Tutorial 101

FEDEM is a comprehensive modern CAE tool for virtual testing and verification of mechanical systems. The integrated use of dynamics and structural solving in FEDEM is an exceptionally efficient way of analyzing mechanisms, in this case a scissor lift.

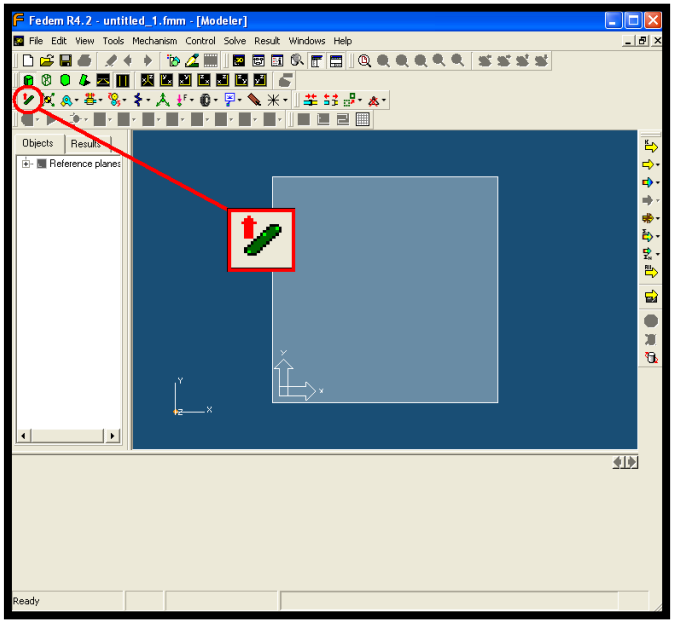
This tutorial will introduce the relevant tools used in order to simulate a simple canterleaver beam spring damped system. The tutorial will present the necessary steps to build the model, insert loads / constraints and perform simple post processing.

Getting Started

- Download the latest version of FEDEM
- Download file containing all the necessary files from www.fedem.com Link…
- Make sure to familiarize with the FEDEM interface before initiation the tutorial

Importing Links

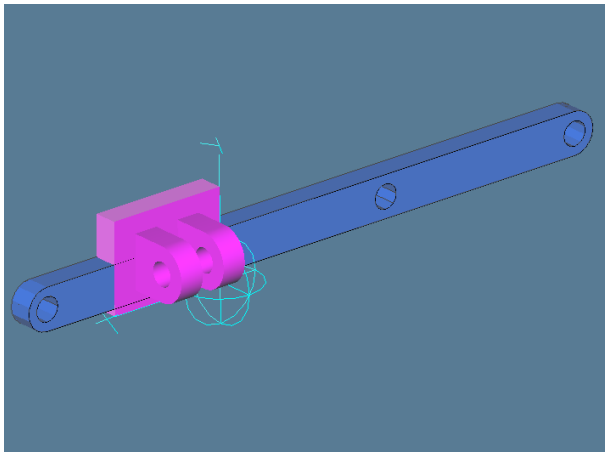
1. Open FEDEM and select the ‘’load link’’ symbol
2. Enter the directory where the files are located (Parts downloaded from ….Link…..)
3. Use the Ctrl key and select “Beam.ftl” And “fixedbracket.wrl” . (make sure to select ’’all files’ in the file type drop menu)

