Altair

HyperWorks

Altair MotionView 2019 Tutorials

MV-2100: Introduction to Non-Linear Finite Element (NLFE) Analysis in MotionSolve

altairhyperworks.com

MV-2100: Introduction to Non-Linear Finite Element (NLFE) Analysis in MotionSolve

In this tutorial, you will learn the following:

- A brief introduction to Non-Linear Finite Element formulation used in MotionSolve.
- Modeling NLFE bodies in MotionView.

Introduction

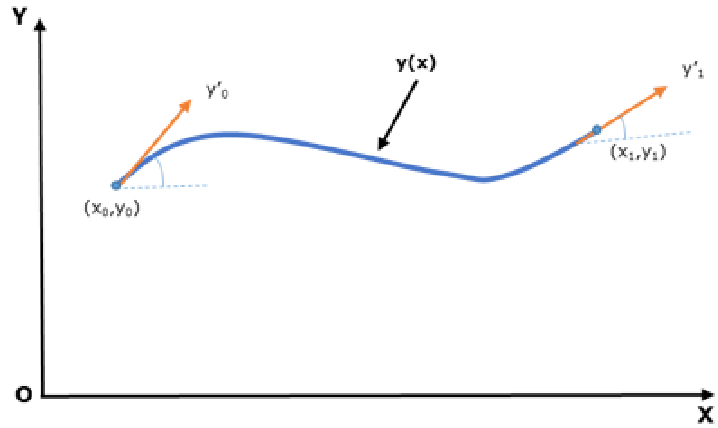
Starting in version 14.0, MotionSolve introduced a new form of flexible body referred as Non Linear Finite Elements (NLFE). Unlike the conventional flexible body, where the body is represented in a modal form, NLFE is modeled in a direct Finite Element form.

NLFE Formulation

MotionSolve uses Absolute Nodal Coordinate Formulation (ANCF) to model large displacement and large deformation (NLFE) bodies. The absolute nodal coordinate formulation[1] is developed based on finite element formulation and is designed for large deformation multibody analysis. In the absolute nodal coordinate formulation, slopes and displacements are used as the nodal coordinates instead of infinitesimal or finite rotations.

ANCF uses the shape function matrix, together with the nodal coordinates to describe arbitrary rigid body motion. For these reasons, the absolute nodal coordinate formulation leads to a constant mass matrix in two- and three-dimensional cases. The constant mass matrix simplifies the nonlinear equations of motion and, consequently, accelerates the time integration of the nonlinear equations of motion.

Below is a standard beam model[2] described in 2D plane using ANCF.



Euler–Bernoulli beam element

Displacement field, $\mathbf{y(x) = s_1(x)y_0 + s_2(x)y'_0 + s_3(x)y_1 + s_4(x)y'_1}$

Where,

$\mathbf{s_1}$ to $\mathbf{s_4}$ are shape functions of beam.

$\mathbf{x_0}$, $\mathbf{y_0}$, $\mathbf{x_1}$, and $\mathbf{y_1}$ are nodal coordinates of 2 grids.

$\mathbf{y'_0}$ and $\mathbf{y'_1}$ are slope coordinates (gradients) at 2 grids.

The absolute nodal coordinate formulation has been applied to a wide variety of challenging nonlinear dynamics problems that include belt drives, rotor blades, elastic cables, leaf springs, and tires.

Non-linear finite element capabilities of MotionSolve

- All type of linear and non-linear elasticity is supported i.e. Isotropic, Orthotropic, Anisotropic and Hyper elasticity.
- Geometric stiffening induced due to elastic forces.

Modeling non-linear behavior above the elasticity limit (like plastic deformation, strain hardening, fracture etc.) is not supported. In the current version, MotionView supports modeling of 1D line elements only (Beam and Cable elements). Beam elements can have 18 types of cross-sections and the dimensions of the cross-section can be changed linearly in the axial direction. Beam elements can resist axial, shear, torsion and bending loads. The Cable element maintains a constant cross-section, thus it can resist only axial and bending loads.

This tutorial has two exercises:

1. Cantilever beam bending.
2. Uniaxial tension of rubber.

Exercise 1

In this exercise, you will model a 1m long cantilever beam with a cross-section dimension of 110mmx14mm to perform a bending test and compare it with an analytical solution.

Copy the `centerline.csv` file, located in the `mbd_modeling\nlfe\intro` folder, to your `<working directory>`.

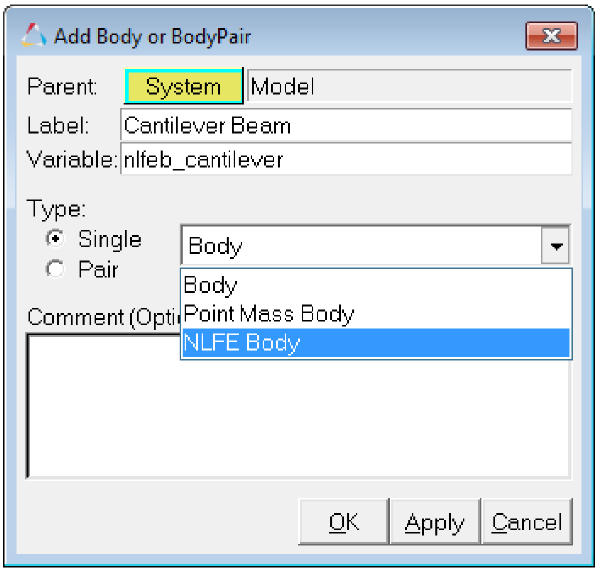
Cantilever beam under end load condition

Step 1: Modeling a Beam with Linear Elastic Material.

1. Start a new MotionView session.
2. Right-click on the ***Body*** icon in the **Model-Reference** toolbar.

   The **Add Body or BodyPair** dialog is displayed.
3. Specify the label as `Cantilever Beam`.
4. Specify the variable name as `nlfeb_cantilever`.

5. Select ***NLFE Body*** from the drop-down menu.



6. Click ***OK*** to close the dialog.

   The NLFE Body is displayed with the **Properties** tab active.

