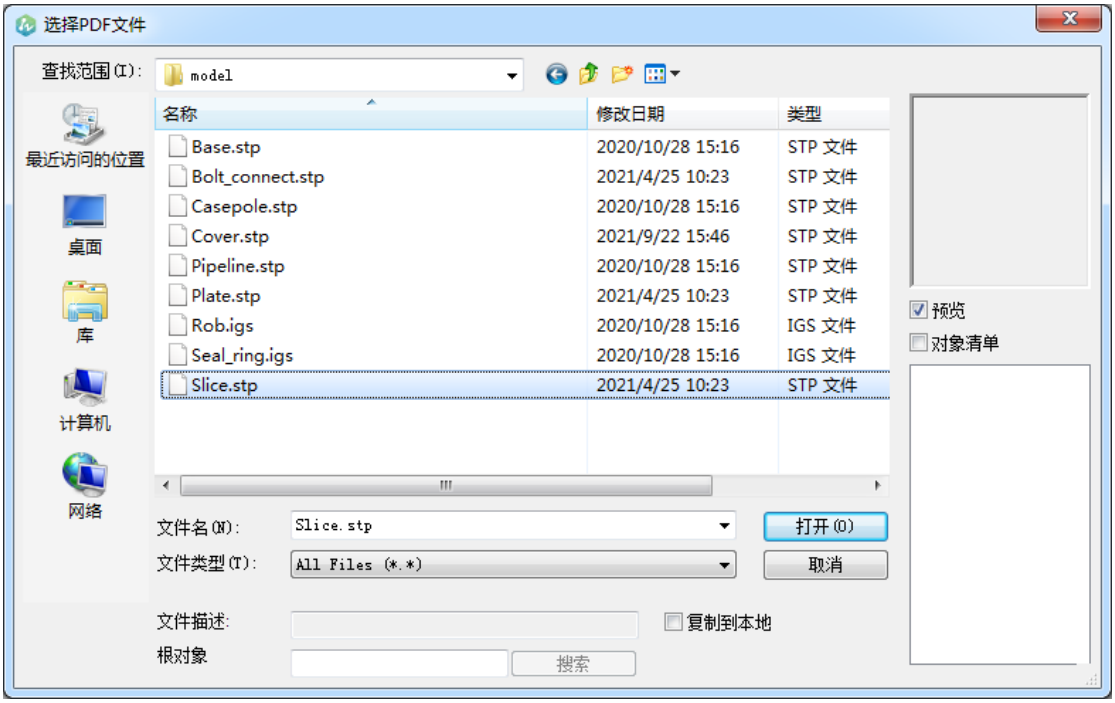


Figure 2 Import model

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.

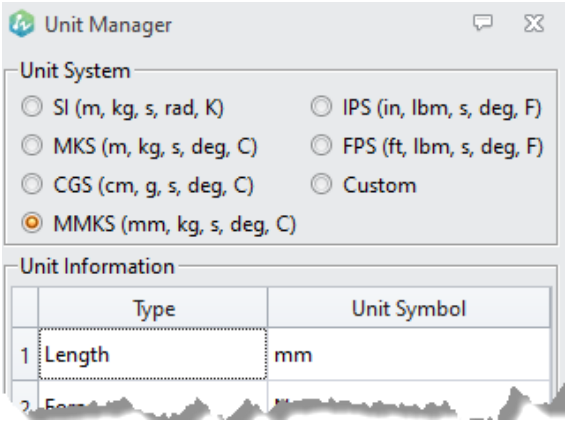


Figure 3 Unit manager

4 Simulation Task

Simulation -> Simulation Task -> New Structure

STEP 01 Open the dialog box [New Structure];

STEP 02 Select [Linear Transient(Modal)] in [Structural Dynamics] and then click [√] to complete.