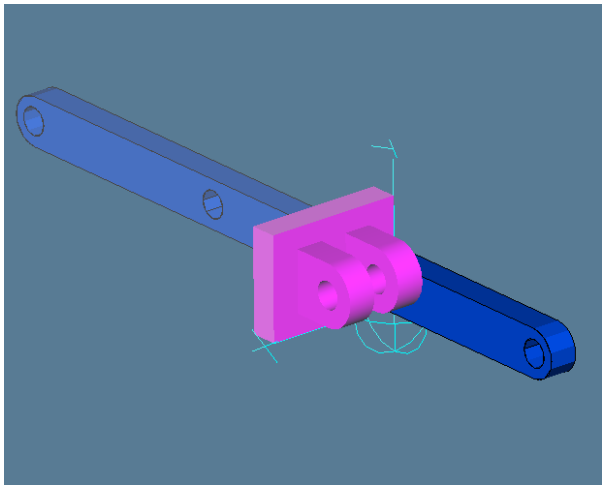


4. Open

Aligning Parts

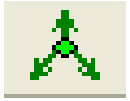
1. When all the parts are imported it is apparent that the base and top needs aligning
2. Click on the Beam. The Property Editor (bottom of the screen) will then appear.
3. Click on the ’’Origin’’ tab in the Property Editor and change the orientation ’’RotY[Deg]’’ from 0 to 90



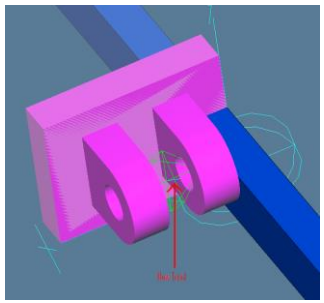


Applying Triads

In order to connect the links and have a mathematical relationship between the degrees of freedom and the link, triads have to be connected to each part. For more information regarding the function and application of triads se FEDEM User manual.

1. Select the triad sign in the toolbar 

2. Click on the bracket to apply. Change the local reference appearing in the bottom left corner to X: 0.05 Y: 0.0375 Z: -0.03 Done / Enter

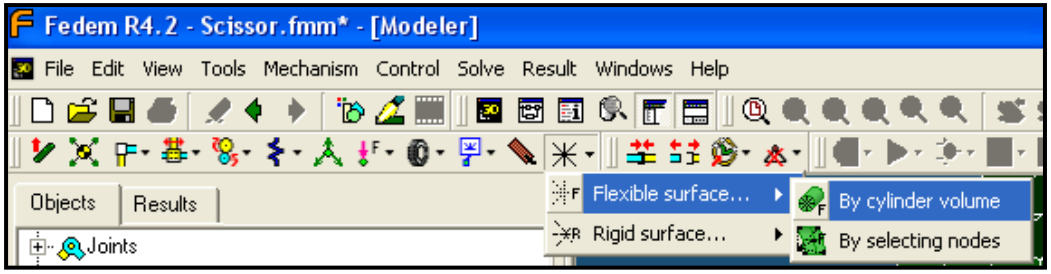


3. Now apply a triad to the work plane and enter the following coordinates
   X: 0.05 Y: 0.3 Z: 0.0. Done

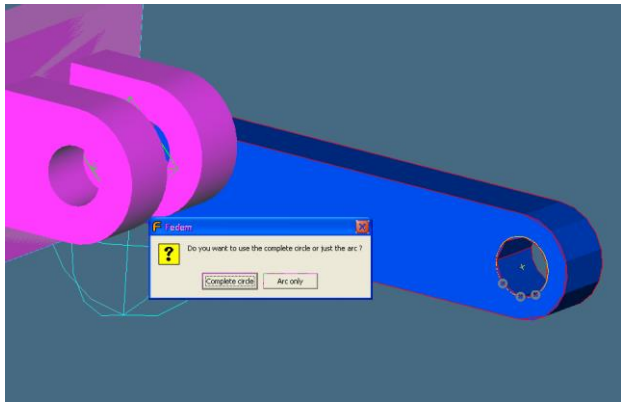
4. Attach the triad to the work plane by selecting  , Triad –Done, Work plane-Done (the triad should now appear green)

Applying Triads

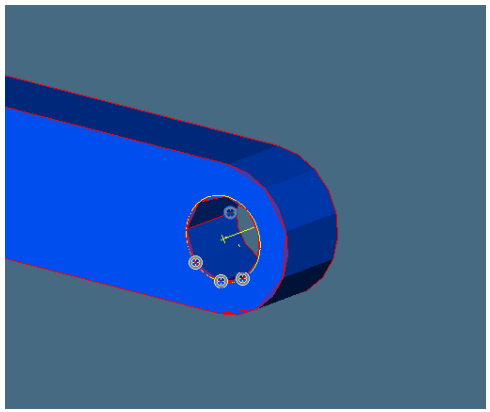
3. Select the ’’Surface connector’’ drop down menu in the toolbar. Select: flexible surface – by cylindrical volume.