   The following table lists the various tabs available in the NLFE Body panel:

Tab name	Sets NLFEBody
Properties	Type (Beam/Cable), Cross-section, and material properties
Connectivity	Center line data or body profile
Orientation	Start and End orientations
Mass Properties	Displayed for information only
Initial conditions	Initial velocities

7. From the **Properties** tab, define the properties as listed below:

   – **Type:** ***Beam***

   – **Cross section:** ***Bar***

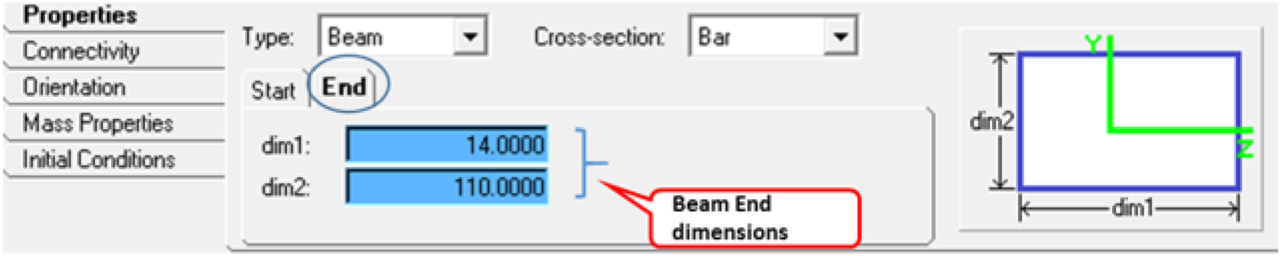
   – **dim1:** ***14.0***

   – **dim2:** ***110.0***

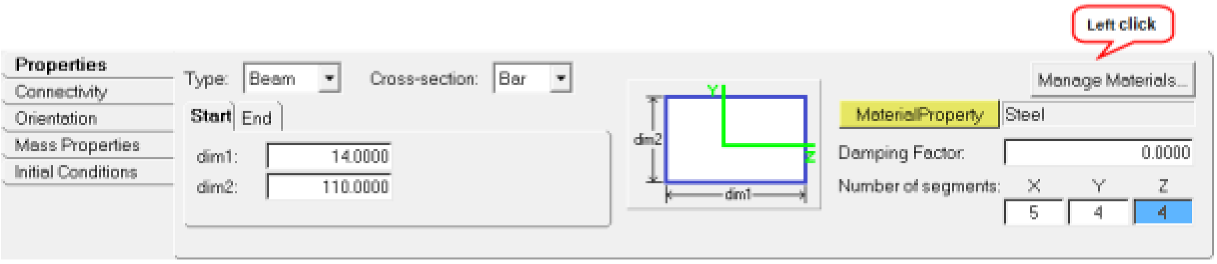
   The panel also displays the image of the cross-section indicating what the different dimensions (**dim1** and **dim2**) refer to.

   Note The **Properties** tab has two sub-tabs called **Start** and **End**. These sub-tabs are used to set the dimensions at two different ends of the beam. By default, the **End** dimensions are parametrically equated to the **Start** dimensions. If a different set of dimensions are provided at the start and end, the cross section varies linearly along the length of the beam.

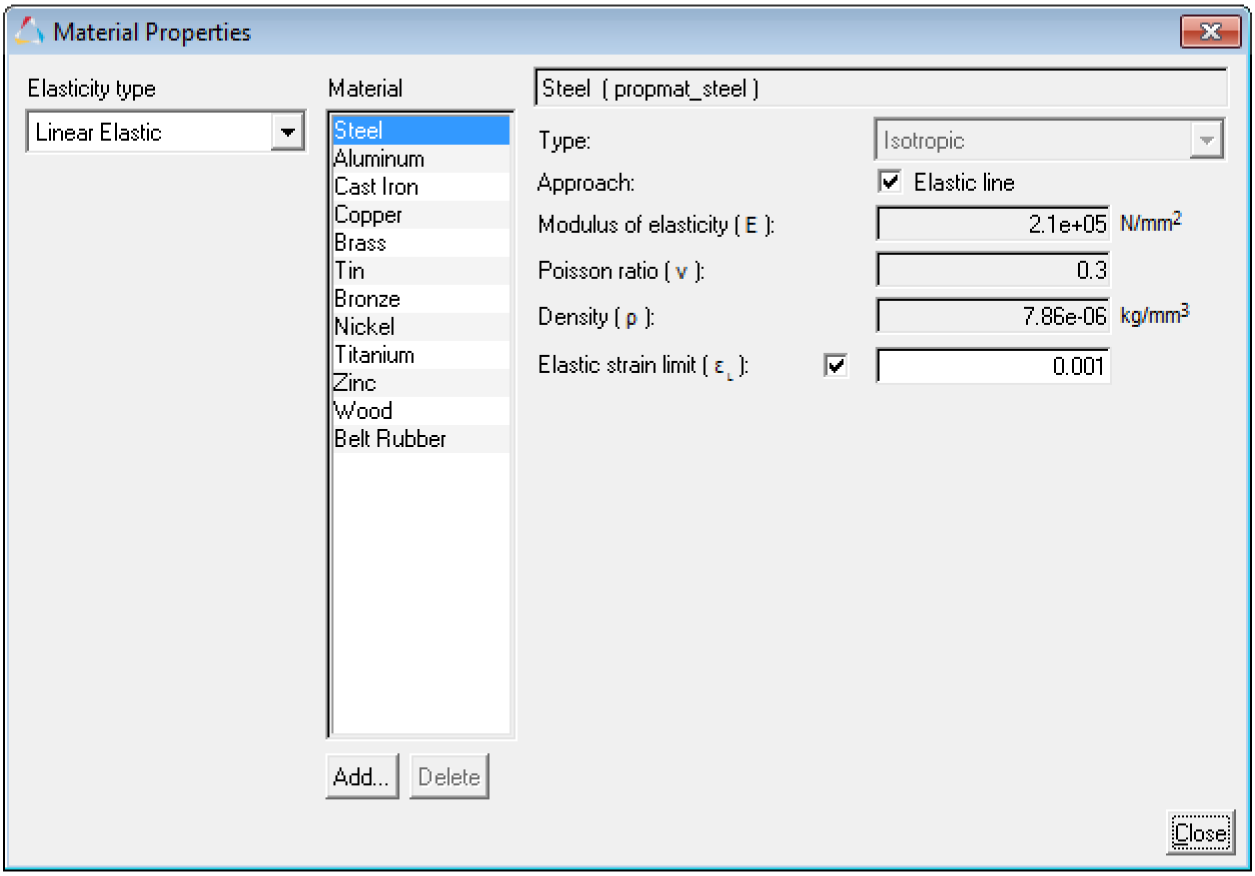
8. Click on the ***End*** sub-tab and review **dim1** and **dim2**.



9. Click on the ***Manage Materials*** button located in the upper right corner of the **Properties** tab.



The **Material Properties** dialog is displayed.



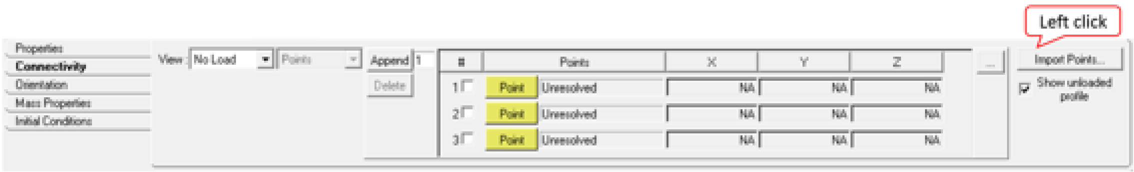
Note The Material Properties dialog can also be accessed from the **Model** menu.

MotionView provides a list of commonly used Linear Elastic, as well as Hyper Elastic material by default.

10. Select ***Steel*** from the **Material** list.
11. Review the property values and notice that the ***Elastic line*** check box for **Approach** is selected.
12. Click ***Close*** to close the dialog.