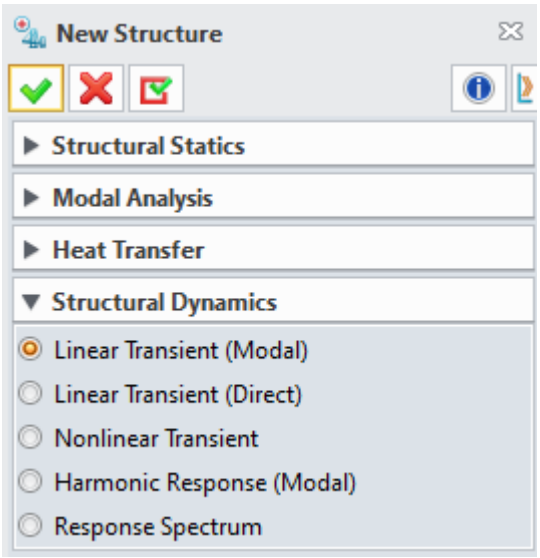


Figure 4 New structure

5 Analysis Option

ZWStructure -> Base Config -> Task Option

STEP 01 Open the dialog box [Modal Transient Analysis Options];

STEP 02 Set [Number of modes] to 5 in tab [Modal];

STEP 03 Set [Start time] to 0 in tab [General]. Then set [End time] and [Time increment] to 0.1 and 0.0004, respectively;

STEP 04 Click [Rayleigh damping] in tab [Damping], set [Alpha] to 0.02, set [Beta] to 0.0001;

STEP 05 Click [OK] to complete.

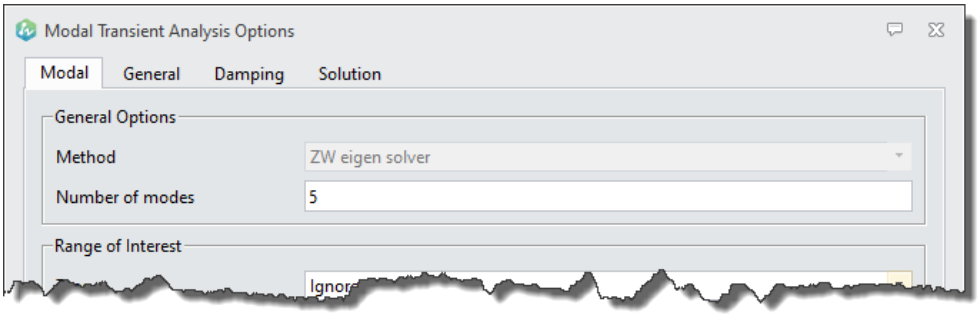


Figure 5 Number of modes

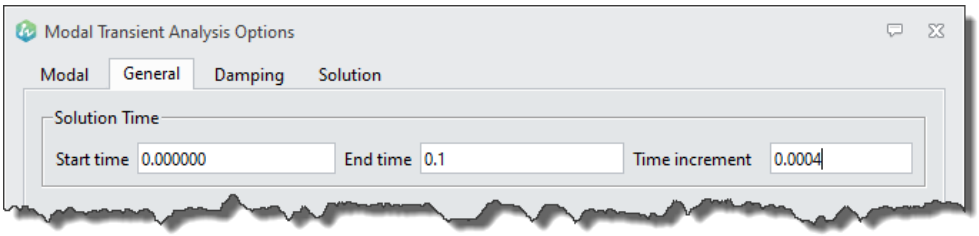


Figure 6 General

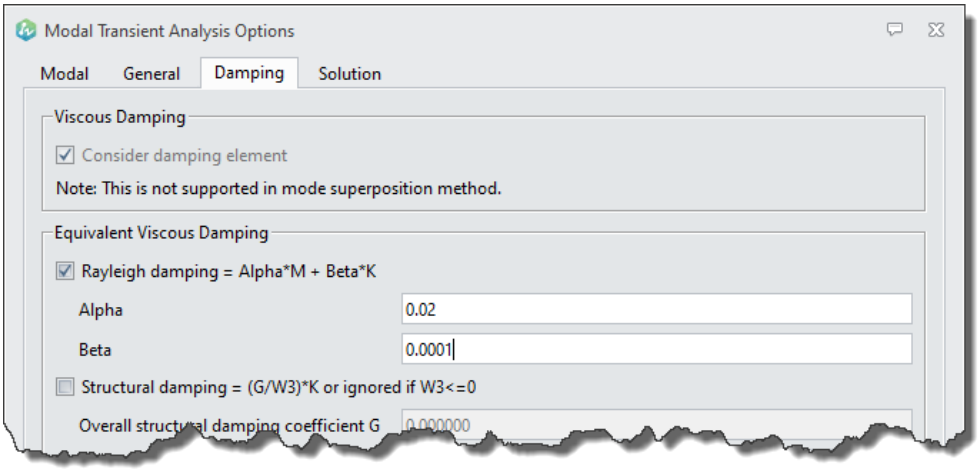


Figure 7 Damping

6 Material

Simulation Tree -> Mode superposition1 -> Part -> Geometry Part -> S1(Shell)

