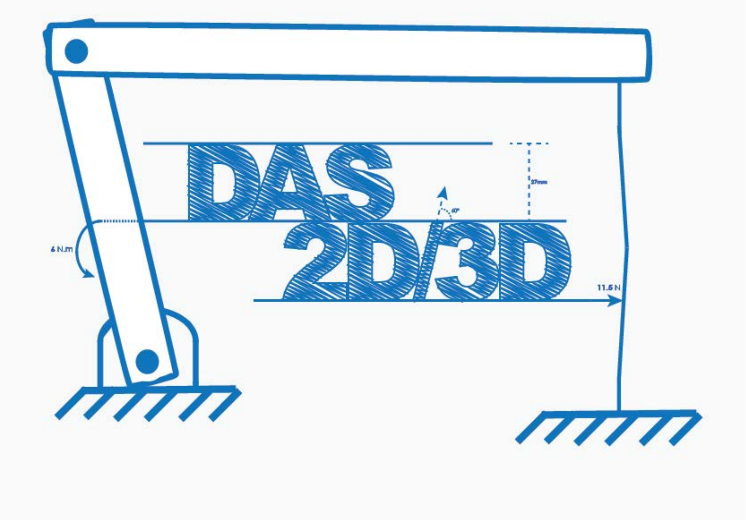
DESIGN, INNOVATION AND SIMULATION LAB.

THE OHIO STATE UNIVERSITY

Design, Analysis and Synthesis (DAS) - 2D: A Design Tool for Compliant Mechanisms



Hai-Jun Su (Associate Professor)
su.298@osu.edu
Department of Mechanical & Aerospace Engineering

Acknowledgement

- Anil O. Turkkan
- Venkat Kalpathy Venkiteswaran
- Prof. Guimin Chen (Xidian University, China)
- National Science Foundation and Air Force Office of Scientific Research

Why a Dedicated CAD Tool?

- Extensive publications including several books on compliant mechanisms since early 90's
  - Pseudo-rigid-body models
  - Beam constraint models (BCM), chained BCM
  - Topological optimization
  - Building blocks
- There are also academic codes that may not be accessible
- Currently design and analysis of compliant mechanisms heavily rely on use of commercial software
  - Finite element solvers, e.g. Abaqus, ANSYS
  - Dynamics simulation software with the function of simulating flexible bodies
    - MSC Software Adams, implement pseudo-rigid-body models
    - WorkingModel 2D: dynamic analysis of planar mechanisms.
- We need a computer-aided-design (CAD) tool for research and education of compliant mechanisms.

What is DAS 2D 2.0?

- **D**esign, **A**nalysis and **S**ynthesis (DAS) 2D is a design tool for compliant mechanisms.
- The software tool is aimed to be:
  - Fully interactive and simple to use
  - Faster than finite element analysis methods
  - Also be employed as an educational tool
- MATLAB is chosen as the programming language to take advantage of built in functions.
- Object oriented programming approach was employed.
  - Different classes can be used in other projects
- The software can be downloaded from:
  - http://compliantanalysis.com
- Currently, there are users in approximately 30 institutions in more than 20 countries.

DAS-2D Overview

- Rich graphical user interface (GUI)
- Kinetostatic analysis of planar compliant mechanisms
  - A general kinetostatic solver based on Matlab's built-in optimizer
  - Force deflection analysis
  - Kinematic driver analysis
- Kinematic analysis of planar rigid-body mechanisms
  - A general kinematic solver
- Implemented various models of flexible bodies like 2D beams
  - The finite segment model
  - Various pseudo-rigid-body models
  - Beam constraint models (BCM), chained BCM
  - Linear beam model
- Design synthesis of compliant mechanisms
  - Mechanical Advantage
  - Flexural Stiffness Synthesis
  - Bistable compliant mechanism synthesis

Basic Steps

1. Download and install software on PC
2. Launch software
3. Select workspace size and unit
4. Roughly sketch a planar mechanism (a graph of nodes)
5. Dimension the geometries (coordinates of nodes, link length, angle etc.) of the mechanism
6. Specify kinematic joints, add linear or torsion springs
7. Pick one or more links to be compliant
8. Select an appropriate model (PRB, BCM, linear beam etc.) for a compliant link.
9. Apply load to any node or link
10. Perform analysis
11. Post-processing: report, plotting

Download and Installation

- System requirements:
  - 1 GB storage space
  - No MATLAB installation is needed.
  - Operation System: Windows.
  - Mac version will be available with the final release
  - Source code will be released with final release of DAS-2D 2.0
  - Final version will be available sometime during September
- Download software from compliantanalysis.com
- Install the software by executing .exe file

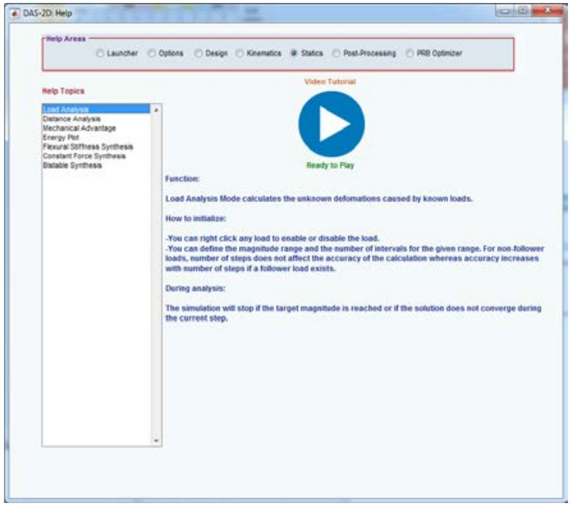
DAS 2D Launcher

Through the launcher you can:

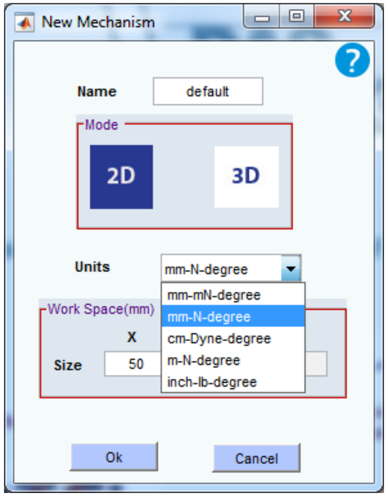
- Create a new mechanism
- Load a previously saved mechanism
- Run the program "Cantilever Beam Analysis"
- Run the program "PRB Optimizer"

DAS 2D Launcher

- The software has a built in help system.
- You can access the help page for any module by clicking the question mark ?.



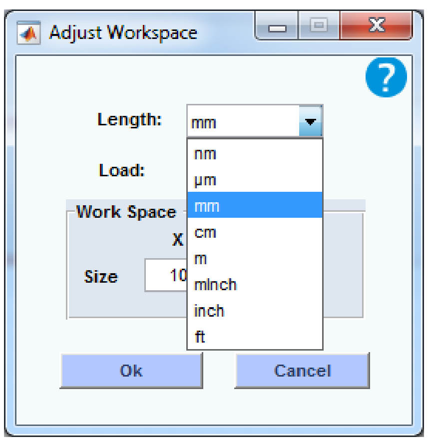
The built in help system



New Mechanism Screen

Before you create a new mechanism, you need to select:

- One of four unit systems
- Workspace size



Options

More unit types are available at:

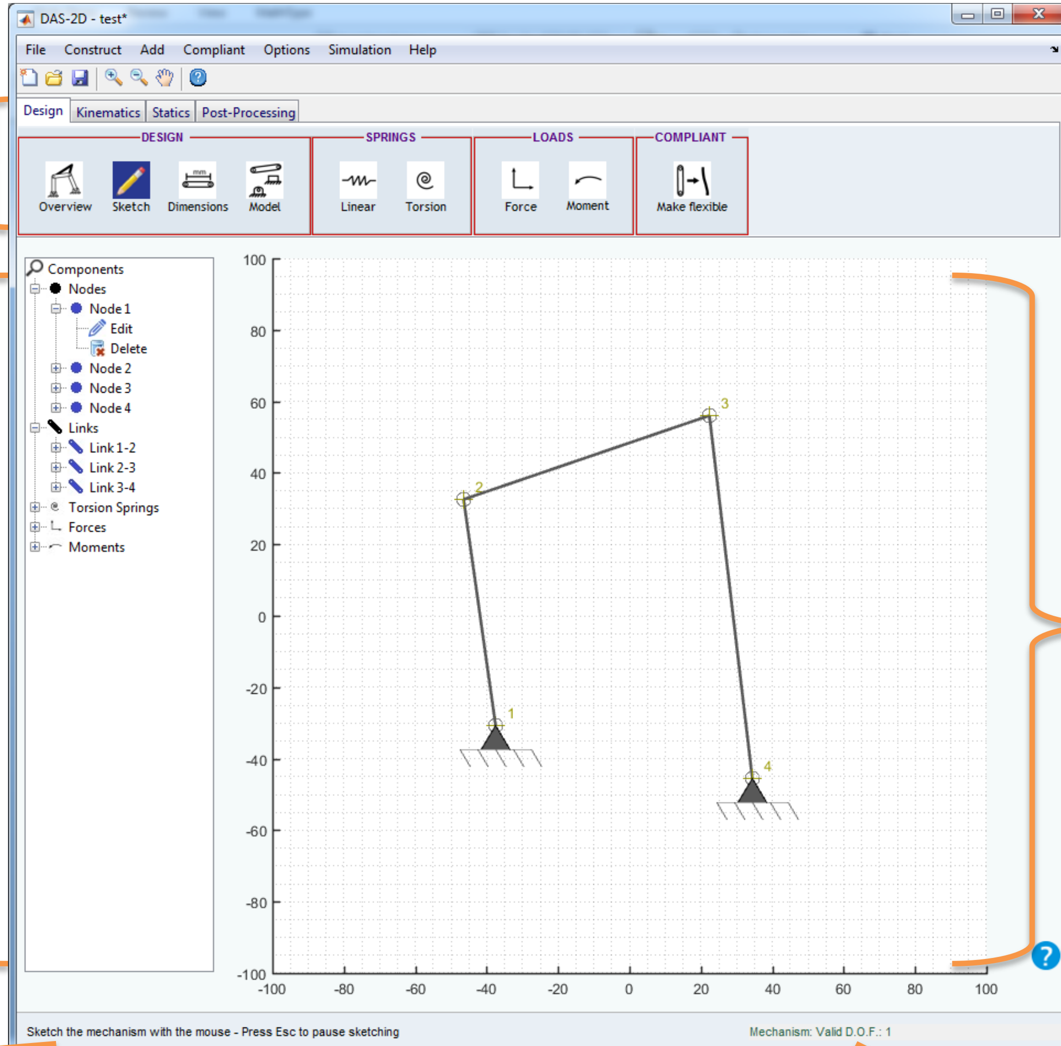
Menu->Options->Workspace

DAS 2D – Main Screen

Different Modules:

- Design
- Kinematics
- Statics
- Post-Processing

Component Tree



Workspace

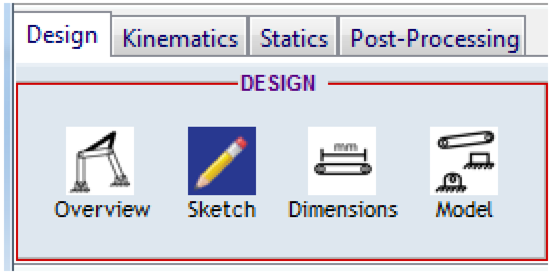
Help System

Current Module Tips

- Mechanism Status
- Total Degrees of Freedom

The Design Module has four tools

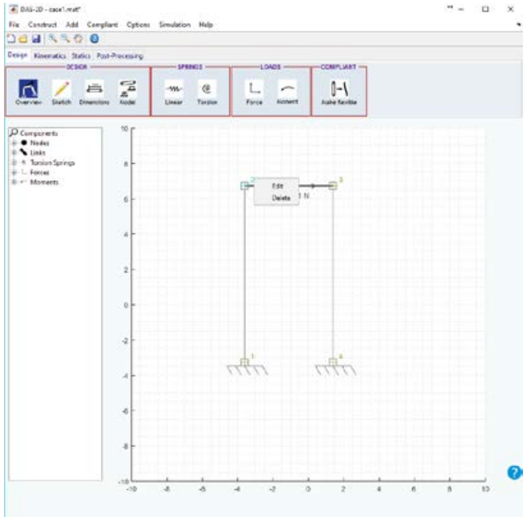
- Overview: add, edit and delete a node or link or joint
- Sketch: sketching schematic view of a mechanism
- Dimension: sizing link length, angle, coordinate etc.
- Model: change joint type etc.



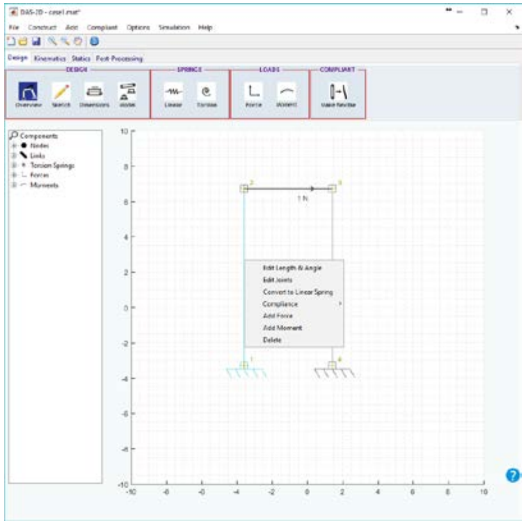
Design Module

The Overview Design Tool

- All components can be edited or new components can be added via this module.
- All components will be drawn in this module.
- Right click a component to edit.
- After sketching the mechanism, this module should be used to finalize the mechanism.
- This module has all the functionality of the other design modules.
- Only torsion springs have to be added using the dedicated module.



Edit a Node



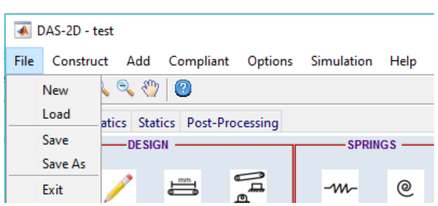
Edit a Link

Component Tree

- All components will be shown in this tree.
- No new component can be added via this tree.
- The individual components can be edited or deleted using the various options below the component names.

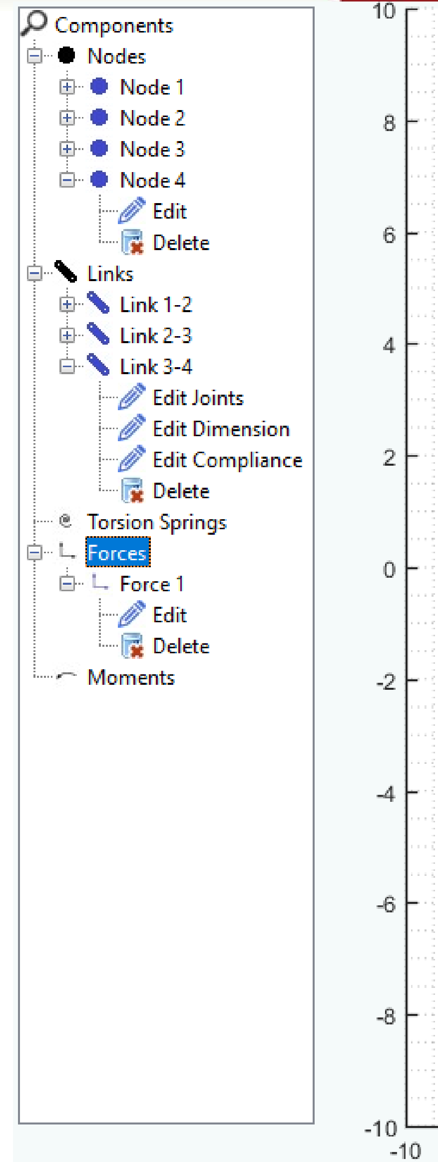
Toolbar and the Menu

- The toolbar and the menu can be used the open, load or save the mechanism.
- Any module can be accessed via the menu.
- The workspace can be adjusted (zoom in-out or pan) using the toolbar.
- The exact workspace size can be set under Options->Workspace



Menu

Toolbar

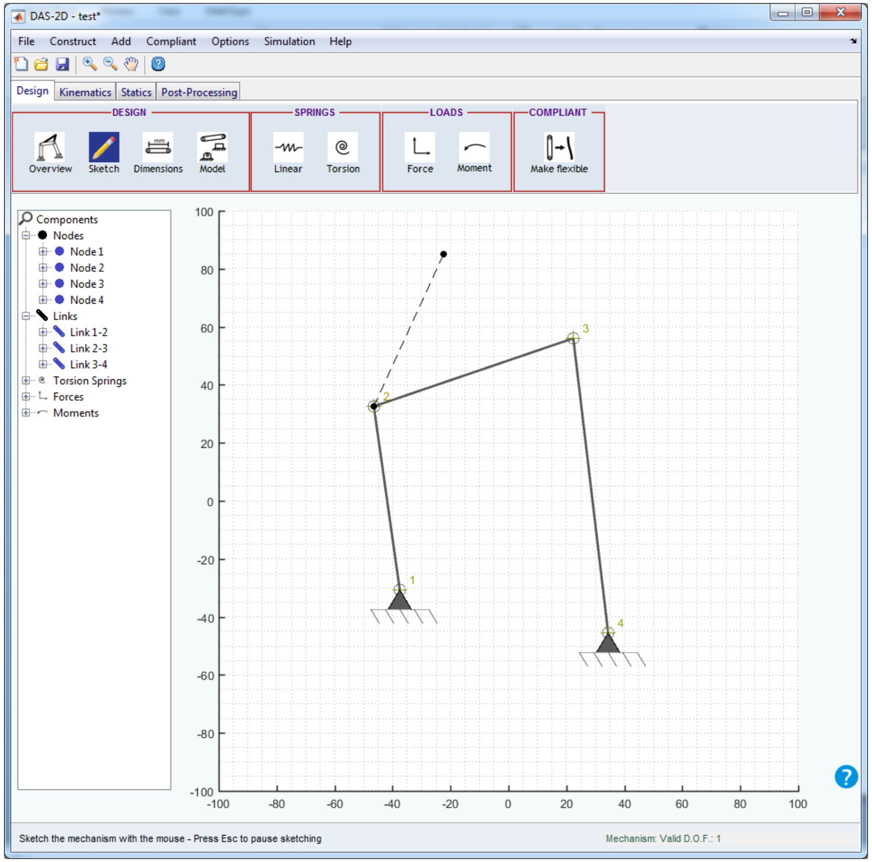


Component Tree

The Sketching Tool

In the sketching tool, you can

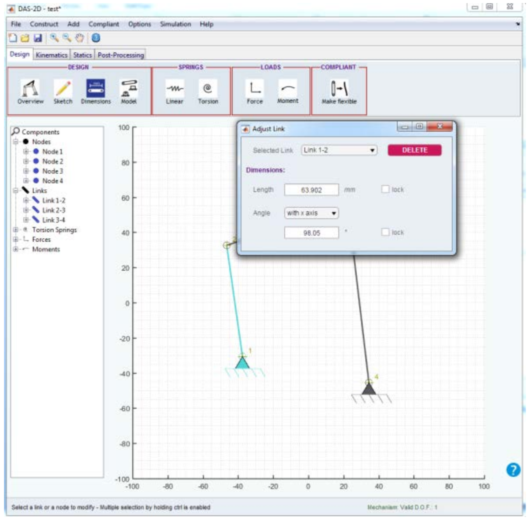
- Draw a rough sketch of the mechanism
- Click workspace to create a new node
- Last two nodes will be joined to create a link
- Click an existing node to lock to that node
- Press Esc to start the procedure from a new node