4. Select three points on the circumference of one hole in the beam – Done (between each selection). Click ’’Complete Circle’’

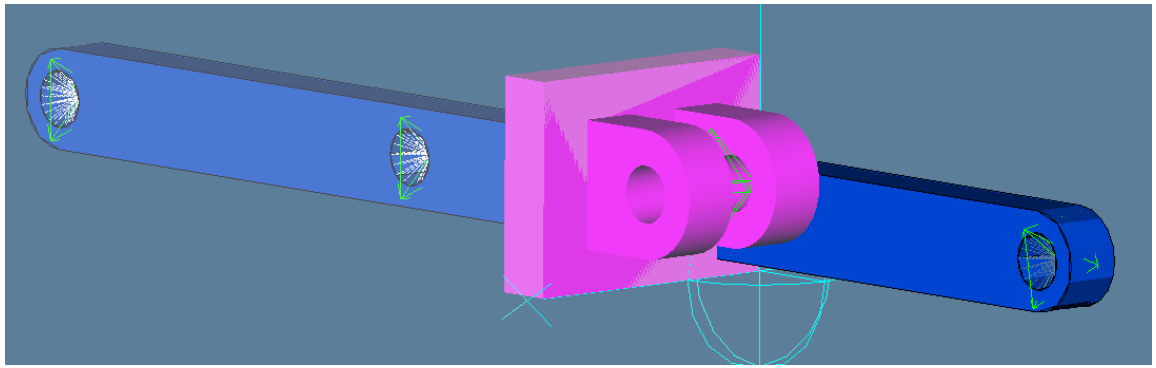


5. Select a point/node on one side of the desired cylindrical volume – Done. Then select a node on the opposite side of the volume - Done

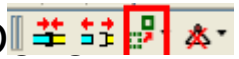


Location Parts

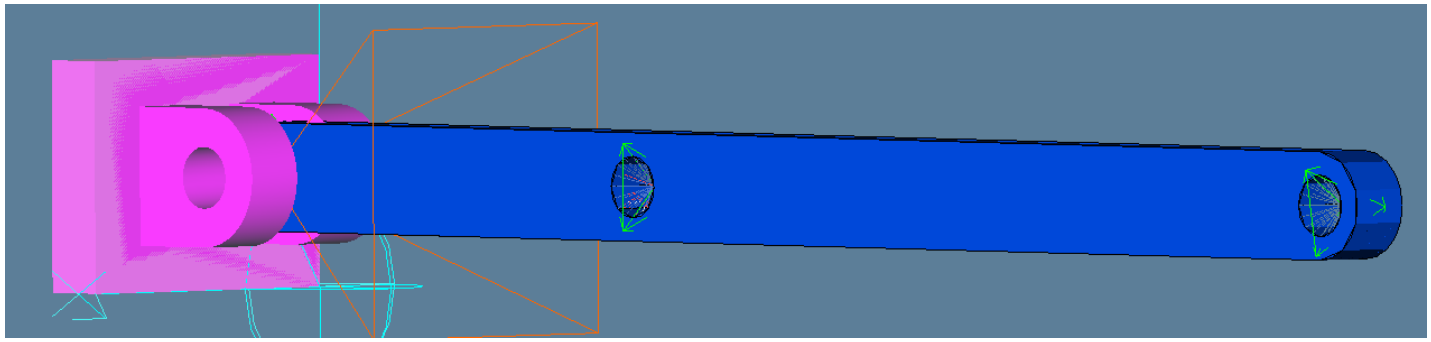
Repeat this excersie for all holes in the beam.