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

STEP 02 Right-click [Local] and then click [Create Material] to open the dialog box [Create Material];

STEP 03 Set the material properties as shown below and name it as “steel”;

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

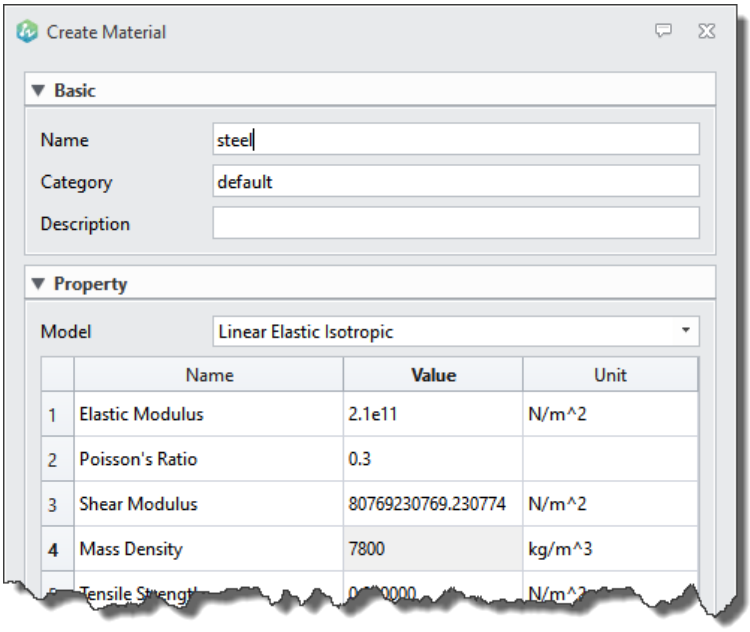


Figure 8 Create material

7 Property

Simulation Tree -> Mode superposition1 -> Part -> Geometry Part -> S1(Shell)

STEP 01 Right-click [S1(Shell)] on the simulation tree and then click [Shell Element Property] to open the dialog box [Shell Element Property];

STEP 02 Set [Thickness] to 1mm;

STEP 03 Click [√] to complete.