Sketching Tool

Edit a Link

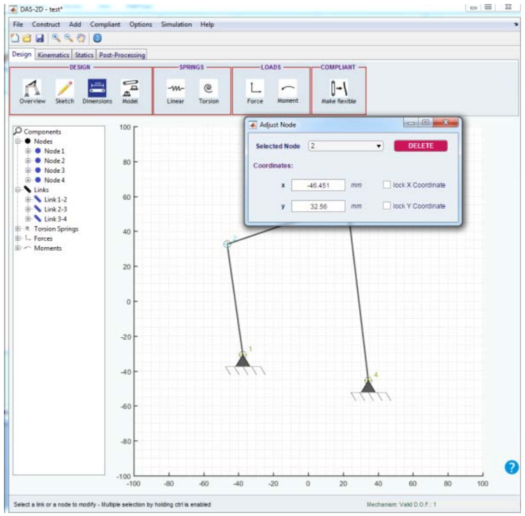
- Click a link to edit length or angle
- Enter a valid length and click lock or hit enter key
- You can specify the angle with the x or y axis
- Current or desired angle can be negative. Make sure to check the link direction



Edit a Link

Edit a Node

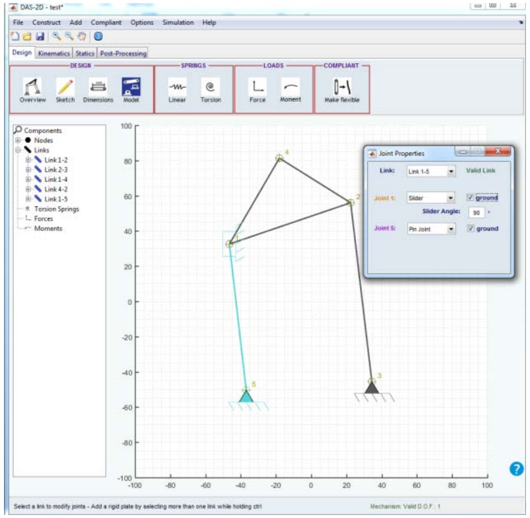
- Click a node to edit x or y coordinates
- Enter a valid coordinate and click lock or hit enter key
- Locked node coordinates will not change if link lengths or angles are altered



Edit a Node

Edit a Link

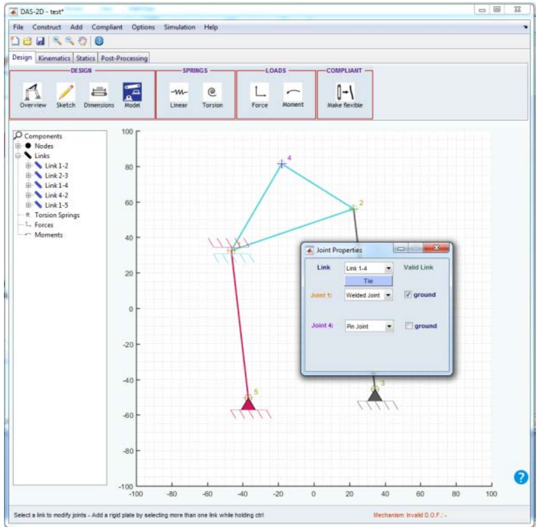
- Click a link to edit joints
- Select joint types for both ends from three available types
- If you select a slider joint, you can enter the slider angle
- After you close the window, if the joint combination is not valid, the link will drawn in red color.



Edit Joints

Add a Rigid Plate

- Ctrl + click more than one links
- Tie button will appear
- Tied links will move together
- You cannot edit joint types of tied links
- Rigid plates should be used for only kinematic analysis (Use welded joint type for kinetostatic analysis)
- If you click a tied link, you can remove it from a rigid plate group



Add a Rigid Plate

Design Module Demonstration

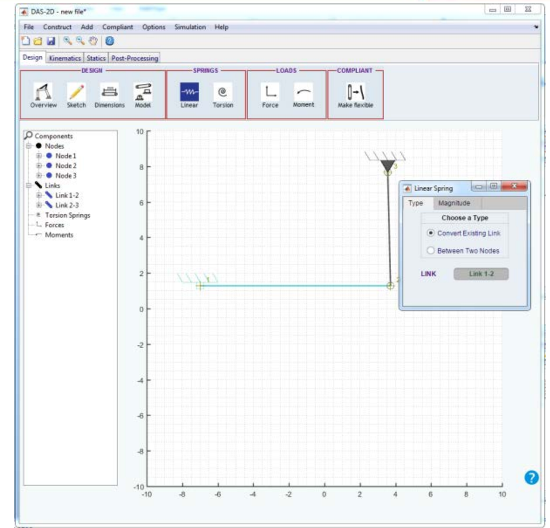
Linear Spring – Convert a Link

- A link can be converted to a linear spring.
- Click a link that you want to convert
- Enter the magnitude

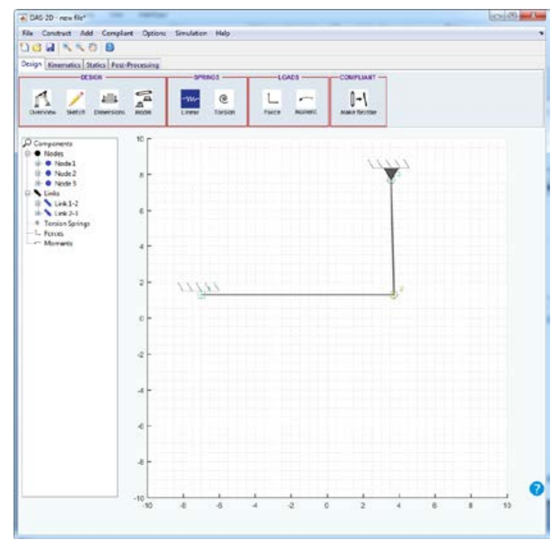
Linear Spring – Add a new Link

- A new link can be added between two nodes.
- Select two nodes
- Enter the magnitude
- A new link will be added and converted to a linear spring.

While at this module, linear springs can be clicked and edited.



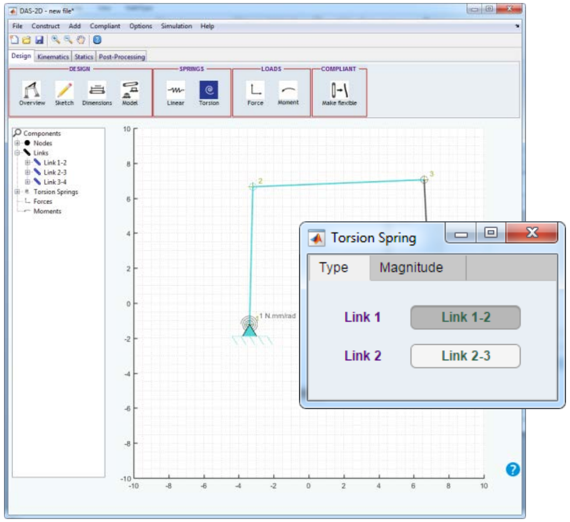
Convert a link



Add a new Link

Add a Torsion Spring

- Select two links to add a torsion spring at the common node.
- Instead of a link, you can select the ground by clicking an empty spot at the workspace.
- Enter the magnitude and hit ok to add a torsion spring.

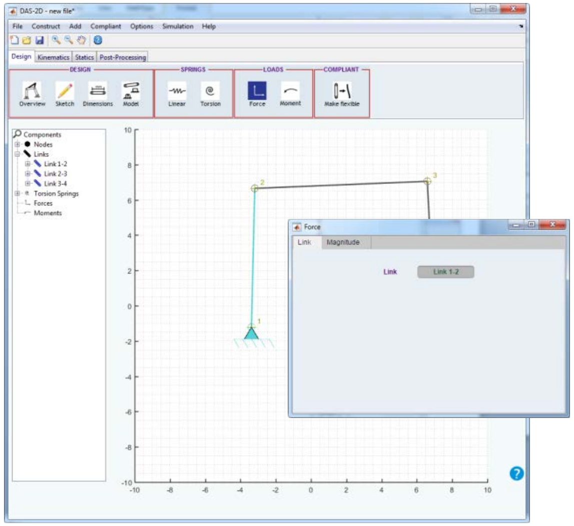


Add a Torsion Spring

In this module, torsion or linear springs can be selected and edited.

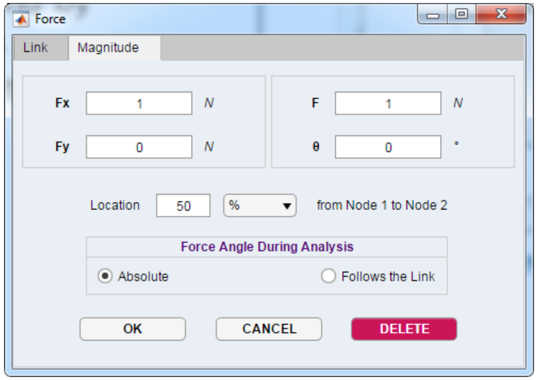
Add a Force

- Select a link that force acts on
- Force angle can be fixed in space or can follow the link
- Force angle is calculated with respect to x axis for non-follower loads
- Force angle is calculated with respect to the link for follower loads



Add a Force – Select a Link

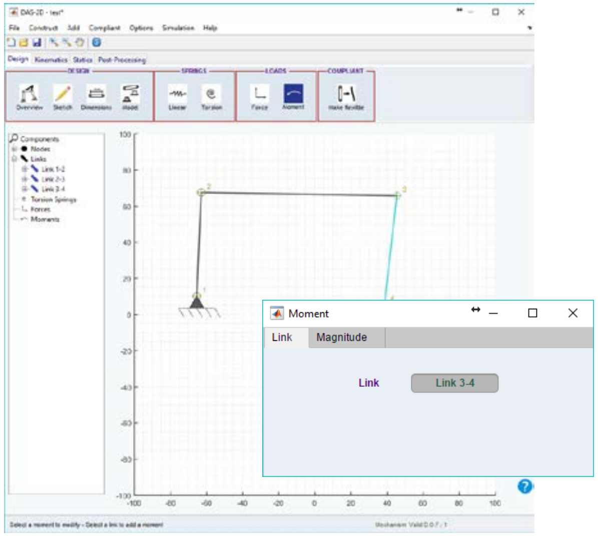
In this module, forces can be clicked and edited.



