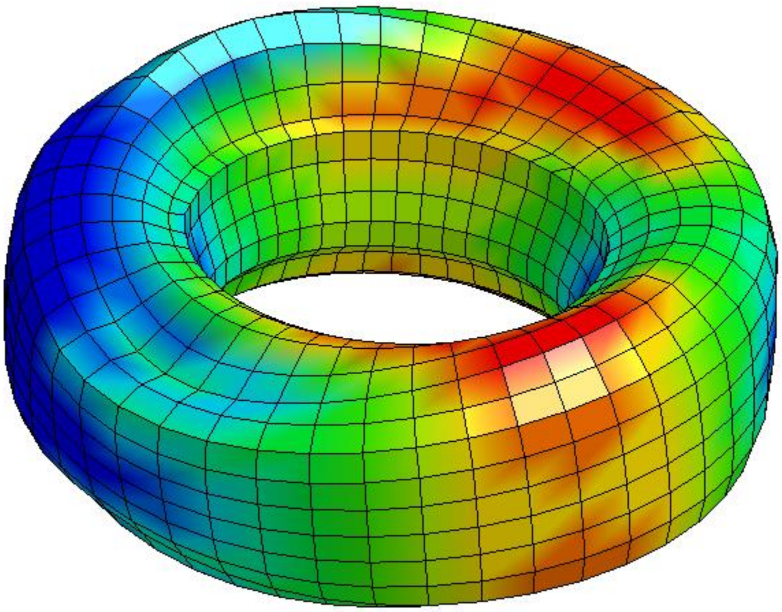
Basic Tutorials

LS-DYNA / LS-PrePost

Ex. 4. Eigenvalue analysis



Contents

1 Introduction

1.1 Purpose

- Learn how to set up an eigenvalue analysis.

1.2 Prerequisites

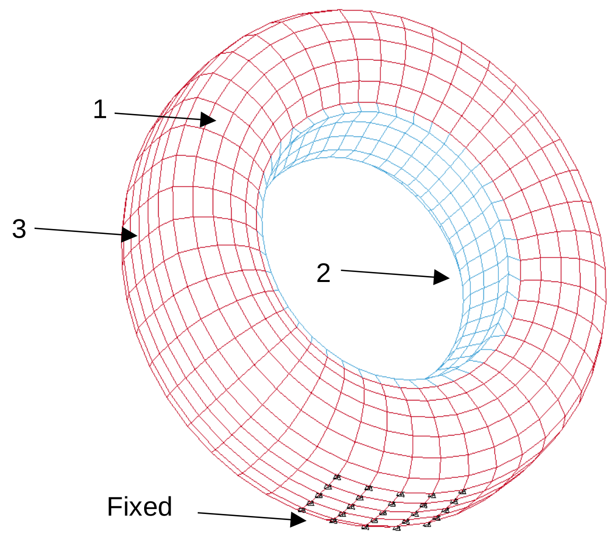
- Basic knowledge in the finite element method.
- Understand the steps in tutorial **1 - Getting Started.**

1.3 Problem Description

The task is to perform an eigenvalue analysis on a coarsely meshed tire. The bottom of the tire is fixed.

The model consists of three parts with shell elements

1. Rubber tire (t = 4 mm)
2. Steel inner rim of wheel (t = 3 mm)
3. Steel belt (t = 2.25 mm, inside tire)



1.4 Data files

- Input file for the exercise: **tire.k.**
- The final model: **tire_results.k**.

2 Create the model

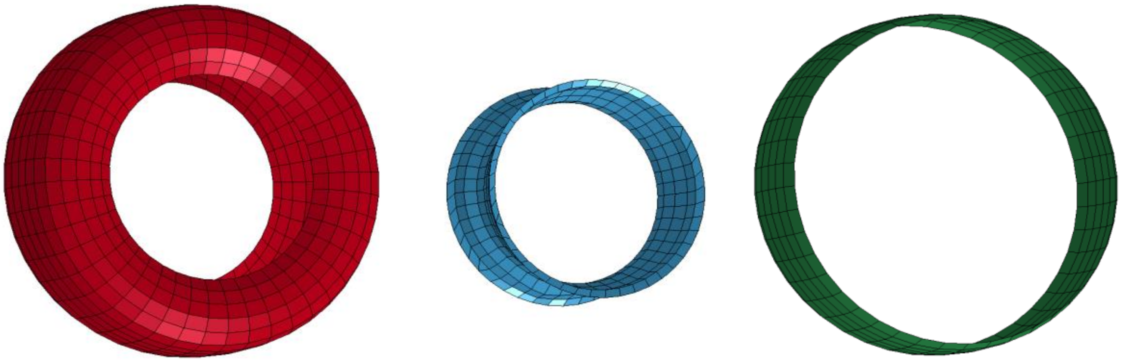
Open the keyword file **tire.k** in LS-PrePost, which contains the mesh and ***PART** cards.