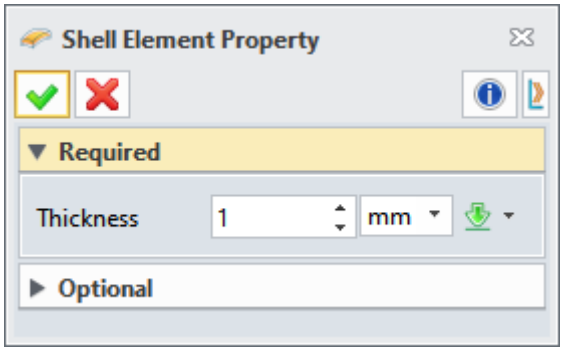


Figure 9 Shell element property

8 Constraint

Simulation Tree -> Mode superposition1 -> Constraint

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

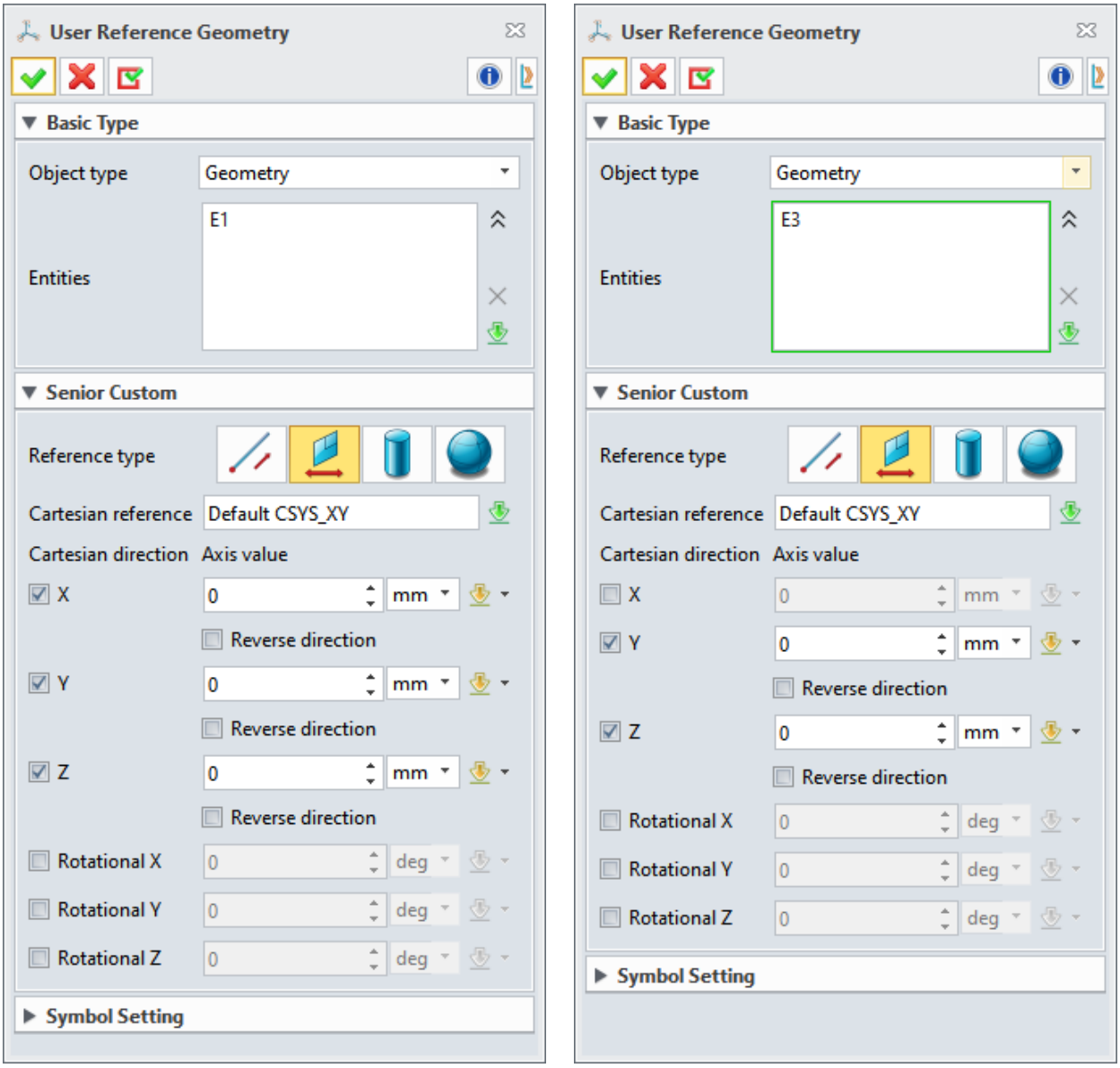
STEP 02 Select [Geometry] in [Object type];

STEP 03 Activate the pick box [Entities] and then pick the the left edge of the slab beam by mouse or inputting, fixing the translation of X, Y and Z;

STEP 04 Click [√] to complete;

STEP 05 Activate the pick box [Entities] and then pick the the right edge of the slab beam by mouse or inputting, fixing the translation of Y and Z;

STEP 06 Click [√] to complete.



(a) (b)

Figure 10 User reference geometry

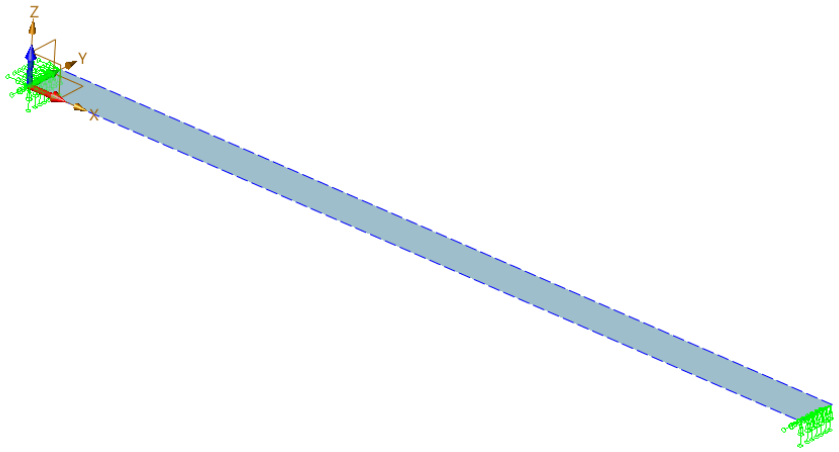


Figure 11 Constraint setting

9 Load

Simulation Tree -> Mode superposition1 -> Mechanical Load

STEP 01 Right-click [Mechanical Load] on the simulation tree and then click [Force Load] to open the dialog box [Force Load];