1. Select the “Smart Move” Button from the Toolbar
2. Now select the far left hole (as displayed in figure above). Done
3. Select the triad in in the bracket (as created in slide 6). D



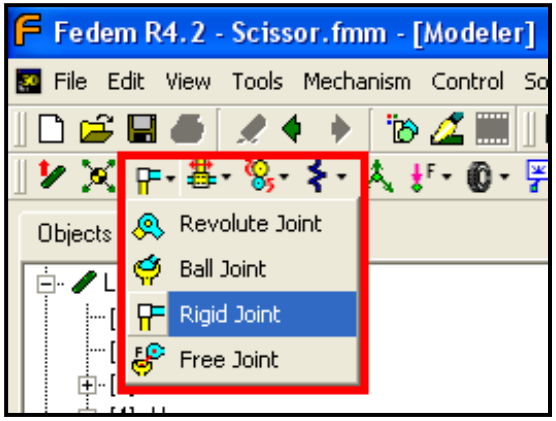
The triads should now align. The model should look as displayed in the figure below.



Applying Revolute joint

In order to relate the separate parts and their triads to each other their individual degrees of freedom has to defined and constrained with respect to each other.

1. Use the joint selection menu select revolute joint



2. Apply the revolute joint the triad conecting the beam and brackt. Note that only one joint is required per location.

Attaching Revolute joint

In order to relate the revolute joints to each part it is necessary to connect it to the individual links where relative motion is occurring.

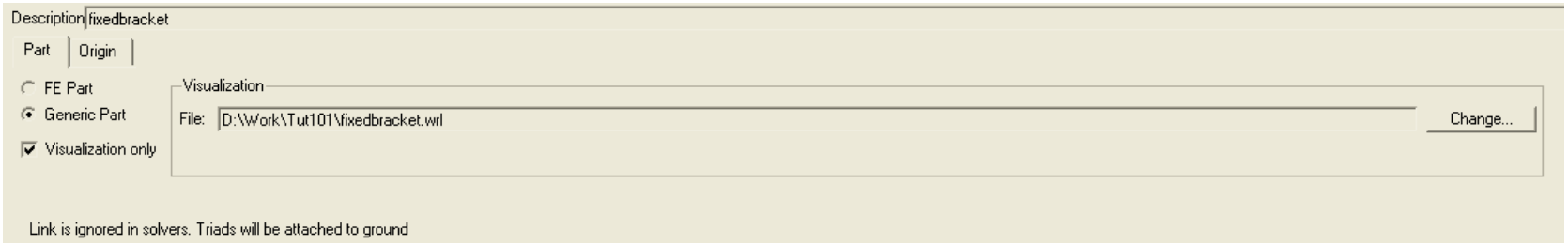
1. Change the origin orientation of the joint to rotx & rotz :0 and roty:90
2. Use the Attach button from the toolbar menu 
3. Select the joint that needs to be connected Done/Enter
4. Select one of the connecting links Done/Enter
5. Select the other connecting link Done/Enter

The joint should appear yellow if it is connected. Master/slave is not relevant unless the joint is connected to ground. (when connecting to base, the master has to be the base)

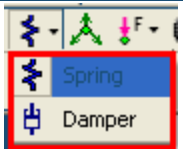
Fixing Bracket

By ticking of the visualization only hatch in the tool panel (for the bracket) the bracket will connected to “earth” and will be fixed with respect to the work plane.