Look at the different keywords to see what the model consists of. Note that **ELFORM = 16** is used in **SECTION_SHELL**. This is a formulation for shell elements that is designed for implicit linear and non-linear simulations.

To examine the model, do as follows:

- Click **Model > SelPart**, this tool is used to select which parts that are displayed.
- Select **1 Tire** to only display that part.
- Use **Ctrl + Click** to display multiple parts.
- By clicking on a part in the work space, the part will be deselected and therefore not displayed.

Before closing **Assembly and Select Part** window, click **All** to display all parts.



2.1 Material

Create the following two materials of the type **MAT_001-ELASTIC:**

- Click **Model > Keywrd.**
- Double-click **MAT > 001-ELASTIC**.
- Enter the values for the first material, the rubber, see the image below.
- Click **Accept**, then **Add** and enter the values for the second material, see the second image below.
- Click **Accept,** then **Done**.

TITLE: Tire - Rubber

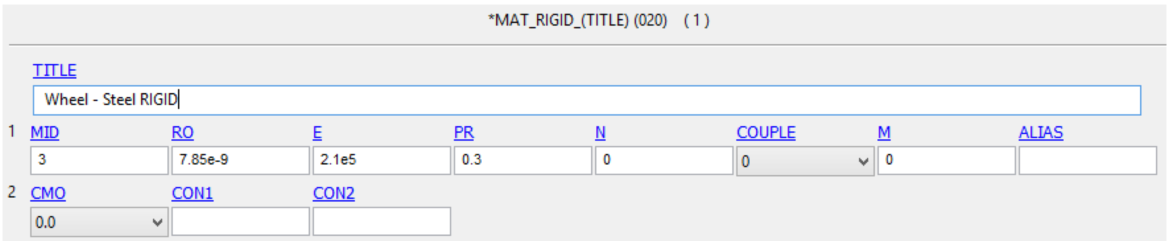
MID	RO	E	PR
1	9.100e-010	50.000000	0.3000000

TITLE: Belt - Steel

MID	RO	E	PR
2	7.850e-009	2.100e+005	0.3000000

The inner rim “Wheel” will be considered as rigid, which means that it cannot be deformed. Do as follows to create the rigid material:

- In Keyword Manager, double-click **MAT > 020-RIGID**.
- Enter the values shown in the image below and click **Accept**.



The parameters **CMO**, **CON1** and **CON2** can be used to constrain the rigid body. These options will be used in an optional exercise later on. Even though the part cannot be deformed, Young’s modulus **E** and Poisson’s ratio **PR** are defined. This is because they are needed to compute contact stiffness, if a contact definition is used.

Assign the two created materials to the different parts using the Keyword Manager.

2.2 Set up the eigenvalue analysis

Do as follows to activate the implicit solver:

- Activate implicit mode, double-click **CONTROL > IMPLICIT_GENERAL** in Keyword Manager. Set **IMFLAG = 1**. Since we will perform an eigenvalue analysis, **DT0** can be left blank.
- Click **Accept**, then **Done**.

Set up the eigensolver parameters:

- Double-click **CONTROL > IMPLICIT_EIGENVALUE** in the Keyword Manager to activate an eigenvalue analysis. Set **NEIG = 10**, which is the number of eigenvalues that will be extracted.
- Click **Accept**, then **Done**.

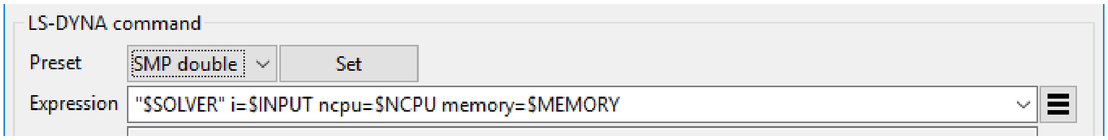
Save the keyword and close LS-PrePost.

Note: **ENDTIM** in **CONTROL_TERMINATION** does not need to be set in this eigenvalue analysis. **ENDTIM** is almost always mandatory for all other types of analyses.

2.3 Run the analysis

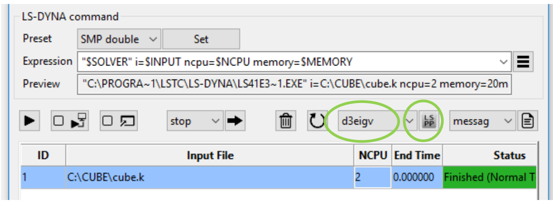
Implicit analyses should always be run using the double precision version of LS-DYNA, as the implicit solver is more sensitive to round-off errors.