STEP 02 Select the first icon in the type icon [Global direction];

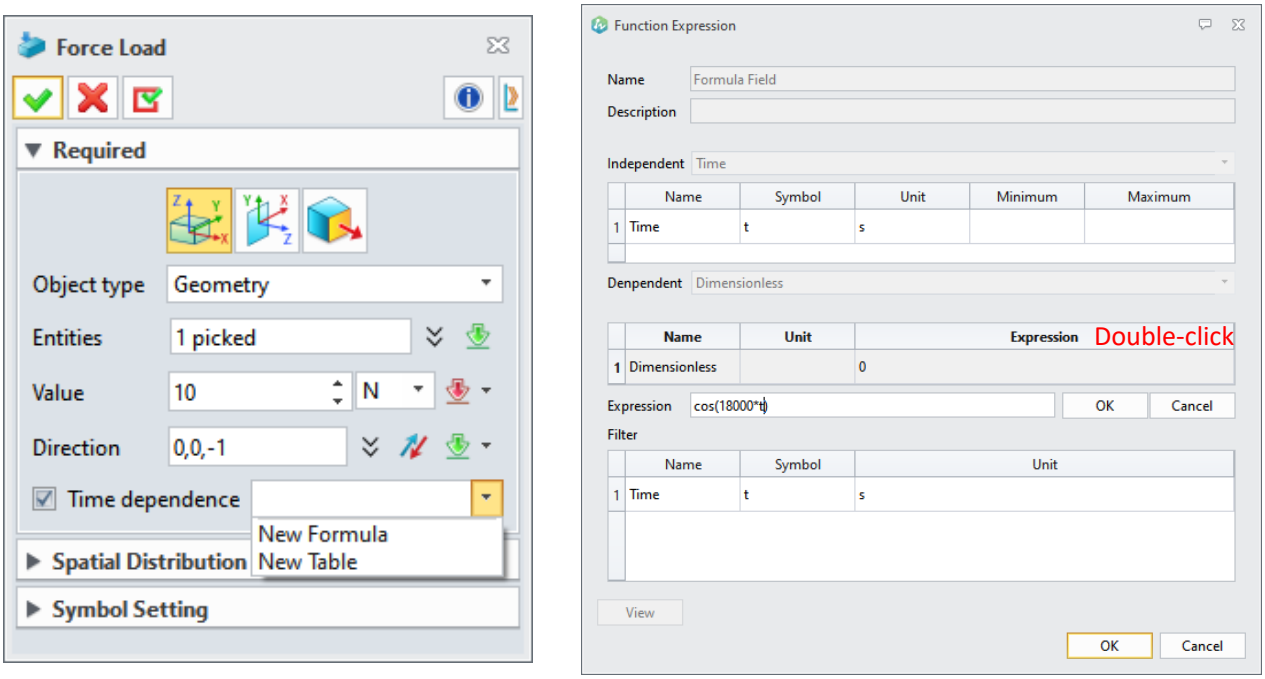
STEP 03 Select [Geometry] in [Object type];

STEP 04 Activate the pick box [Entities] and then pick up the upper surface of the geometry by clicking with the mouse or typing, the direction is -Z;

STEP 05 Check [Time dependence], select [New Formula] in the drop-down options, and enter the formula setting window;

STEP 06 Double-click [Expression] in the formula editing window, the formula editing bar appears, enter cos(18000*t) and click [OK] on the right, then click [OK] below to complete the setting and return to the [Force Load] dialog box;

STEP 07 Click [√] to complete.



(a) (b)