(Note: This can also be achieved by using a rigid joint and attaching it to the bracket and the work plane)

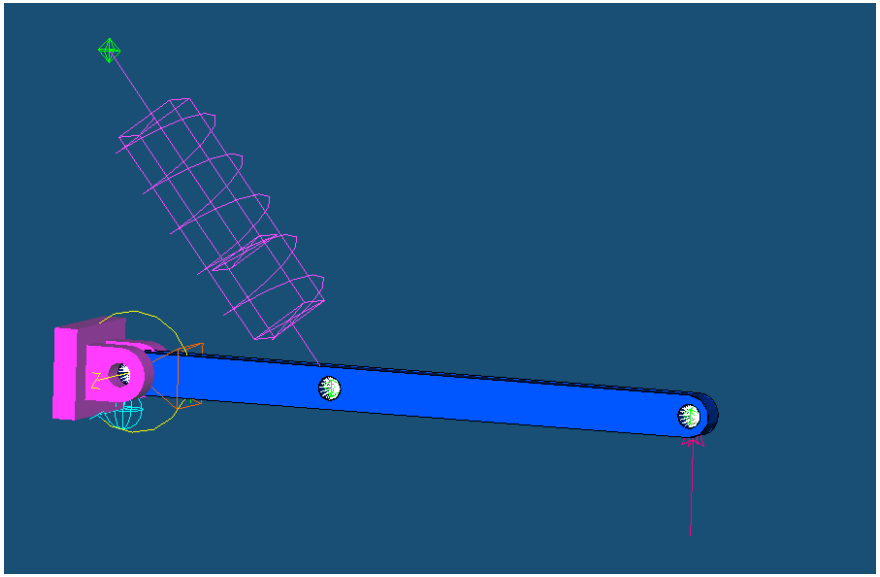


Adding A Spring and Damper



1. Select the spring icon on the toolbar
2. Select the triad in the centre hole of the beam. Done
3. Select the triad placed on the workplane. Done

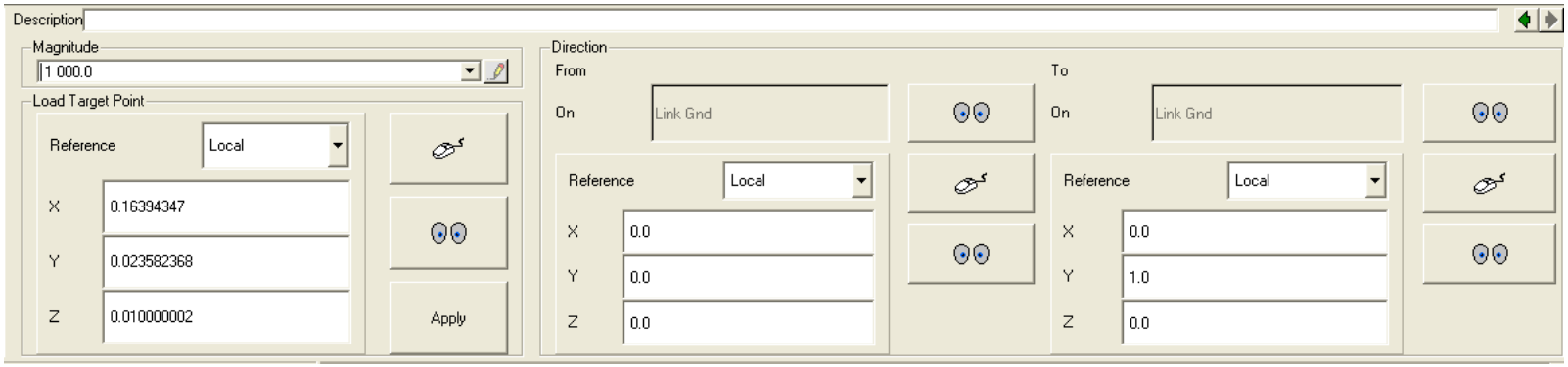
Repeat the same exercise for the damper. The model should then look like the figure below.