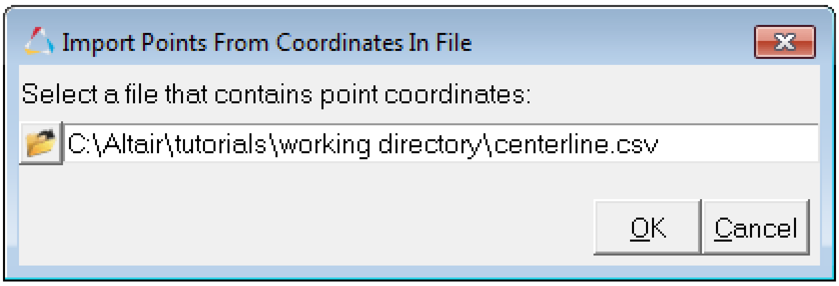
    The default standard materials provided are defined with the **Elastic Line** option checked on. Materials without the Elastic Line are solved using the continuous mechanics approach, where in the cross-section deformation is taken into consideration. The Elastic Line approach ignores cross-section deformation effects, which gives results closer to an analytical solution.

13. From the **Connectivity** tab, define the beam centerline by importing point data from a CSV file. Click on the ***Import Points*** button located in upper right corner.



    The **Import Points From Coordinates In File** dialog appears.

14. Browse and select the ***centerline.csv*** file from your working directory.

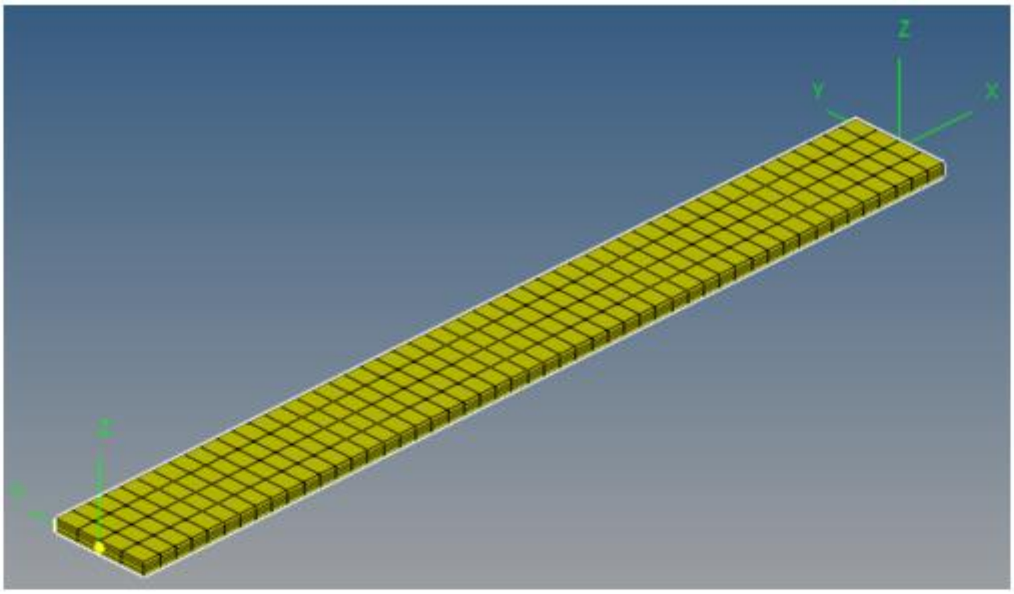


15. Click ***OK*** to import the points.

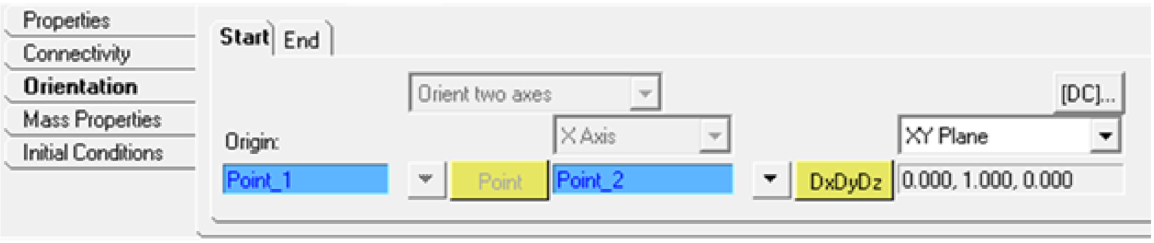
    The `.csv` file must be in the following format: the first column must hold the X-coordinates, the second column the Y-coordinates, and the third column the Z-coordinates. There can also be a header row that begins with a # indicating a commented line.

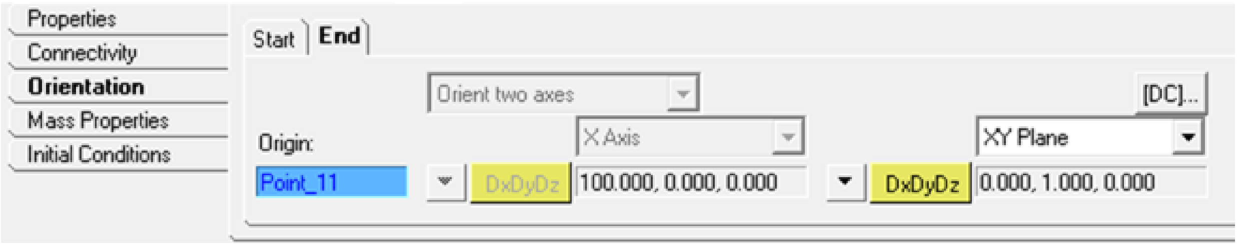
	A	B	C
1	#X	Y	Z
2	1120	-544	989
3	1120	-520	989
4	1130	-510	989
5	1177	-505	981

16. Fit the model (by hitting '**F**' on the the keyboard) to view the NLFE Body that was created.



17. Click on the ***Orientation*** tab to review the **Start** and **End** orientations.



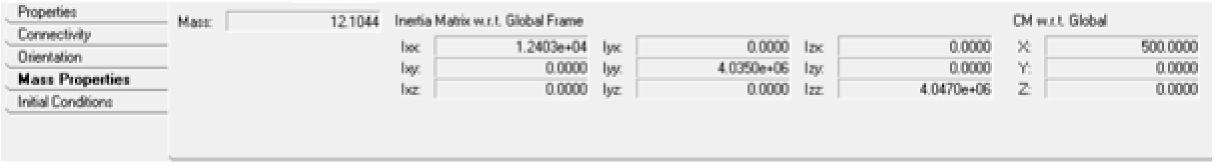


Note The **Orientation** tab is used to set the cross section orientation (YZ plane of the beam). Use the XY Plane or XZ Plane option to position the Y or the Z-axis (the remaining axis will be positioned so that it is orthonormal to the remaining two axes).