Figure 12 Force load

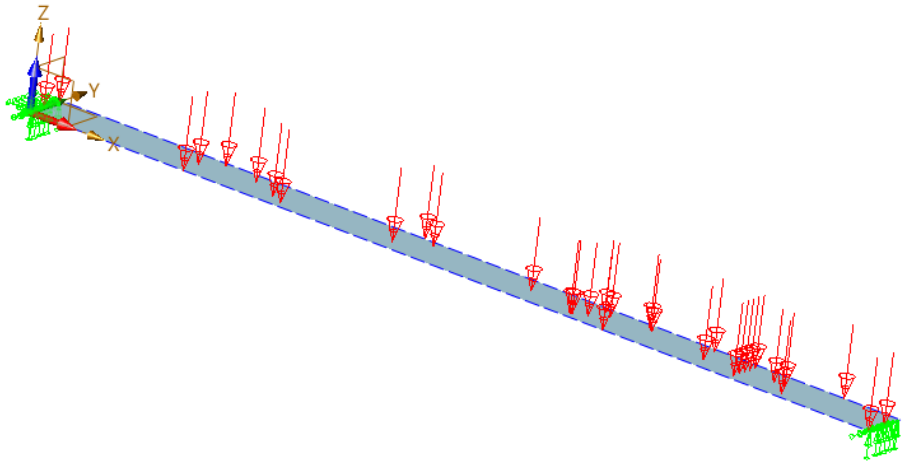


Figure 13 Load setting

10 Mesh

Simulation Tree -> Mode superposition1 -> Mesh

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

STEP 02 Set the options as shown below, set [Mesh size] to 2;

STEP 03 Click [√] to complete.

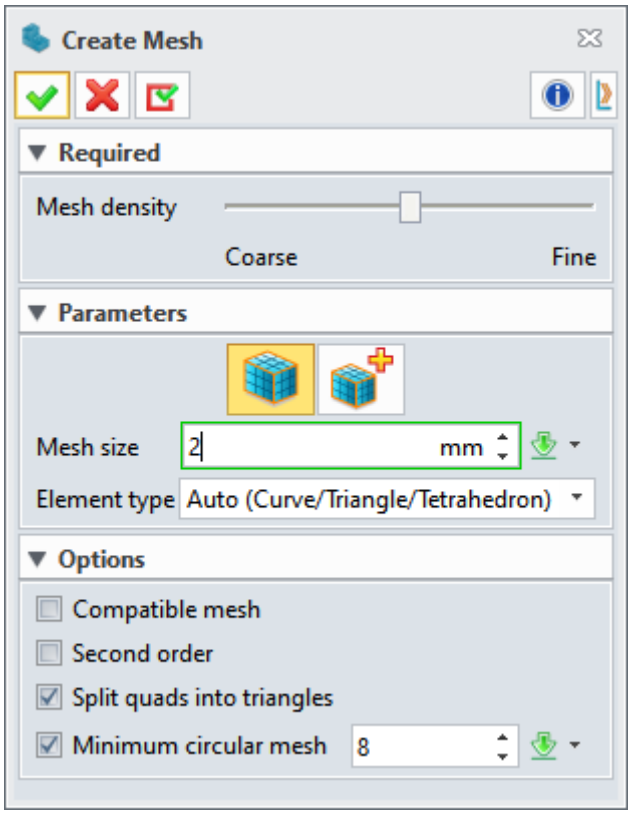


Figure 14 Create mesh

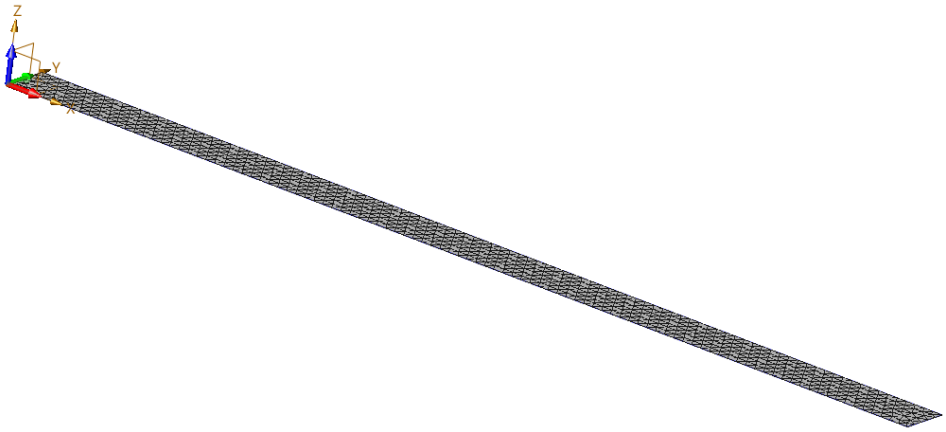


Figure 15 Mesh

11 Calculation

Simulation Tree -> Mode superposition1 -> Result

STEP 01 Right-click [Result] on the simulation tree and then click [Calculate] to analyze current simulation task.

12 Post-processing

The cloud diagram of results of each time step can be viewed on the tree after calculation, as shown below.