Add a Force – Properties

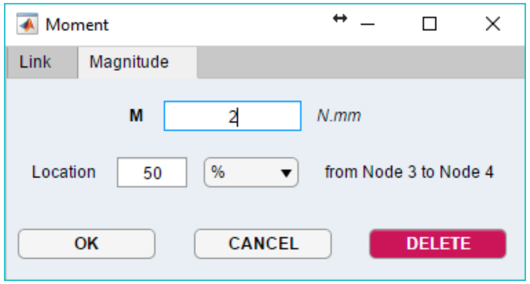
Add a Moment

- Select a link that moment acts on
- Enter the magnitude and hit ok to add a moment



Add a Moment – Select a Link

In this module, moments can be selected and edited.



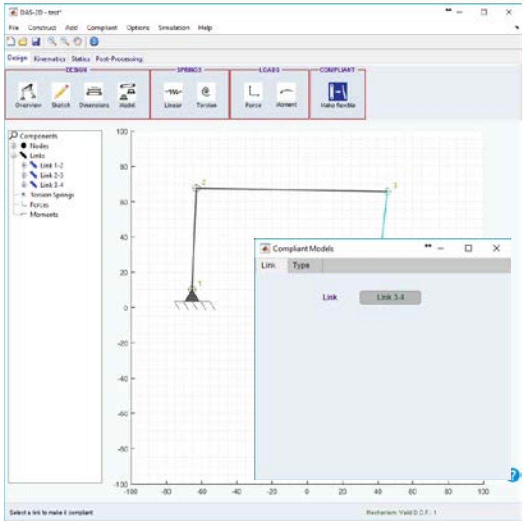
Add a Moment – Properties

Spring and Load Module Demonstration

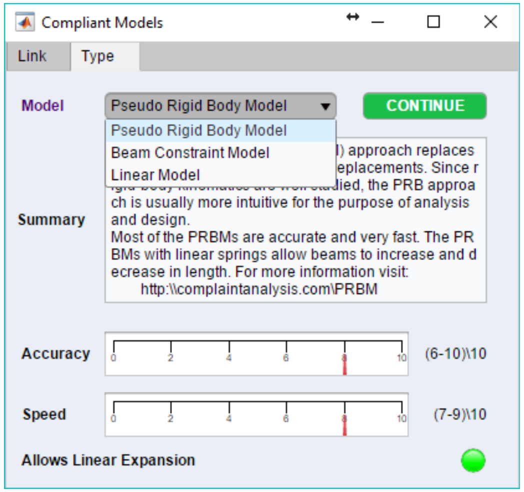
The Compliant Link Module

Make a Link Compliant

- Select a link that will be compliant
- There are three different models you can select
  - Pseudo-Rigid-Body Models
  - Beam Constraint Model
  - Linear Model
- Beam Constraint Model and Linear Model can consist of multiple segments
- To switch between PRB and other models, the link must be converted to a rigid link first.



Make Compliant – Select a Link



Make Compliant – Models

Models Implemented

- The beam equation model
- The finite segment model
- Various pseudo-rigid-body models
- Beam constraint models (BCM), chained BCM
- Linear beam model

More accurate
Large deflection
Slower

Less accurate,
Small deflection
faster

Cantilever Beam Analysis Tool

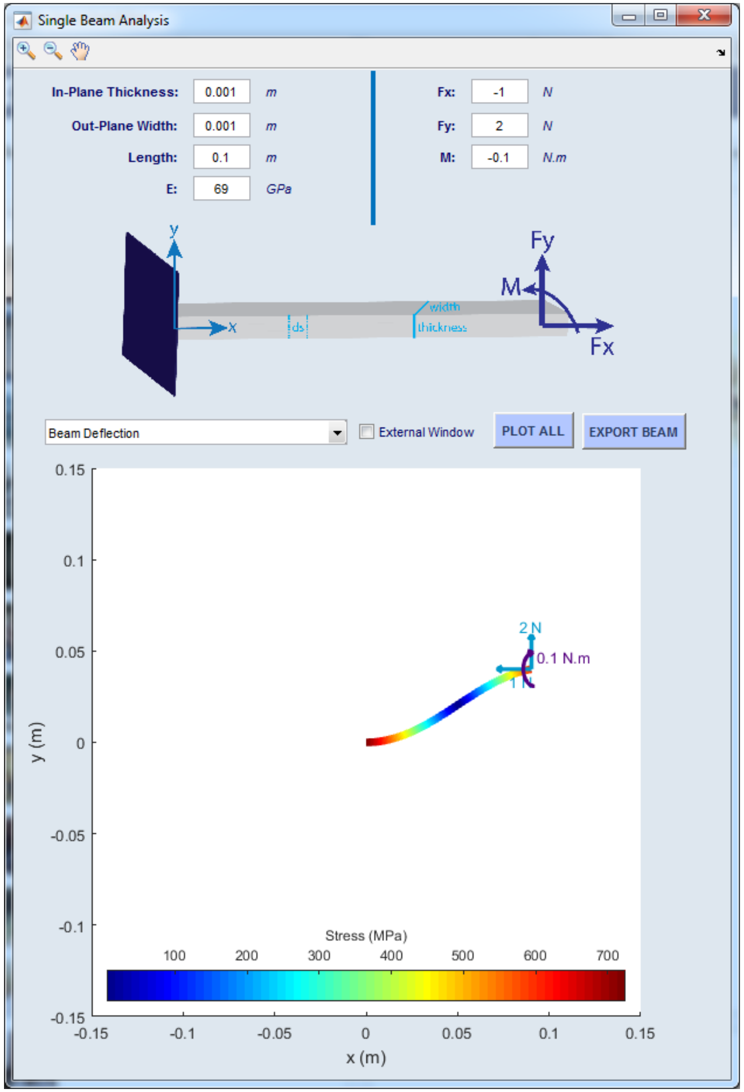
- Single beam subject to arbitrary end load
- Large deflection Euler-Bernoulli beam equation

$$\frac{M}{EI} = \frac{d\theta}{ds}, M = (a - x)F_y - (b - y)F_x + M_0$$

- Numerically solves as a boundary value problem for $\theta(s)$

$$\theta'' = F_x sin\theta - F_y cos\theta,$$
$$\theta(0) = 0, \theta'(L) = M_0,$$

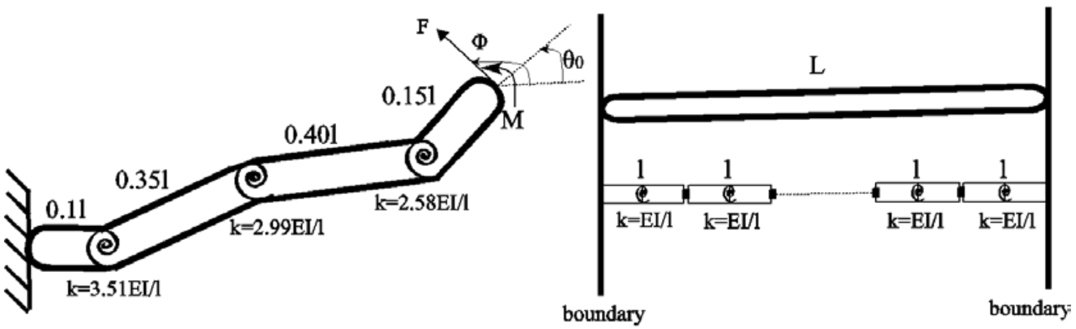
- Very fast analysis (20 ms ~ 30 ms)
- Aimed for classroom use
- Available to be downloaded as an external program with source codes



Pseudo-Rigid-Body Models

A Pseudo-Rigid-Body (PRB) model converts flexible beams into a series of rigid links connected with torsion springs. The advantages of using PRB methods are:

- Use approaches developed for rigid body mechanisms
- Solve large deflection
- Intuitive designs
- Less computationally intensive



PRB-3R

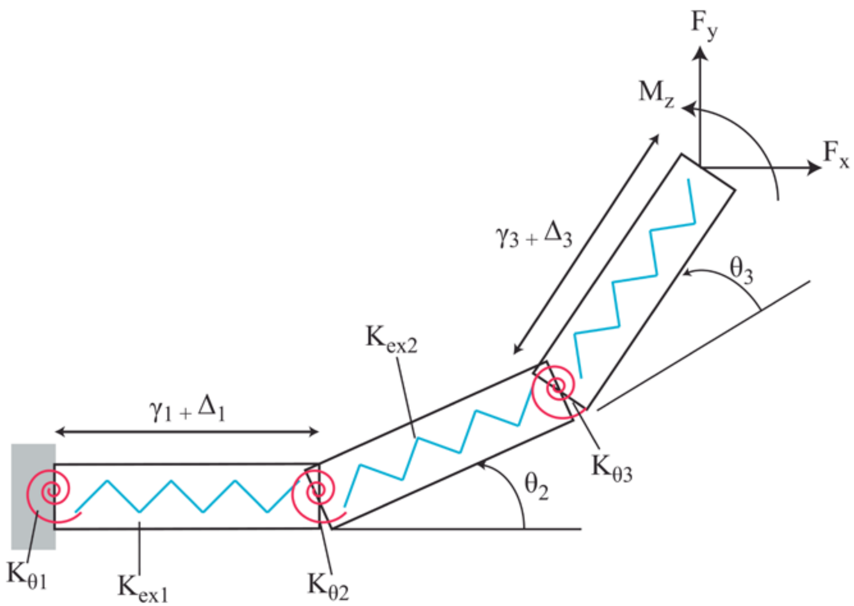
Finite Segment Model (FSM)