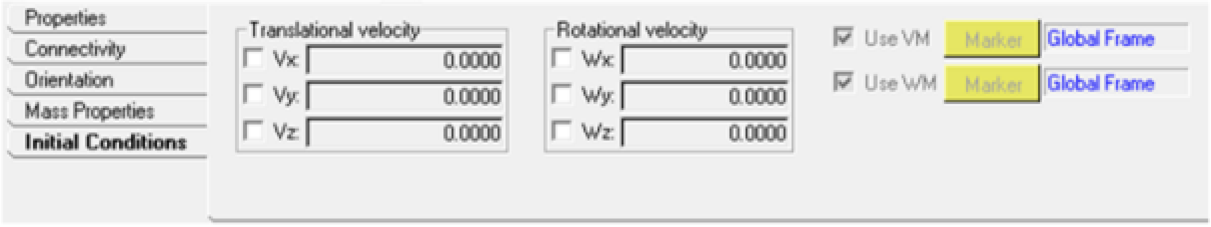
Use the default orientation for this exercise.

Intermediate Beam elements orientation is linearly varied from Start orientation to End orientation. The Orientation option is useful in defining twist along the beam length.

18. Click on **Mass Properties** tab to review the calculated values.



19. Click on **Initial Conditions** tab to review the NLFEBody initial velocities.



Leave initial velocities equal to zero.

Step 2: Adding a constraint and force.

1. Create a Fixed joint at the beam origin point (`Point_1`) as specified in the table below:

S. No	Label	Variable name	Type	Body 1	Body 2	Origin(s)	Orientation Method	Reference 1	Reference 2
1	Fix Joint	j_fix	Fixed Joint	Cantilever Beam	Ground Body	Point_1			

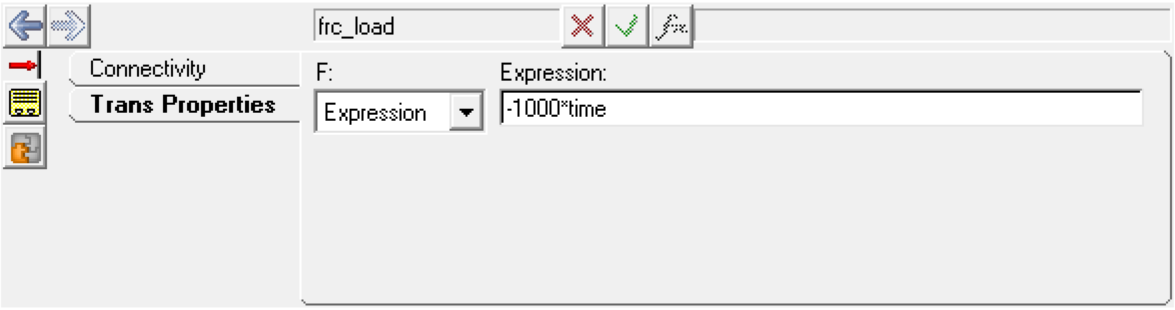
Note Each grid on an NLFE body has 12 DOFs: 3 translational, 3 rotational, and 6 related to the length and angle between the gradient vectors. Using a fixed joint constrains the positions of the grid and the rigid body rotations. However, the gradients at the grid are free. This means that the cross-section at the fixed joint can twist about the grid and also deform based on Poisson's ratio. To arrest these DOFs, an NLFE element called "CONN0" can be used.

There is no graphical user interface support for creating this constraint. By default, MotionView creates a CONN0 element at all of those grids of the NLFE body through which it is attached to a constraint/force entity.

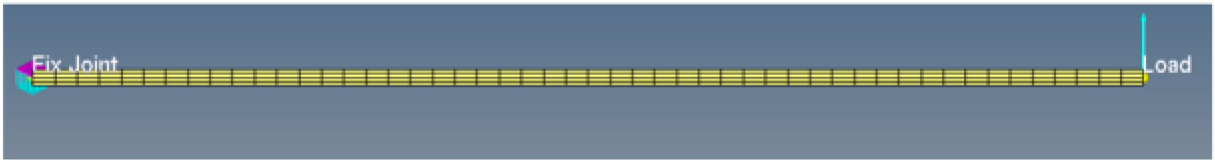
2. Create a load at the cantilever beam end point (`Point_11`) as specified in the table below:

S. No	Label	Variable name	Force	Properties	Action force on	Apply force at	Ref. Marker
1	Load	frc_load	Action only	Scalar Force along Z axis of Ref Frame	Cantilever Beam	Point_11	Global Frame

3. From the **Trans Properties** tab, specify the expression for force as ` -1000*time`.

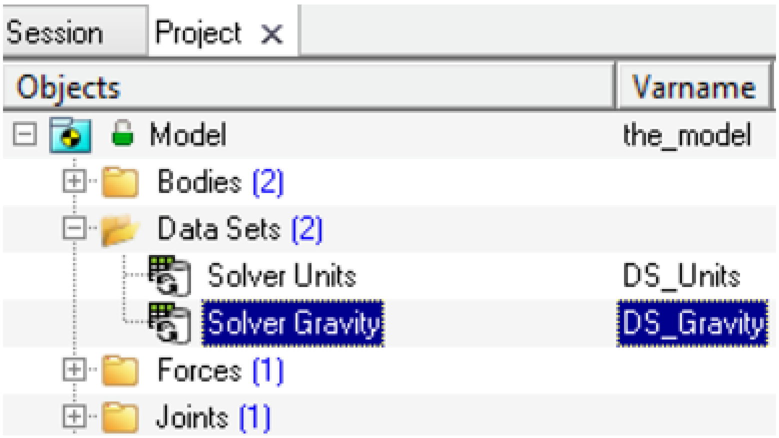


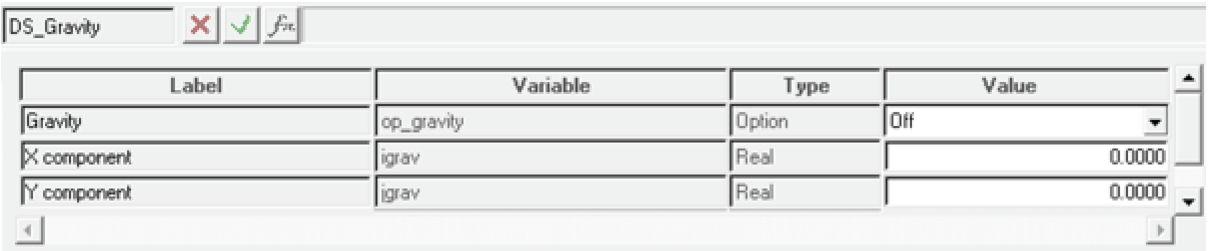
Note Negative value is specified to apply load along negative Z-axis direction.



Cantilever beam with end load

4. Turn off gravity to eliminate deflection due to beam self-weight.





Step 3: Creating outputs to measure cantilever beam end deflection.

Cantilever beam end deflection from linear-elasticity theory.