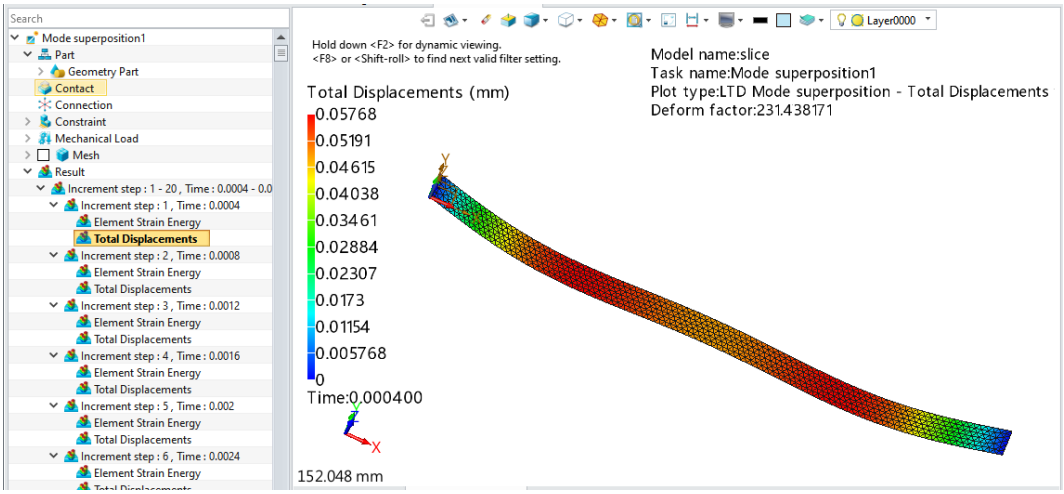


Figure 16　Result

To view more results, you can use [New Result Display].

STEP 01 Right-click [Result] on the simulation tree and then click [New Result Display] to open the dialog box [New Result Display];

STEP 02 Set the options as shown below, output velocities result cloud diagram;

STEP 03 Click [√] to complete.

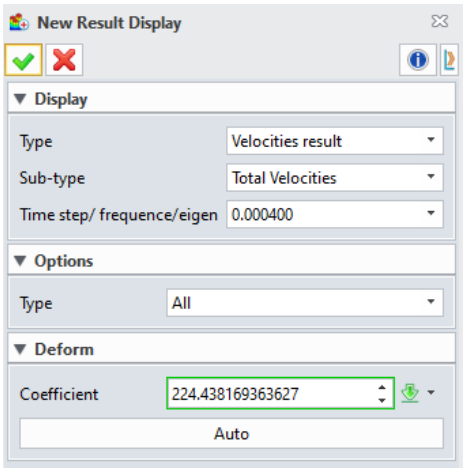


Figure 17　New result display

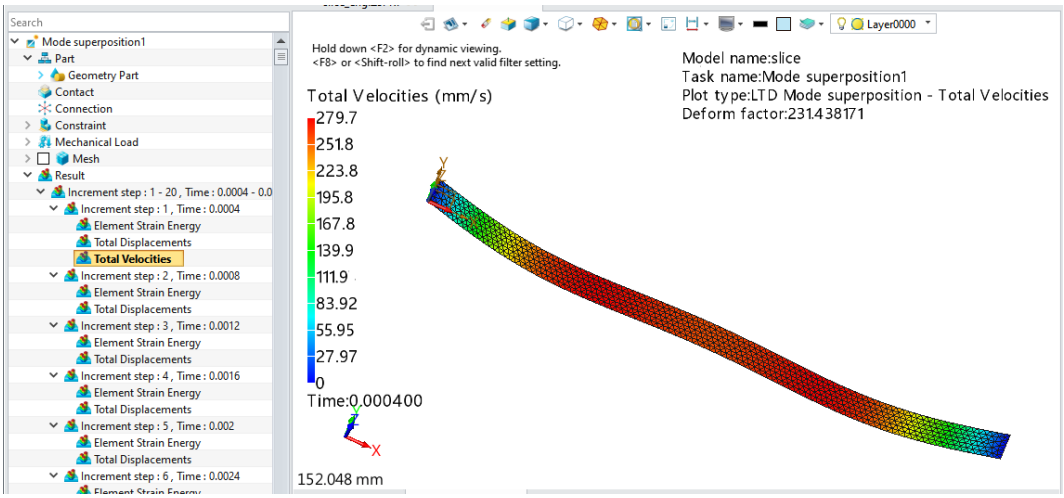


Figure 18　View more results

ZWStructure ->Result -> New XY Plot

To view the response curve in a specific location, you can use [New XY Plot].