PRB Matrix

- PRB Matrix approach was developed by Venkiteswaran and Su.
- Any PRB Model can be represented with a PRB Matrix Ω

$$\Omega = \begin{bmatrix} k_{\theta 1} & k_{ex1} & \gamma_1 \\ k_{\theta 2} & k_{ex1} & \gamma_2 \\ k_{\theta 3} & k_{ex1} & \gamma_3 \end{bmatrix}$$

- Each segment has 3 parameters : $\gamma, k_\theta, \mathrm{k_{ex}}$
- Segment length: $l_i = \gamma_i L_{beam}$
- i^{th} Torsion Spring Magnitude: $K_{\theta i} = k_{\theta_i}\frac{EI}{L}$
- Segment Axial Stiffness: $K_{exi} = k_{ex_i}\frac{EA}{L}$
- If i^{th} torsion spring is not present, $k_{\theta i}$ = Inf
- If the segment is rigid, k_{ex_i}= Inf

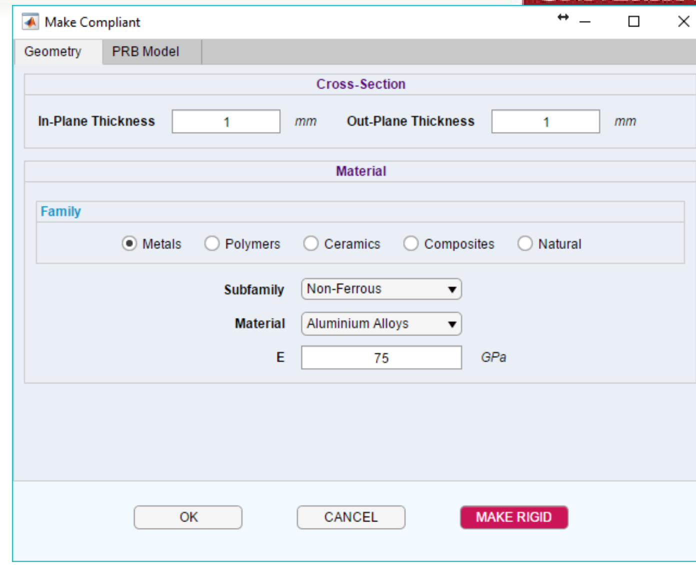


General PRB model with 3 elements

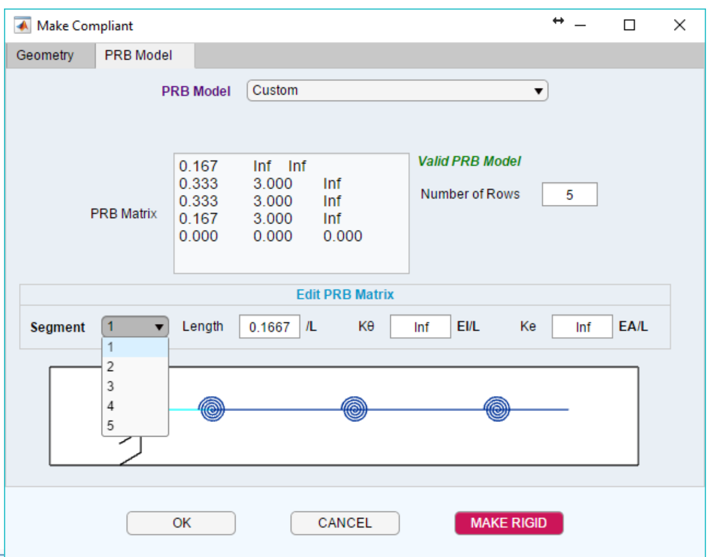
Venkiteswaran VK, Su H-J. A parameter optimization framework for determining the pseudo-rigid-body model of cantilever-beams. Precis Eng (2014),

PRB Models

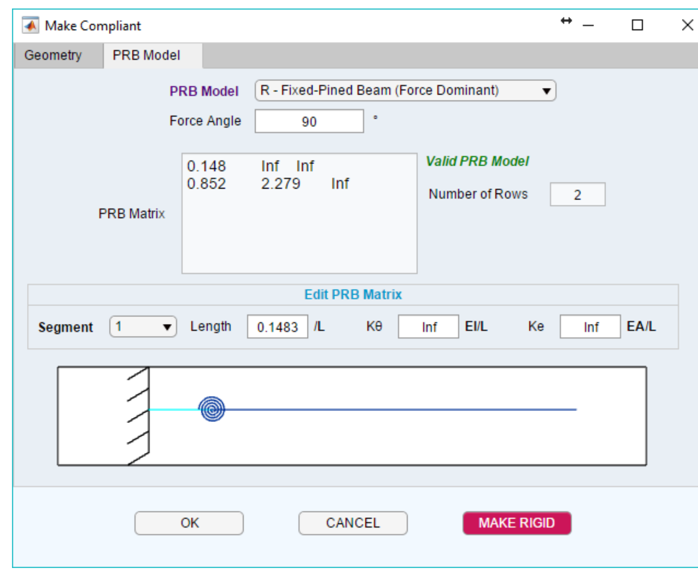
- You need to enter cross-section properties for a PRB beam.
- Since PRB-R is load dependent, an approximate end force angle can be specified for more accurate analysis.
- All PRB Models are customizable. Select a segment and update the three parameters.
- Number of segments can be specified for the PRB-FSM and PRB–Custom Models.



Cross-Section Properties



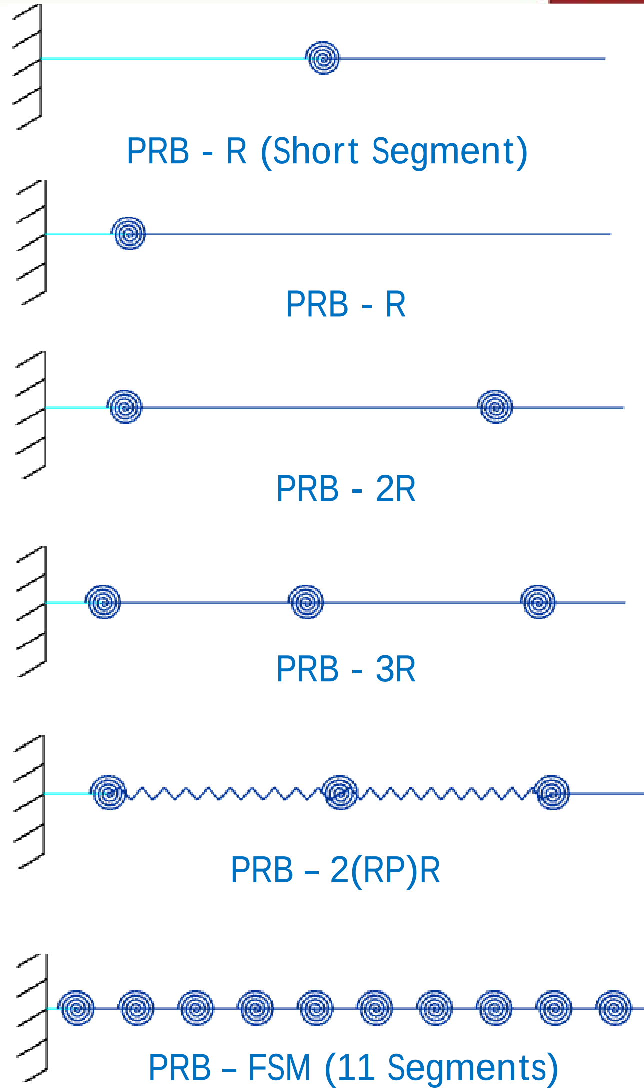
PRB - Custom



PRB - R

Finite Segment Model & PRB Models

- 6 built in PRB Models are available for DAS 2D.
- PRB-R(short segment) is only accurate for very short beams.
- PRB-R is a load dependent model.
- PRB-2R and PRB-3R are optimized for a range of loads and thus, load independent.
- PRB-2(RP)R is the only model that allows axial extension.
- PRB-FSM (Finite Segment Model) is a model that consists of many identical segments.
- You can create your own PRB Model in DAS 2D.
- Increasing number of segments in a PRB Model, increase the analysis time.
- Since PRBM approach converts the links to rigid links, any link (any valid joint combination) can be specified as PRB beam.



Compliant Link Module Demonstration

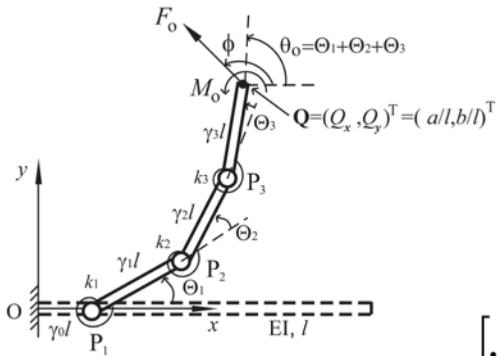
Computing Beam Shape and Stress

- One of the biggest shortcomings of PRB Models is the lack of actual beam shape and stress information.
- PRB Models have high accuracy for tip location and tip angle but three or four rigid segments that typically present in a PRB Model are not enough to accurately represent the beam shape.
- After energy minimization, deflections of the PRB links is the only information.
- Tip loads can be found from the energy stored in the torsion springs.
- Euler-Bernoulli Beam equation can be solved for these tip loads.
- This procedure is executed once after energy minimization is done and the Euler-Bernoulli Beam equation can be solved very fast. Therefore, the time impact is minimal to the software.