Deflection for load applied at end $= \frac{Pl^3}{3EI}$

Where,

P = *Load (N)*

l = *Beam length = 1000mm*

E = *Youngs Modulus = 2.1e+05 N/mm²*

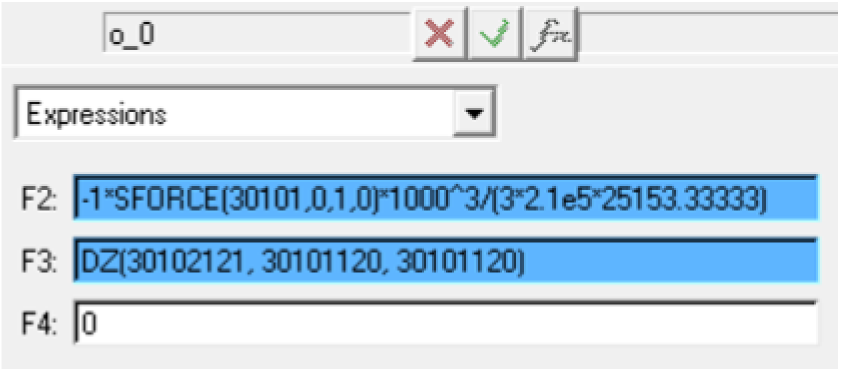
I = *Second Moment of Area* $= \frac{bh^3}{12} = 114 * \frac{14^3}{12} = 25153.33mm^4$

1. Add an **Output** of the **Type *Expressions*** with the following:

   – For **Label**, enter `Deflection - Analytical (F2), NLFE(F3).`

   – For **F2**, enter `` `-1*SFORCE({frc_load.idstring},0,1,0)*1000^3/(3*2.1e5*25153.33333)`. ``

   – For **F3**, enter `` `{frc_load.DZ}`. ``



2. Click on ***Check Model*** to verify the model.
3. Add an output request **Load** to measure the magnitude of the applied Load.

4. Save your model as `nlfe_cantilever.mdl`.

Step 4: Solving the model and post-processing.

The model is now complete and can be solved in MotionSolve.

1. Invoke the **Run** panel by clicking on the ***Run Solver*** button on the toolbar.
2. Specify MotionSolve file name as `Cantilever_beam.xml`.
3. Select the **Simulation type** as ***Quasi-static***, the **end time** as `1 sec`, and the **Print interval** as `0.01`.
4. Click on the ***Run*** button.
5. After the simulation is completed, click on the ***Animate*** button to view the animation in HyperView.

6. Click the ***Start/Pause Animation*** icon, , on the **Animation** toolbar to start the animation.

7. Click the ***Plot*** button in the MotionView **Run** panel to load the `.abf` file in HyperGraph.

8. Plot Deflection vs Load calculated from linear elasticity theory and NLFE by selecting the data below in HyperGraph.

   Select following for X-axis data:

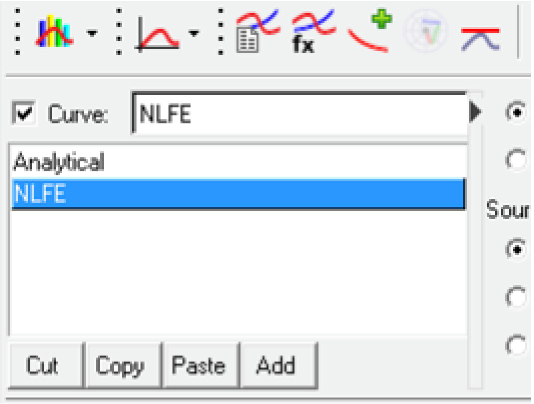
X Type	`Marker Force`
X Request	`REQ/70000001 Load- (on Cantilever Beam)`
X Component	`FZ`

   Select the following Y-axis data:

Y Type	`Expression`
Y Request	`REQ/70000000 Deflection - Analytical (F2), NLFE (F3)`
Y Component	`F2 & F3`

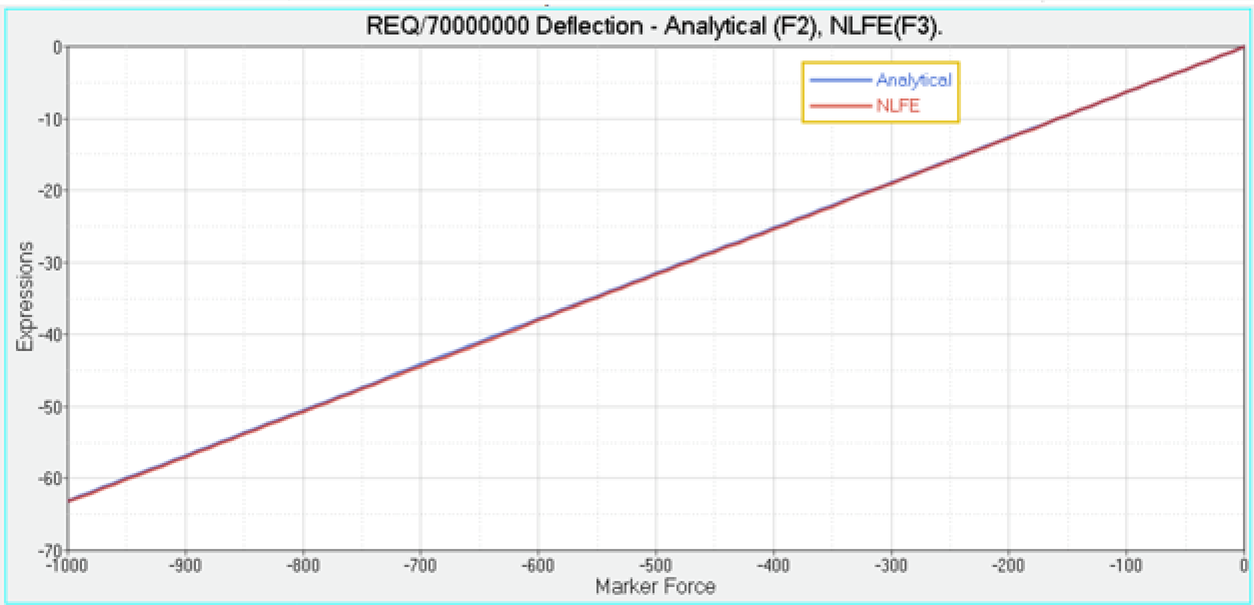
9. Click on the ***Define Curves*** toolbar icon .

10. Rename the two curves as **Analytical** and **NLFE** as shown below:



Define Curves panel

It can be observed from the plot that the NLFE and Analytical curves almost overlap.