STEP 01 Click [New XY Plot] in the toolbar to open the [New XY Plot] dialog box;

STEP 02 Choose displacement results. Select [At location] in [Options] and then pick some interested nodes in view area by mouse;

STEP 03 Click [Update response data] to show the results.

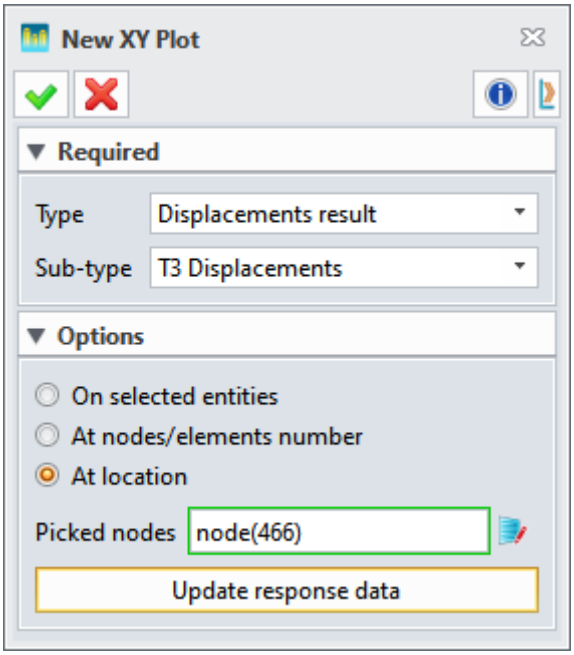


Figure 19 New XY plot

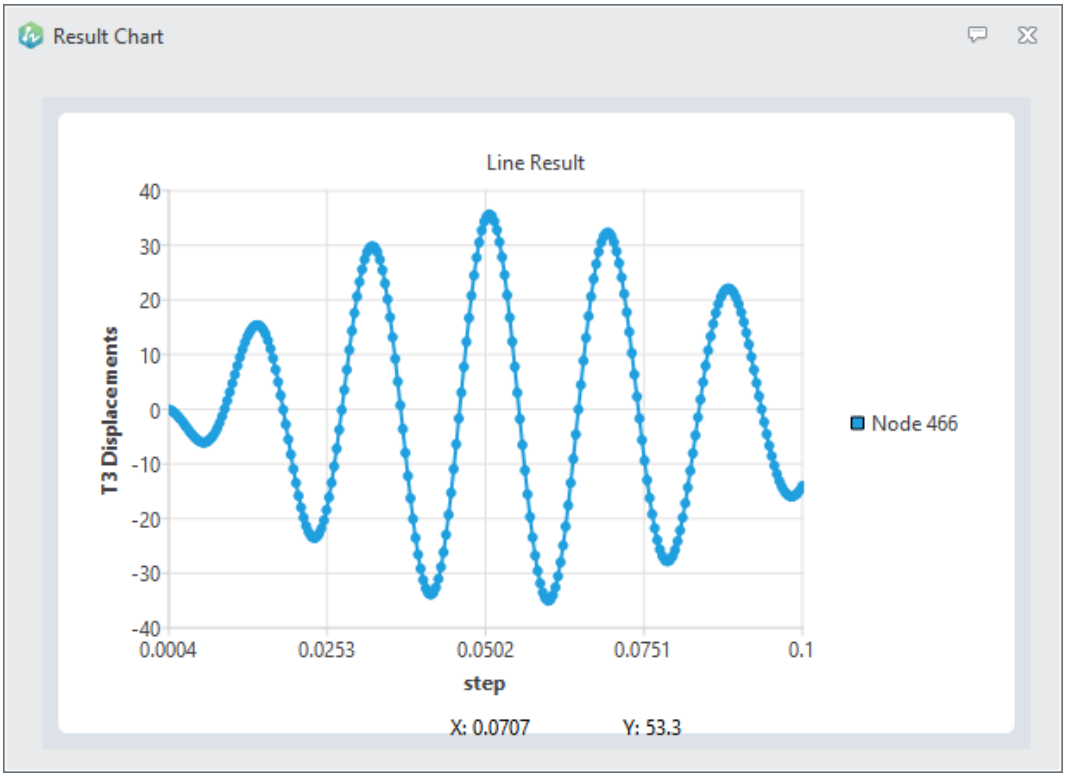


Figure 20 Response curve