- The non-dimensional Jacobian of a PRB model can be used to solve static equations:

$$\begin{Bmatrix} \tau_1 \\ \tau_2 \\ \tau_3 \end{Bmatrix} = [J^T] \begin{Bmatrix} F_x l \\ F_y l \\ M_\circ \end{Bmatrix}$$

Where τ_1, τ_2 and τ_3 are torsion spring torques



A pseudo-rigid-body 3R model for cantilever beam subjected to combined end force and moment

Su, Hai-Jun. "A Pseudorigid-Body 3R Model for Determining Large Deflection of Cantilever Beams Subject to Tip Loads." *Journal of Mechanisms and Robotics* 1, no. 2

$$[J^T] = \begin{bmatrix} -\gamma_1 s_1 - \gamma_2 s_{12} - \gamma_3 s_{123} & \gamma_1 c_1 + \gamma_2 c_{12} + \gamma_3 c_{123} & 1 \\ -\gamma_2 s_{12} - \gamma_3 s_{123} & \gamma_2 c_{12} + \gamma_3 c_{123} & 1 \\ -\gamma_3 s_{123} & \gamma_3 c_{123} & 1 \end{bmatrix}$$

J^T has three columns and n (number of torsion springs + linear springs) rows.

- Jacobian transpose needs to be inverted in order to find the tip load.

Computing Beam Shape and Stress

- The J^T can be inverted very easily for PRB models with

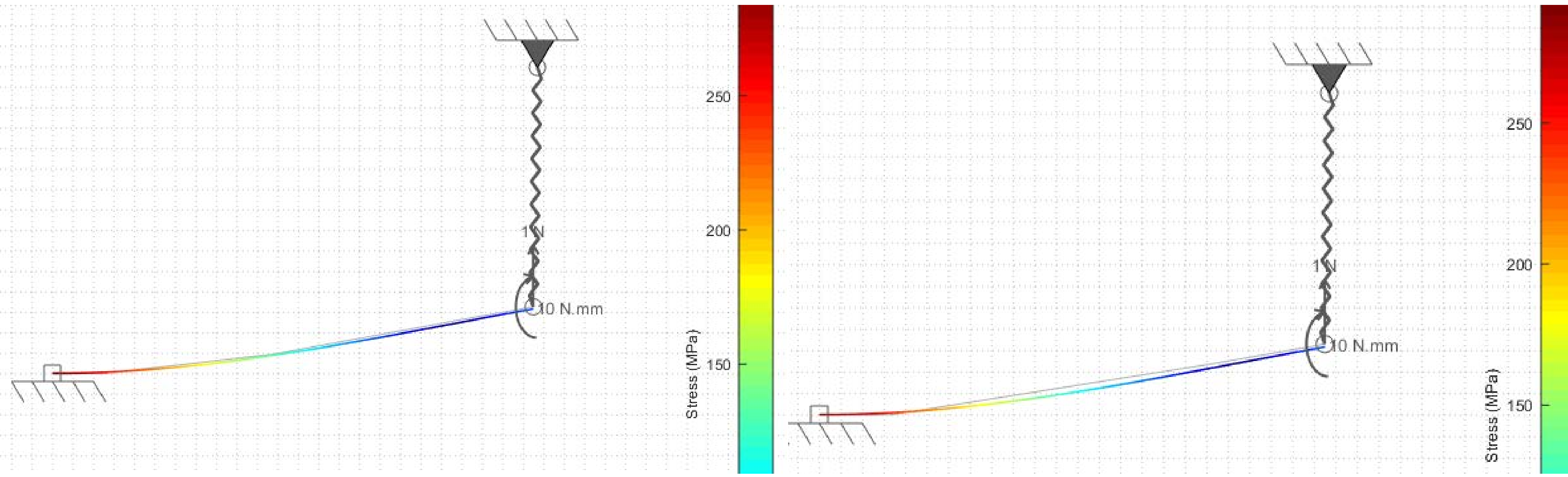
n (number of torsion springs + number of linear springs) =3.

- It is not possible to invert J^T for models that n < 3 (PRB–R and PRB-2R) or n > 3 (PRB-FSM and PRB-2(RP)R).
- Moore-Penrose pseudoinverse can be used to calculate J^T inverse.
- Models with n > 3 can perfectly extract correct tip loads with the Moore-Penrose pseudoinverse.
- Models with n < 3 can only extract two of the correct tip loads with the Moore-Penrose pseudoinverse.
  - This issue can be dealt with by converting some of the rigid links to extremely stiff linear springs $\left(\sim 10^5 \frac{EA}{I}\right)$ until n = 3.

Computing Beam Shape and Stress

- This approach works pretty well for a large range of loads.
- The difference between tip location of the PRB Model and the Euler Beam is an indicator for the accuracy of the PRB Beam for the current problem.

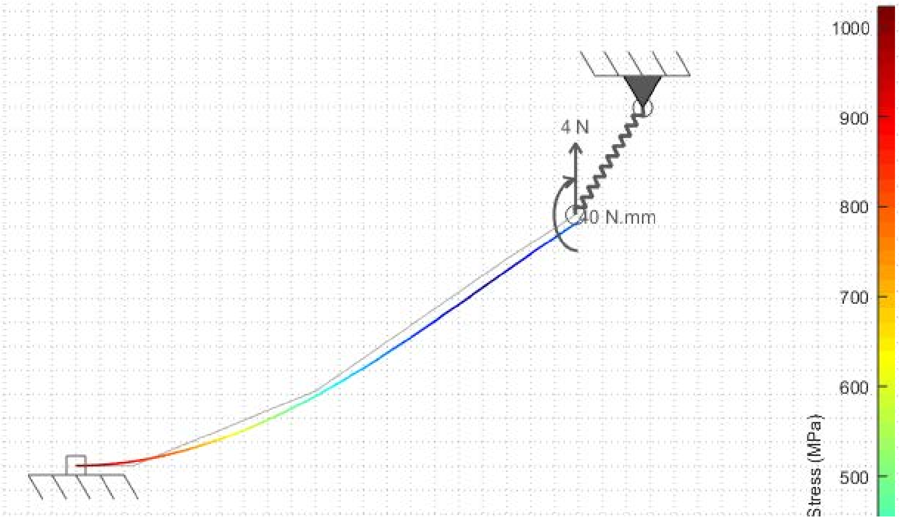
Single Beam Analysis (Linear Spring has no stiffness)



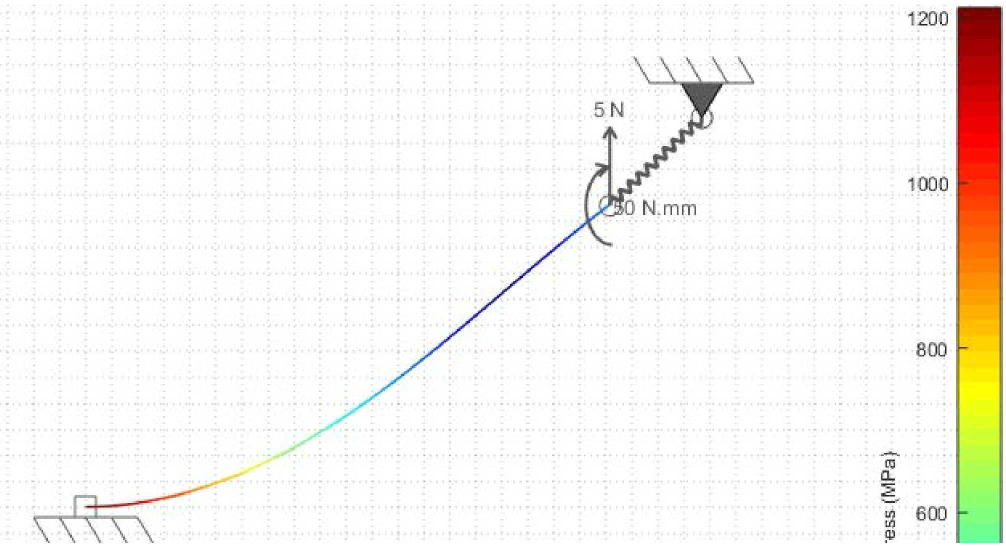
PRB – 3R

PRB – 2R

- When extremely large loads are present, the accuracy may decrease.
- FSM Model usually is in agreement with the Euler Beam equation even for very large loads.



PRB – 3R

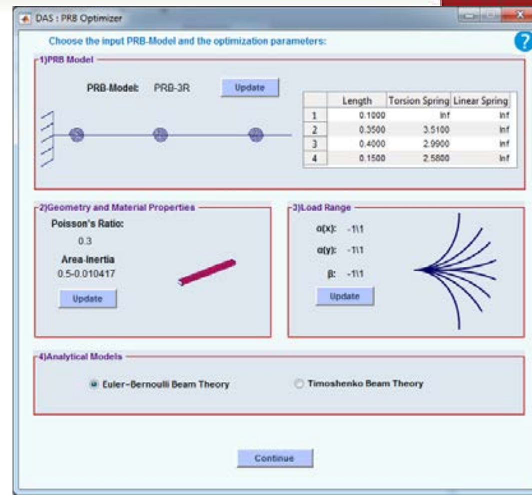


PRB – FSM (11 Segments)

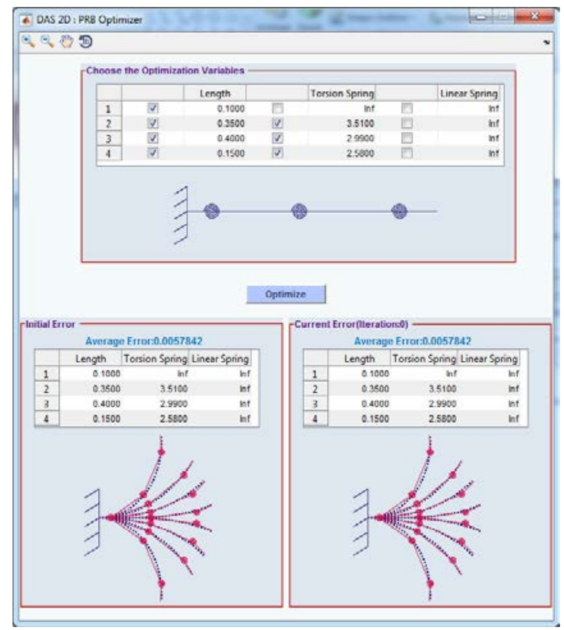
PRB Optimizer

Pseudo-Rigid-Body Model Optimizer

- Optimizes user defined PRB Model
- Create an initial PRB Model
- Define the cross section
- Define a normalized load range
- Select the beam model
- Optimization procedure will come up with updated PRB parameters that result in minimal tip error with the analytical model.