(Note: If spring and damper appears large change the size in general appearance to 0.05  , You can also hide the work plane in this manager)

Applying Force

1. Click the force button in toolbar and apply to hole furthers away from the bracket. Done 
2. Change the force direction and magnitude to match the following



3. Select the spring and set spring constant magnitude to 1e5
4. Select the damper and set the damper constant magnitude to 1e4

Result Setup, Strain Coat and Stress

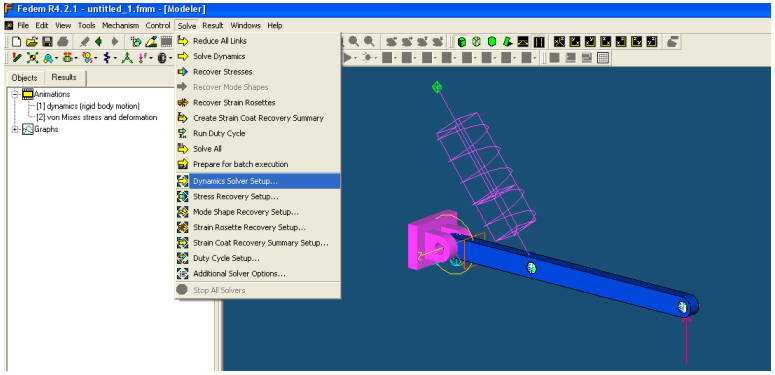
In order to calculate and view the stress in the beam a strain coat has to be generated on the part.

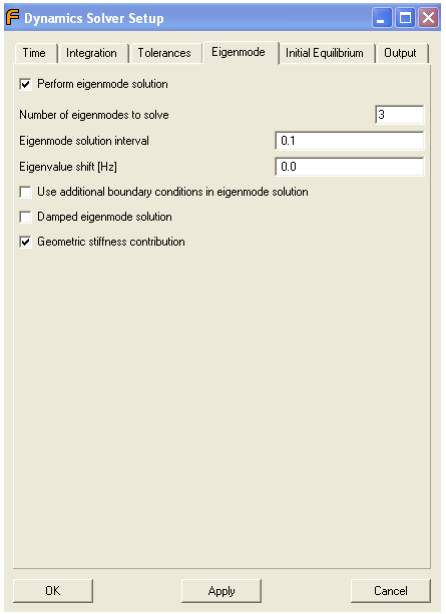
1. Right click on the Beam link in the Model Manager (objects)
2. Select “Generate Stain Coat”

Result Setup, Eigen modes

The purpose of calculating the Eigen modes is to determine the frequencies the dynamic structure is resonating at. IN this exercise only the first three modes are investigated, however, the sae procedure as presented below can e used to find all necessary modes.

1. Go to the “Dynamic Solver Setup” in the Menues toolbar





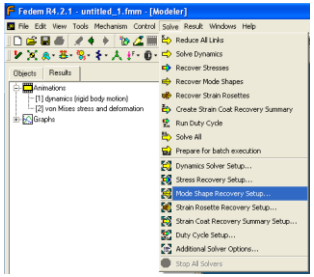
2. In the Solver setup, under the Eigenmode tab:

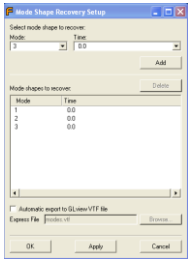
   Tick of the hatch for “Perform eigenmode solution.