Deflection versus load plot

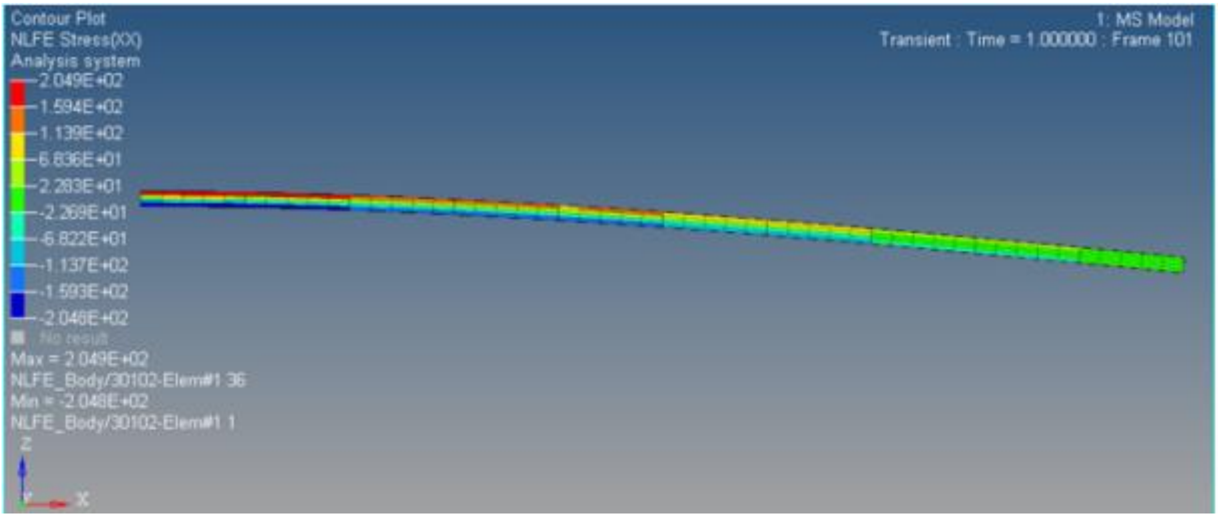
11. Activate the Hyperview animation window.

12. Click on the ***Contour*** button  on the toolbar to activate the **Contour** panel.

13. Under **Result type**, select ***NLFE Stress (t)*** and ***XX***.

14. Click ***Apply*** to view bending stress contours.

    Similarly, you can view Displacement, Strain, etc. for an NLFE body in HyperView. All FE contours and types are available in HyperView for an NLFE body.



Bending stress contour of NLFE beam

15. Save this session as `nlfe_cantilever.mvw`.

Exercise 2: Tensile test of Elastomer

Hyper-elastic materials are large strain materials when compared to metals. In case of hyper-elastic materials the non-linear relation between stress and strain is derived from a strain energy density function. Currently MotionSolve supports three hyper-elastic material models: Neo-Hookean, Mooney–Rivlin, and Yeoh.

In this exercise, a uni-axial tensile test on a rubber strip (2mmx25mmx50mm) will be performed. Hyper-elastic material constants have been sourced from reference[3].

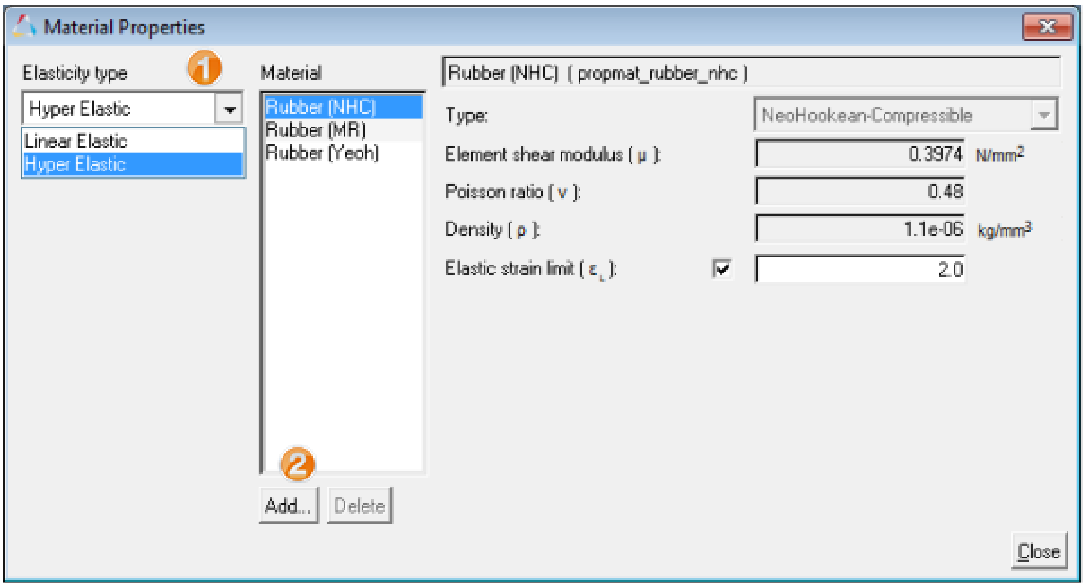
Clear the existing session through ***File>New>Session***.

Step 1: Adding a new material property.

1. From the **Main** menu, select ***Model>Materials***.
2. In **Material Properties** dialog:

   – For **Elasticity type**, select **Hyper Elastic**.

   – Click on the ***Add*** button.



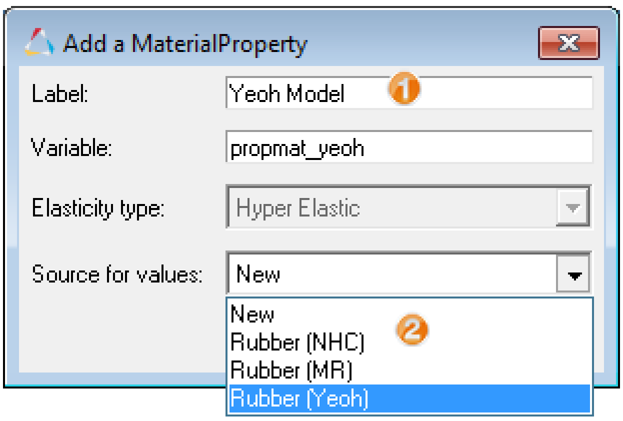
Adding new HyperElastic material

3. From **Add a MaterialProperty** dialog:

   – Specify the **Label** as `Yeoh Model` and **Variable** name as `propmat_yeoh`.

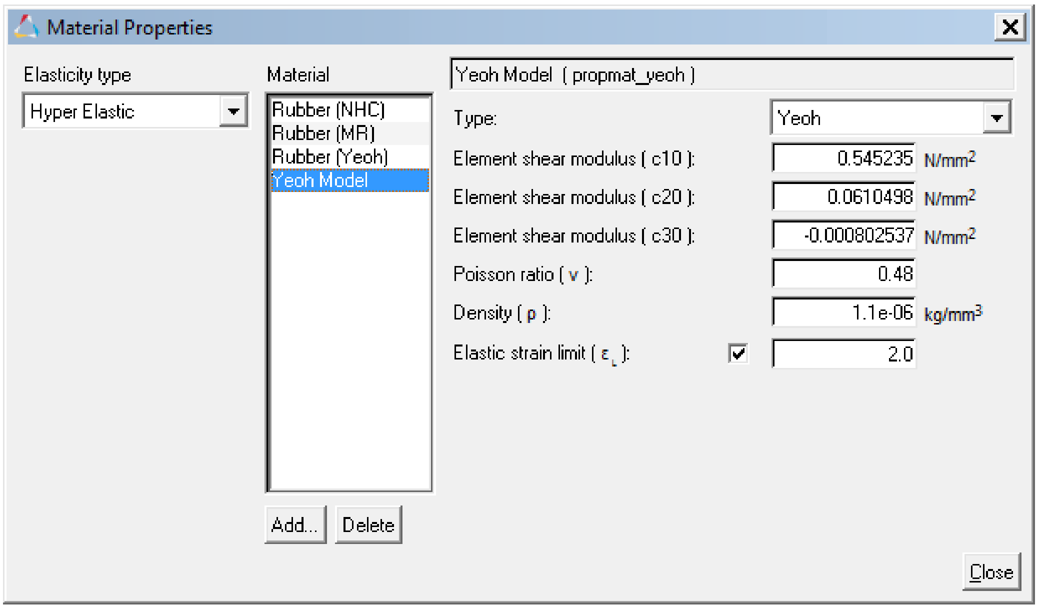
   – For **Source for values**, select ***Rubber (Yeoh)***.