Configuration Screen

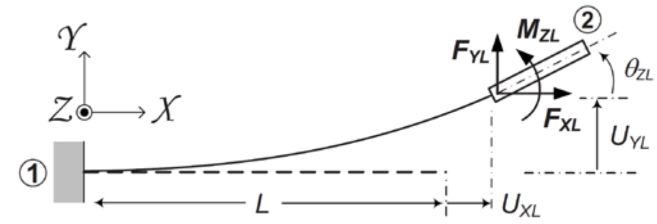


Optimization Screen

Venkiteswaran VK, Su H-J. A parameter optimization framework for determining the pseudo-rigid-body model of cantilever-beams. Precis Eng (2014),

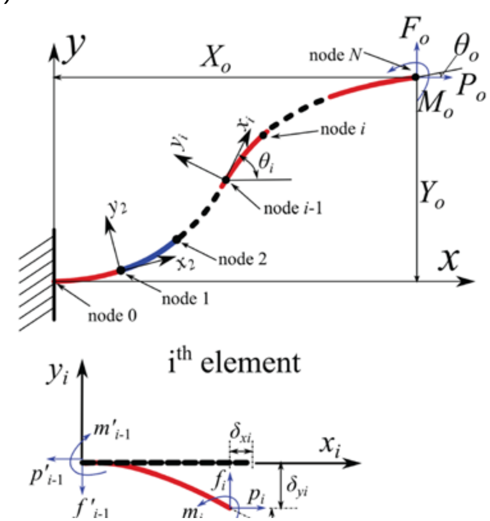
Beam Constraint Model

- BCM was developed for accurately analyzing beams in the intermediate deflection range.
- Deflections within 10% slope change of the beam length can be accurately captured by a single BCM beam.
- BCM is parametric and have closed form equations for the beam deformation and strain energy.
- Axial strain is present in the BCM Beam.
- Chained BCM was developed in order to accurately capture large deflections.
- CBCM consists of multiple BCM segments.



Simple Beam Flexure

Awtar, Shorya, and Shiladitya Sen. "A Generalized Constraint Model for Two-Dimensional Beam Flexures: Nonlinear Strain Energy Formulation." *Journal of Mechanical Design* 132, no. 8 (2010): 081009.



Chained Beam Constraint Model

Ma, Fulei, and Guimin Chen. "Modeling Large Planar Deflections of Flexible Beams in Compliant Mechanisms Using Chained Beam-Constraint-Model." *Journal of Mechanisms and Robotics* 8, no. 2 (2015).

BCM – Energy Equations

- Force – deformation equations for BCM Beams are:

$$\begin{bmatrix} f \\ m \end{bmatrix} = \begin{bmatrix} g_{11} & g_{12} \\ g_{21} & g_{22} \end{bmatrix} \begin{bmatrix} \delta_y \\ \alpha \end{bmatrix} + p \begin{bmatrix} p_{11} & p_{12} \\ p_{21} & p_{22} \end{bmatrix} \begin{bmatrix} \delta_y \\ \alpha \end{bmatrix} + p^2 \begin{bmatrix} q_{11} & q_{12} \\ q_{21} & q_{22} \end{bmatrix} \begin{bmatrix} \delta_y \\ \alpha \end{bmatrix}$$

$$\delta_x = \frac{t^2 p}{12L^2} - \frac{1}{2} [\delta_y \quad \alpha] \begin{bmatrix} u_{11} & u_{12} \\ u_{21} & u_{22} \end{bmatrix} \begin{bmatrix} \delta_y \\ \alpha \end{bmatrix} - p [\delta_y \quad \alpha] \begin{bmatrix} v_{11} & v_{12} \\ v_{21} & v_{22} \end{bmatrix} \begin{bmatrix} \delta_y \\ \alpha \end{bmatrix}$$

Ma, Fulei, and Guimin Chen. "Modeling Large Planar Deflections of Flexible Beams in Compliant Mechanisms Using Chained Beam-Constraint-Model." *Journal of Mechanisms and Robotics* 8, no. 2 (2015).

Where p is the normalized axial force, f is the normalized transverse force and m is the normalized moment. δ_x, δ_y and α are the normalized tip deflections.

$$v = \frac{VL}{EI} = \frac{1}{2} \frac{t^2 p^2}{12L^2} + \frac{1}{2} [\delta_y \quad \alpha] \begin{bmatrix} g_{11} & g_{12} \\ g_{21} & g_{22} \end{bmatrix} \begin{bmatrix} \delta_y \\ \alpha \end{bmatrix} - \frac{1}{2} p^2 [\delta_y \quad \alpha] \begin{bmatrix} q_{11} & q_{12} \\ q_{21} & q_{22} \end{bmatrix} \begin{bmatrix} \delta_y \\ \alpha \end{bmatrix}$$

Strain Energy stored in a BCM Beam

- δ_x, δ_y and α are the selected as the parameters for an BCM beam.
- p can expressed as a function of δ_x, δ_y and α and substituted to the energy equation.

BCM – Beam Shape

$$u_x(x) = \frac{t^2 x}{12} p - [f \quad m] \begin{bmatrix} c_{11} & c_{12} \\ c_{21} & c_{22} \end{bmatrix} \begin{bmatrix} f \\ m \end{bmatrix}$$

$$\begin{bmatrix} u_y(x) \\ \theta(x) \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{bmatrix} f \\ m \end{bmatrix}$$

Ma, Fulei, and Guimin Chen. "Modeling Large Planar Deflections of Flexible Beams in Compliant Mechanisms Using Chained Beam-Constraint-Model." *Journal of Mechanisms and Robotics* 8, no. 2 (2015).

where $x \in [0, 1]$ and k's and c's for $p > 0$ ($r = \sqrt{p}$) are given as

$$k_{11} = \frac{\tanh r}{r^3}[\cosh(rx) - 1] - \frac{\sinh(rx)}{r^3} + \frac{x}{r^2}$$

$$k_{12} = \frac{\cosh(rx) - 1}{r^2 \cosh r}$$

$$k_{21} = \frac{1 - \cosh(rx) + \tanh r \sinh(rx)}{r^2}$$

$$k_{22} = \frac{\sinh(rx)}{r \cosh r}$$

$$c_{11} = \frac{\begin{bmatrix} 4rx + 2rx\cosh(2r) - 4\sinh(rx - 2r) - 4\sinh(rx) \\ -\sinh(2r - 2rx) - 3\sinh(2r) \end{bmatrix}}{8r^5 \cosh^2 r}$$

$$c_{12} = c_{21} = \frac{\begin{bmatrix} 4\cosh(rx) - 2\cosh^2(rx) \\ +\tanh r(\sinh(2rx) - 2rx) - 2 \end{bmatrix}}{8r^4 \cosh r}$$

$$c_{22} = \frac{\sinh(2rx) - 2rx}{8r^3 \cosh^2 r}$$

and for $p < 0$ ($r = \sqrt{-p}$), k's and c's are given as

$$k_{11} = \frac{\sin(rx)}{r^3} - \frac{x}{r^2} - \frac{\tan r}{r^3}(\cos(rx) - 1)$$

$$k_{12} = \frac{1 - \cos(rx)}{r^2 \cos r}$$

$$k_{21} = \frac{\cos(rx) + \tan r \sin(rx) - 1}{r^2}$$

$$k_{22} = \frac{\sin(rx)}{r \cos r}$$

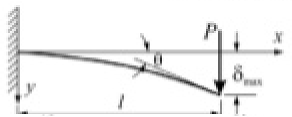
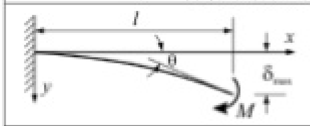
$$c_{11} = \frac{\begin{bmatrix} 4rx + 2rx\cos(2r) - 4\sin(rx - 2r) - 4\sin(rx) \\ -\sin(2r - 2rx) - 3\sin(2r) \end{bmatrix}}{8r^5 \cos^2 r}$$

$$c_{12} = c_{21} = \frac{\begin{bmatrix} 4\cos(rx) - 2(\cos(rx))^2 \\ -\tan r[\sin(2rx) - 2rx] - 2 \end{bmatrix}}{8r^4 \cos r}$$

$$c_{22} = \frac{2rx - \sin(2rx)}{8r^3 \cos^2 r}$$

- Beam shape equations are available for BCM beams.
- f, p and m are found after energy minimization using final δ_x, δ_y and α.
- These equations goes to infinity for small values of p.
- It is found that using Taylor series expansion for $\sqrt{p} < 0.5$ yields very accurate results.

Linear Beam Model

BEAM TYPE	SLOPE AT FREE END	DEFLECTION AT ANY SECTION IN TERMS OF x	MAXIMUM DEFLECTION
1. Cantilever Beam – Concentrated load P at the free end			
	$\theta = \frac{Pl^2}{2EI}$	$y = \frac{Px^2}{6EI}(3l - x)$	$\delta_{max} = \frac{Pl^3}{3EI}$
5. Cantilever Beam – Couple moment M at the free end			
	$\theta = \frac{Ml}{EI}$	$y = \frac{Mx^2}{2EI}$	$\delta_{max} = \frac{Ml^2}{2EI}$

Linear Model

- Linear Model is the approximate solution of the Euler-Bernoulli beam equation.
- Closed form equations for the beam deformation and strain energy are available.
- Axial strain is not present in the Linear Beam.
- A beam can be represented with many Linear Beams segments.
- Since energy equation is the simplest, linear model is the fastest model while being reasonably accurate.