Run the simulation using **LS-Run**, see exercise **1. Getting Started** for more information on how to do this. The simulation runs in less than a minute to completion. To use the double precision version, make sure to use the “SMP double” preset in LS-RUN, see the image below.



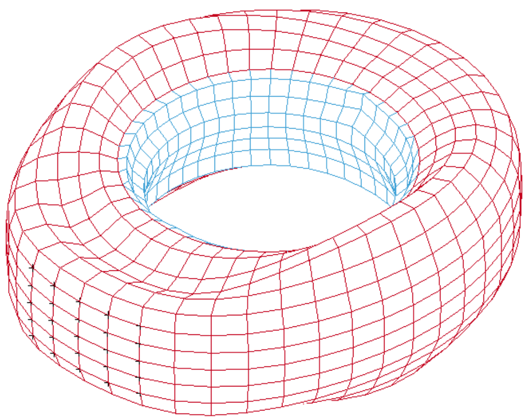
2.4 Post-processing

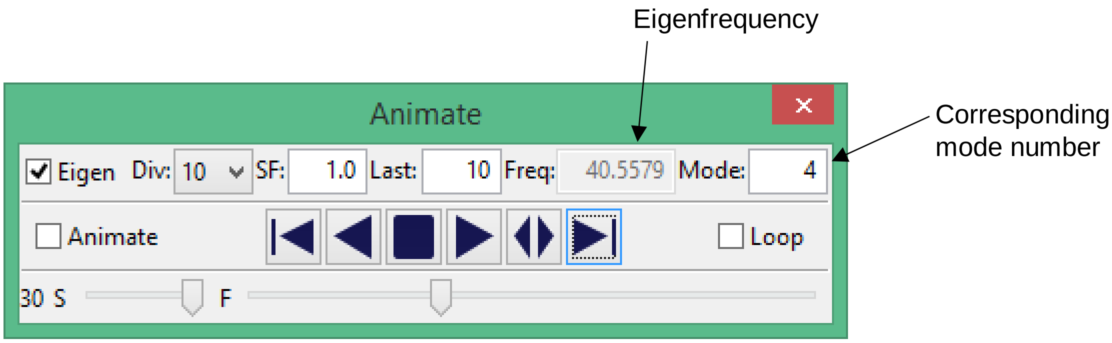
Open the result file **d3eigv**, *not* **d3plot**, in LS-PrePost. One way to do this is to use he LS-PrePost-button in LS-Run (remember to set the option to **d3eigv**), see the image below.



To view the eigenmodes do as follows:

- Click **File > Open > LS-DYNA Keyword File**. Select the keyword file you just created. This provides additional information about the model to LS-PrePost, which improves the visualization.
- Click **Model > Display** and select the boundary condition to see where it is fixed, this is just for visualization. Click **Done**.
- In the **Animate Toolbar**, if **Eigen** is activated, the **Forward** button will animate one mode. Change the mode by clicking on **Previous State** or **Next State**.
- If **Eigen** is deactivated, the **Forward** button will animate the extreme displacement of all modes.
- Play around with the **Animate Toolbar** and look at the different modes.





For an overview of available eigenmodes:

- Click **Post > State**. The different modes and eigenfrequencies can be seen here.
- Close LS-PrePost.

Some results are also reported to a separate text file: In the Working Directory, there is a file called **eigout**. Open **eigout** with a text editor. There you can find more information about the eigenvalue analysis. The contents of the file are as follows:

$$\lambda \qquad \omega = \sqrt{\lambda} \qquad f = \omega / 2\pi \qquad T = 1/f$$

MODE	EIGENVALUE	RADIANS	CYCLES	PERIOD
1	2.910091E+03	5.394526E+01	8.585654E+00	1.164734E-01
2	7.516690E+03	8.669885E+01	1.379855E+01	7.247138E-02
3	9.609495E+03	9.802803E+01	1.560165E+01	6.409580E-02
4	6.493976E+04	2.548328E+02	4.055790E+01	2.465611E-02

3 Summary

This was a short exercise on how to set up an eigenvalue analysis and how to post-process it. There are several applications where this type of analysis can be used. One type of application is to check eigenfrequencies for buildings or bridges. The forces from winds and earthquakes causes vibrations on the structures. These vibrations should not coincide with the eigenfrequency of the structure. Because if they become equal, the amplitude of the vibrations becomes larger, which can lead to collapse. One famous example is the Tacoma Narrows Bridge in 1940.

4 Optional exercises

Optional exercises:

1. Delete the belt in the tire (delete the part in the Keyword Manager) and perform a new simulation. What will happen with the eigenfrequencies?

2. Delete the Boundary (SPC_SET). Set **CMO = 1, CON1 = 7** and **CON2 = 7** in **MAT_020-RIGID**. This will constrain the center of mass for the wheel in all degrees of freedom. How does this affect the eigenmodes?