   – Click ***OK*** to add the material property.



Selecting source values for new material

4. Specify the following values for this material:
   - **Element shear modulus (c10):** `0.545235`
   - **Element shear modulus (c20):** `0.0610498`
   - **Element shear modulus (c30):** `-0.000802537`
   - **Poisson ratio (ν):** `0.48`
   - **Density (ρ):** `1.1e-6`
   - **Elastic strain limit (ε_L):** 2.0



Specifying material constant values

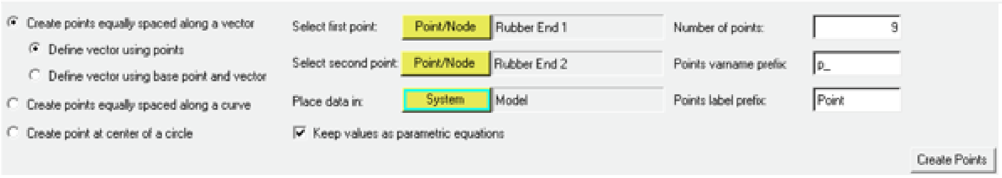
5. Click ***Close***.

Step 2: Modeling the rubber strip.

1. Create two points for the rubber strip length profile with the details show in the table below.

S.No	Label	Variable name	x	y	z
1	Rubber End 1	p_rub_end1	0.0	0.0	0.0
2	Rubber End 2	p_rub_end2	50.0	0.0	0.0

2. Create 9 intermediate points between the above points using the ***Create Points Along a Vector*** macro.



Creating intermediate points using "Create Points along a Vector" macro.

3. Right-click on the ***Body*** icon in the **Model-Reference** toolbar. Add a new NLFE body with the **Label** as `Rubber Strip` and the **Variable** name as `nlfeb_rubber_yeoh`.

   In the **Properties** tab, specify the properties below:

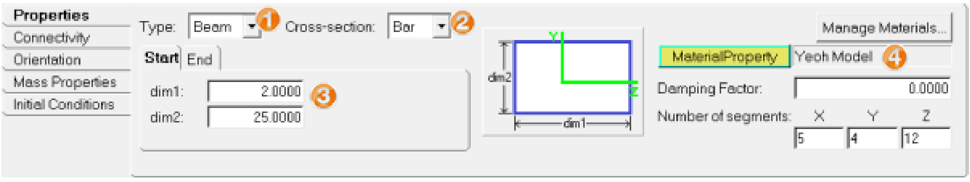
   – **Type** as ***Beam***

   – **Cross-section** as ***Bar***

   – **dim1** = 2

   – **dim2** = 25

   – Select the ***Yeoh Model*** created in previous step as the **MaterialProperty**.

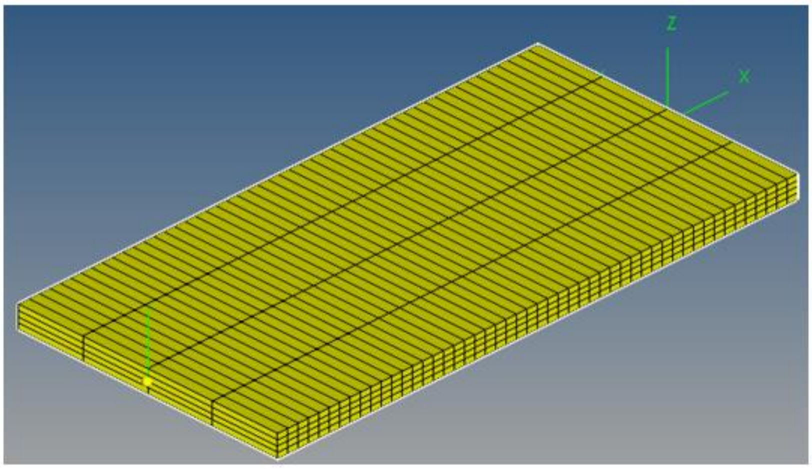


Specifying beam properties

   – Go to **Connectivity** tab; ***Append*** 8 more points to the displayed table.

   – Activate the first **Point** collector and select the point ***Rubber End 1***.

   – Select the next available intermediate points one by one into other collectors, with the last point being **Rubber End 2**.



Rubber strip model

Step 3: Adding constraints.

1. Create Fix joint at one end and Translation joint at other as specified in below table:

S	Label	Variable name	Type	Body 1	Body 2	Origin(s)	Orientation Method	Reference 1	Reference 2
1	Fix Joint	j_fix	Fixed Joint	Rubber Strip	Ground Body	Rubber End 1			
2	Translation Joint	j_trans	Translational joint	Rubber Strip	Ground Body	Rubber End 2	Alignment axis (Vector)	Global X	

2. Create motion on the Translation joint to apply axial pull.

S.No	Label	Variable name	Define Motion	Joint	Property
1	Axial Motion	mot_axial	On Joint	Translation Joint	Displacement

3. In **Properties** tab, specify the following properties:

– **Define by: *Expression***

– **Expression**: `` `50*time` ``

Motion expression

4. Go to the **Solver Gravity** dataset and change the **Gravity** option to ***Off***.

5. Click the ***Save Model*** icon on the **Standard** toolbar and save your model as `rubber_strip.mdl` in your `<working directory>`.

Step 4: Adding outputs.

Create outputs to measure engineering strain and engineering stress values.