   Enter the value 3, in “numbers of eigenmodes to solve

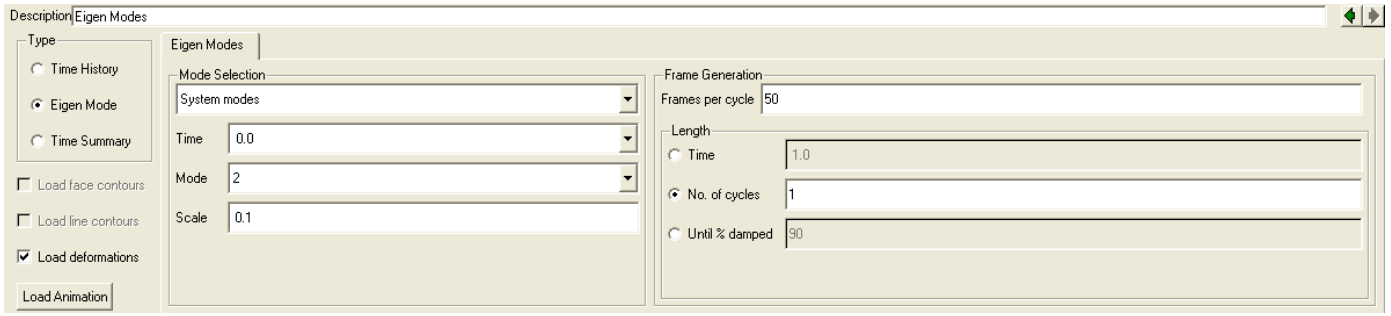
Result Setup, Eigen modes

3. Now enter “Mode shape recovery setup” under “Solve” in Menues toolbar.
4. Add mode 1, 2, 3 at time 0,0 to “mode shapes to recover:” OK



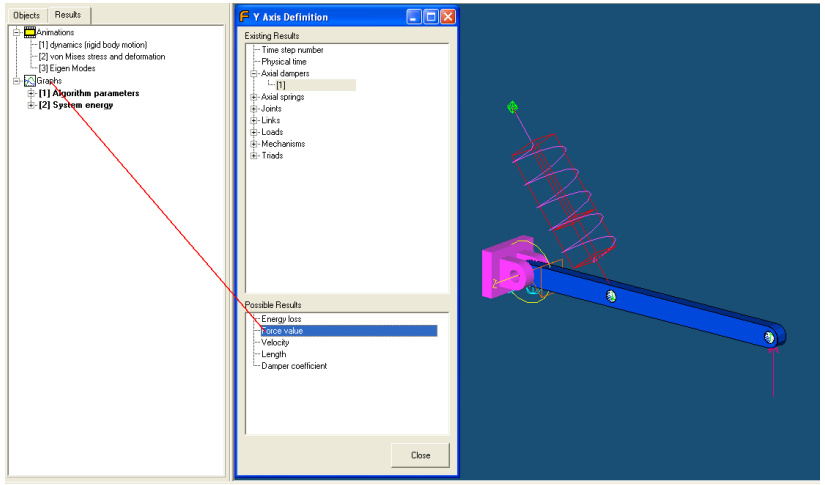


5. Go to the Results tab in the model Model Manager and right click on animations.
6. Name the new animation “Eigen Modes” in the Property Editor panel.
7. Set Type to: Eigen Modes, Tick off Load deformations and set the scale to 0.1



Result Setup, Graphs

1. Go to the Result tab in the Model Manager and right click on Graphs
2. Click on “Result Selector”
3. In the Y Axis Definition window, select Axial Dampers, [1]. From “Possible Results” window below drag and drop “Force Value” to the Graph branch in the result tab.



4. Repeat this exercise for Axial Springs (and all other desirable results)

Now click “Solve All” located on the right hand side of the screen

This will solve everything that is set up in the previous steps.
By using the solve buttons above the “Solve All” button will solve the indevidual parts as indicated when right clicking (and holding) on The button.

Results

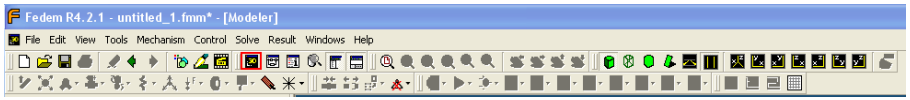
It is now possible to view the results by opening the Result tab in the Model Manager panel.

To view the animations, select the desired animation and click “load animation” in the Property Editor panel (can also be done by left click on the animation and promptly performer the same selection)

To view the graphs left click on the selected graph and select “Show Graph” Note that the curves can be drag dropped into the same graph branch in order to display them in the same coordinatesystem