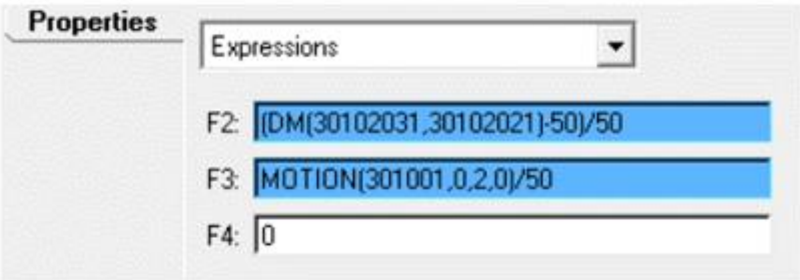
Engineering Strain = $\frac{\Delta l}{l_0}$

Engineering Stress = $\frac{F}{A_0}$

1. Add an **Output** of the **Type *Expressions*** with the **Label** as `Eng strain(F2)`, `Eng Stress (F3)` and the expressions as shown below:

   – **F2:** `` `(DM({j_trans.i.idstring},{j_fix.i.idstring})-50)/50` ``

   – **F3:** `` `MOTION({mot_axial.idstring},{0},{2},{0})/50` ``



Output requests

Note In the above expression **F2**, the solver function `DM()` measures the distance magnitude between two markers; the I marker of the **Translation Joint** and the I marker of the **Fix Joint**. Expression **F3** uses the solver function `MOTION()` which measures the reaction force due to the imposed motion **Axial Motion**.

Step 5: Solving the model and post-processing.

1. To solve the model, go to **Run** panel .
2. Specify the MotionSolve file name as `rubber_yeoh.xml`.
3. Select the **Simulation type** as ***Transient***; the **End time:** `4`, and the **Print interval:** `0.01`.
4. Click ***Run***.

5. As the solver is executing, a warning message similar to the one shown below may be displayed:

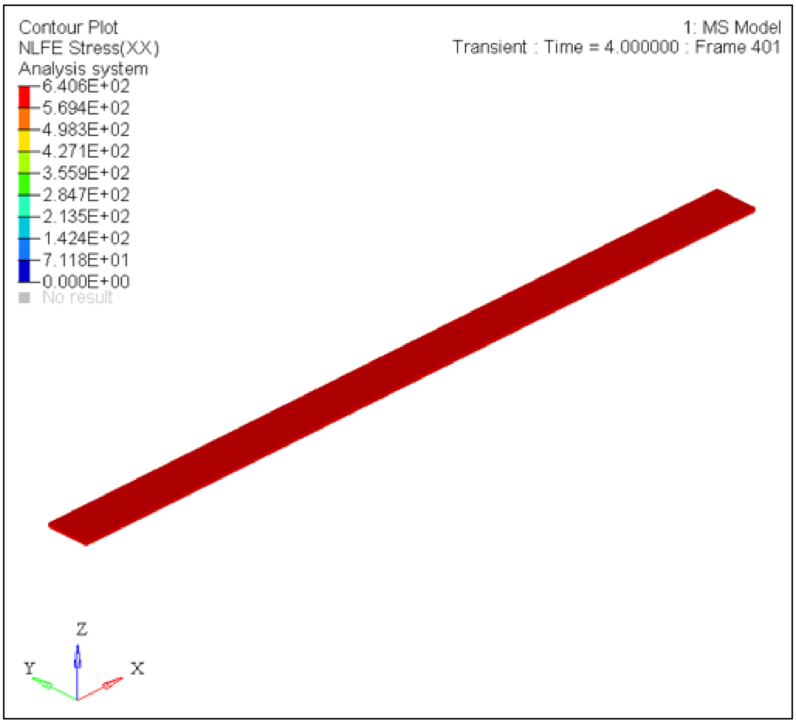
```
WARNING: Maximum vonMises strain exceeded maximum strain (YS) specified
for NLFE element BEAM12 (id=20000000) on Body_Flexible (id=30102) at
time=1.183E+00

          Maximum strain Computed : 2.015E+00

          Maximum strain Specified: 2.000E+00

          Future warning for yield strain violation suppressed.
```

   This message states that the maximum vonMises strain in your NLFE component exceeded what was specified (2.0) at time 1.18s. This message lets you know if your component is deforming more than what you would expect it to, which allows you to inspect your results and make corrections in modeling your system if required.

6. After the simulation is completed, click on ***Animate*** to view the animation in HyperView.

7. Click the ***Start/Pause Animation*** button to play the animation.

8. Go to the **Contour** panel and select ***NLFE Stress (t)***, ***XX***, and click ***Apply***.



Stress contour

9. Return to MotionView **Run** panel and click on ***Plot*** button to load the `.abf` file in HyperGraph.

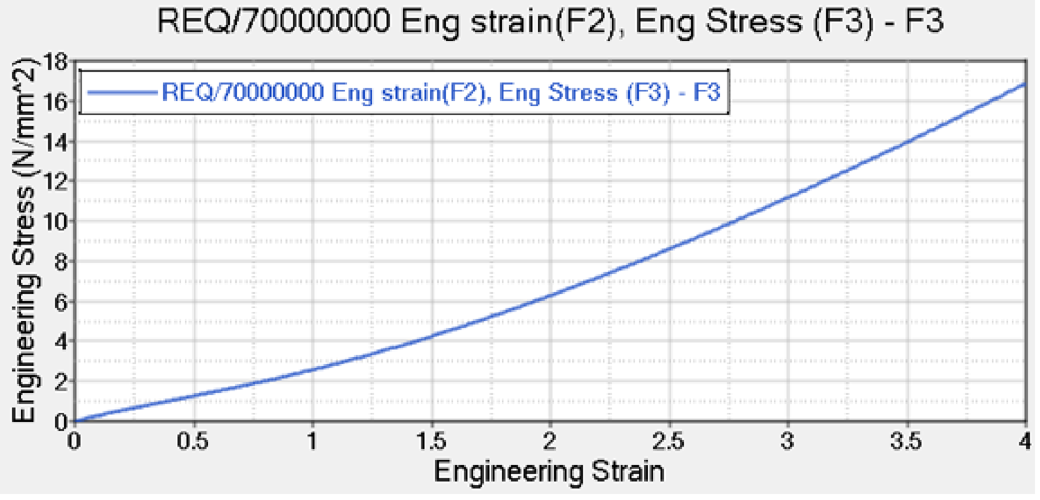
10. Plot Engineering Stress vs Engineering Strain by selecting the data below in HyperGraph.

   – Select following for X-axis data:

X Type	`Expression`
X Request	`REQ/70000000 Eng strain(F2), Eng Stress(F3)`
X Component	`F2`

   – Select the following Y-axis data:

Y Type	`Expression`
Y Request	`REQ/70000000 Eng strain(F2), Eng Stress(F3)`
Y Component	`F3`



Stress versus strain curve

Note The animation shows the true stress due to cross-section deformation.

11. Save the model.

12. Save this session as `hyperelastic.mvw`.

References:

1) **JUSSI T, SOPANEN and AKI M. MIKKOLA**: Description of Elastic Forces in Absolute Nodal Coordinate Formulation. Journal of Nonlinear Dynamics 34: 53–74, 2003.

2) **Oleg Dmitrochenko**:Finite elements using absolute nodal coordinates for large-deformation flexible multibody dynamics. Proceedings of the Third International Conference on Advanced Computational Methods in Engineering (ACOMEN 2005).

3) **Sung Pil Jung, TaeWon Park, Won Sun Chung**: Dynamic analysis of rubber-like material using absolute nodal coordinate formulation based on the non-linear constitutive law. Journal of Nonlinear Dyn (2011) 63: 149–157.