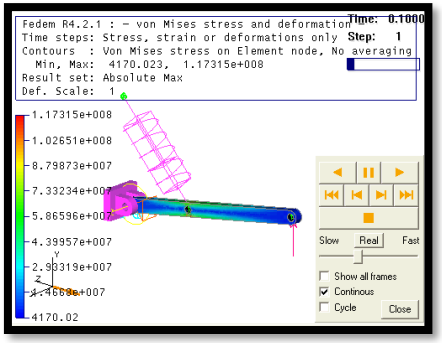
Animations

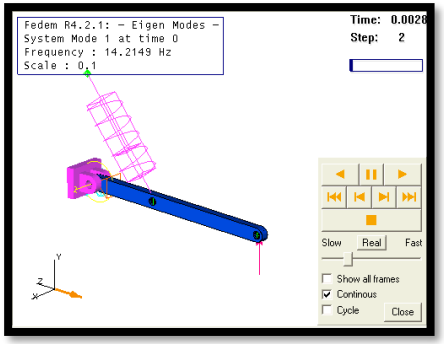
In order to view the animations the Modeller window has to be open. The Modeller can be opened by selecting the button as indicated below.

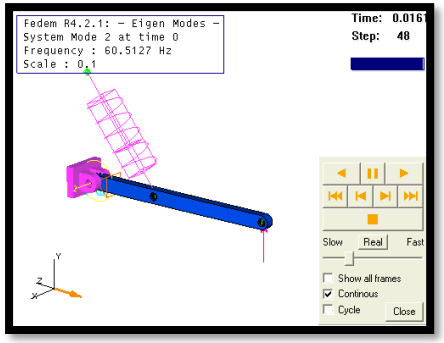


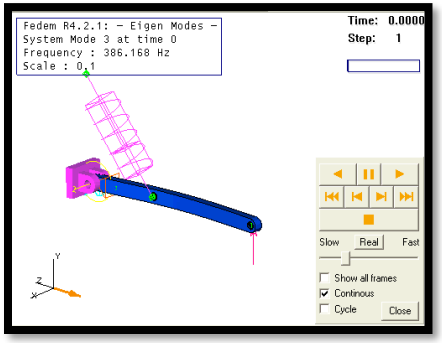
The animations setup can be modified in the Property Manager panel , however, when a change is performed the animation has to be loaded again to view the changes.

The von Mises Stress and Deformation animation shows the stress in the beam. The colourbar and the properties of the plot can be changed with the Animation Controls located in in the Menus toolbar 







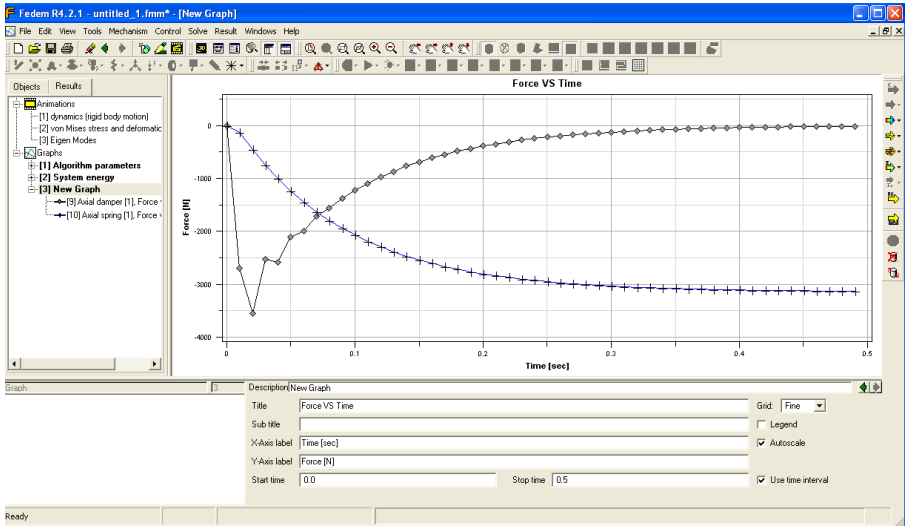


Graphs

The graphs is view in a seperat window that apears when “Show Graph” is selected (right click on desired graph)

The property (axis info, scaling, name, visual attributes etc) of the graphs can be changed in the Property Editor panel.

New curves can be added at any time however, in order to add and view them the current results have to be deleted.



End