Linear Beam Model - Energy

- δ_y and α are the selected as the parameters for an Linear beam.
- Beam energy can be found by integrating $\dot{\theta}$

$$V = \int_0^L \dot{\theta} dx = \frac{2EI}{L^3}\left(3{\delta_y}^2 + L^2\alpha^2 - 3L\delta_y\alpha\right)$$

- Bending stress is:

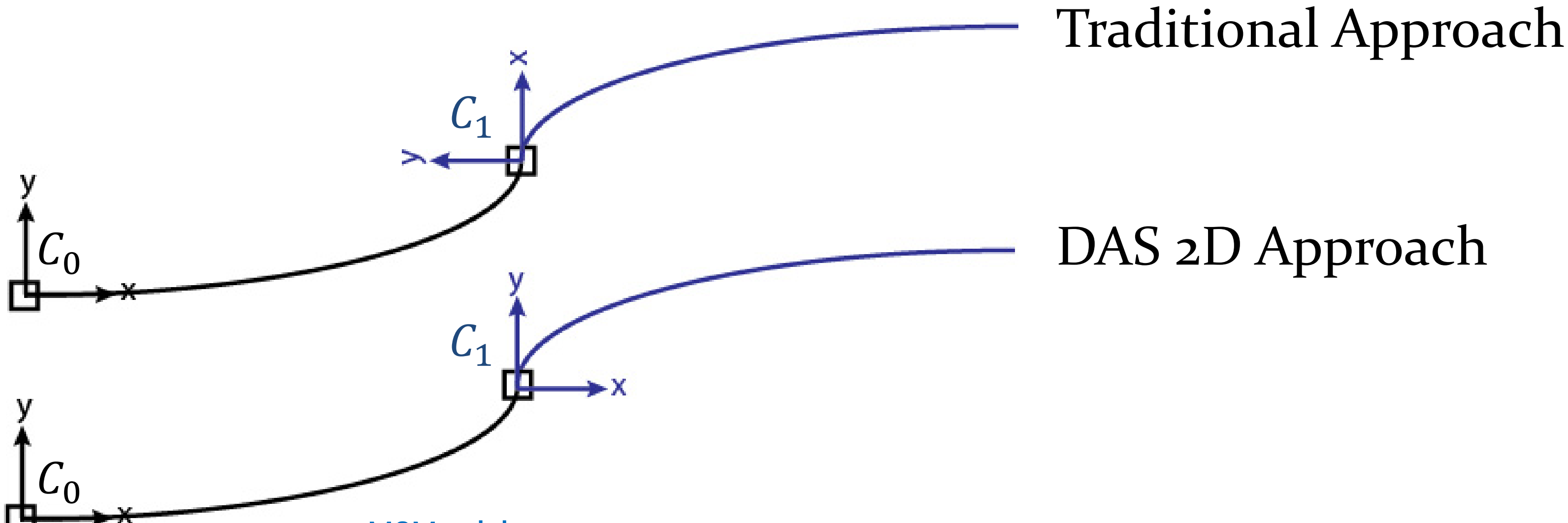
$$\sigma(x) = \dot{\theta}\frac{Et}{2} = \frac{Et}{L^3}\left(3L\delta_y - 6\delta_y x - L^2\alpha + 3L\alpha x\right)$$

Where t is the in-plane thickness.

- It can be seen that equations for the linear model are much simpler than the BCM. Therefore, linear beam model is faster during analysis.

Multiple Segment Models

- Models that have closed form energy equation and have tip angle deflection as a parameter are called Multiple Segment Models.
- PRB Beams and MSM are handled with two different algorithms.



MSM with two segments

- Usually in kinematics, the coordinate system of a segment is parallel to the tip angle of previous segment. The orientation of the i^{th} coordinate system is:

$$\theta_{C_i} = \theta_{c_0} + \sum_0^{i-1} \alpha_i \; (\, i^{th} \; tip \; angle)$$

- Alternatively, all coordinate system can be fixed to be parallel with the global coordinate system.

Analytical Hessian

- Although the difference may seem trivial, it has a great impact in calculating gradient and Hessian matrix of the energy function.
- MATLAB calculates the gradient and hessian matrices by finite difference methods during analysis. If these functions are supplied analytically, the convergence speed gains a huge boost.
- With the traditional approach, a segments end angle is coupled with all the end angles before the segment. In the current approach, the segment angle is coupled only with the previous segment. The disadvantages of the traditional approach is:
  - The optimization problem is more complex and harder to converge because of couplings.
  - The analytical Hessian and gradient is much more difficult to formulate.
  - Hessian matrix has many nonzero elements and cannot be represented with a sparse matrix (consumes a lot of memory).

Multiple Segment Models

The impact of supplying analytical Hessian and gradients are studied using a 26 degrees of freedom (each beam is 2 segment CBCM) mechanism.

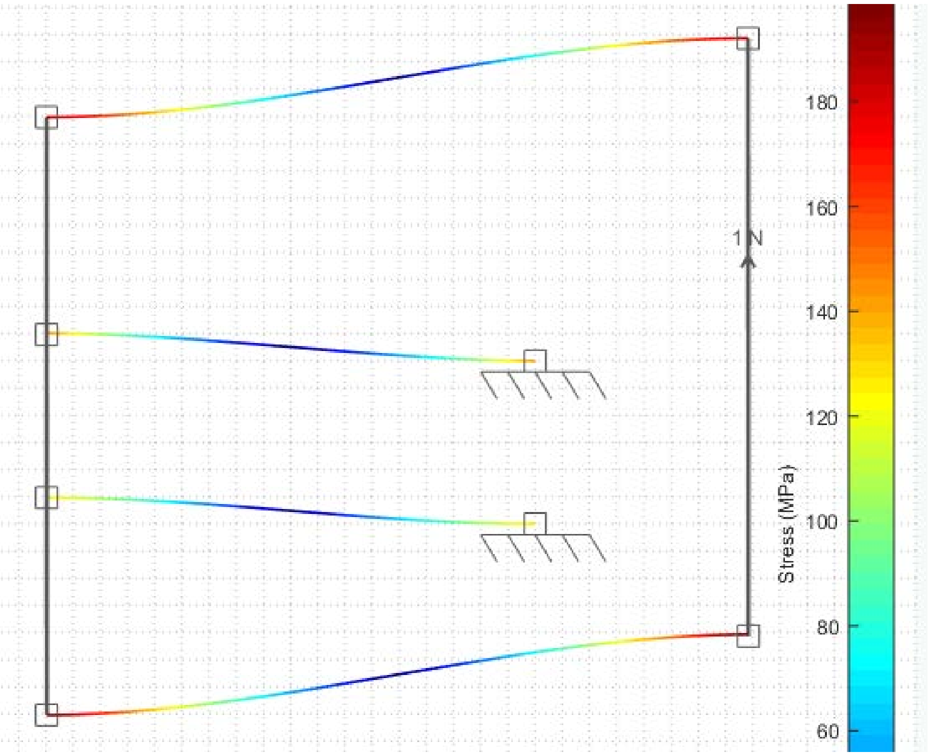
The run times for a single static analysis on a laptop computers are:

No analytical gradient or Hessian
~122 seconds

Only energy gradient
~81 seconds

Energy and Constraint Gradient
~7 seconds

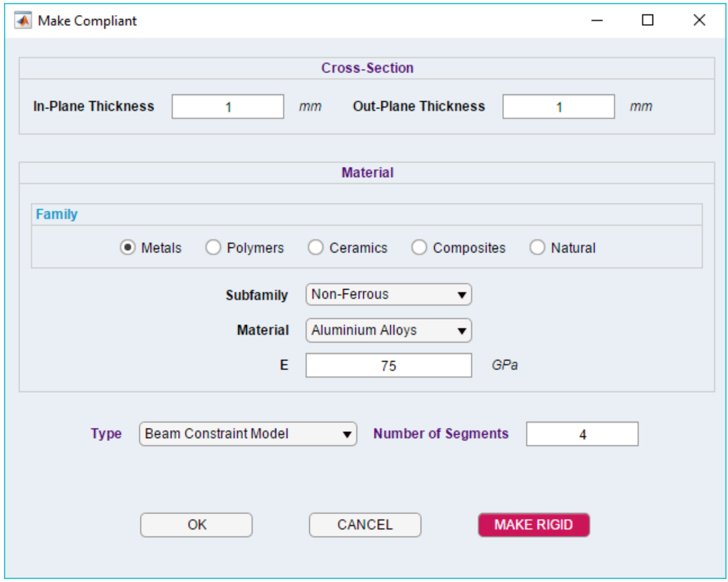
Gradients and Hessian
~1 second



Parallel Guiding Mechanism

Multiple Segment Models

- You need to enter cross-section properties before you create a beam.
- You can decide on the number of segments.
- Only beams that are fixed to the ground can be converted.
- These two models are processed with the same algorithm.
- This algorithm can handle any model which has closed form equation for strain energy.
- Strain energy equation can have arbitrary number of parameters. However, one of them must be the end angle.
- Final version of the program will have customizable beam models.



BCM and LM Properties

BCM and Linear Beam Model Demonstration

Analysis/Synthesis Routines

- Kinematic analysis of rigid-body linkages
  - Interactive kinematic analysis: free dragging
  - Range Kinematics: a range of kinematic motion
- Kinetostatic analysis of compliant mechanisms
  - Load analysis
  - Distance analysis
  - Mechanical advantage
- Synthesis routines
  - Flexural stiffness synthesis
  - Bistable compliant mechanism synthesis

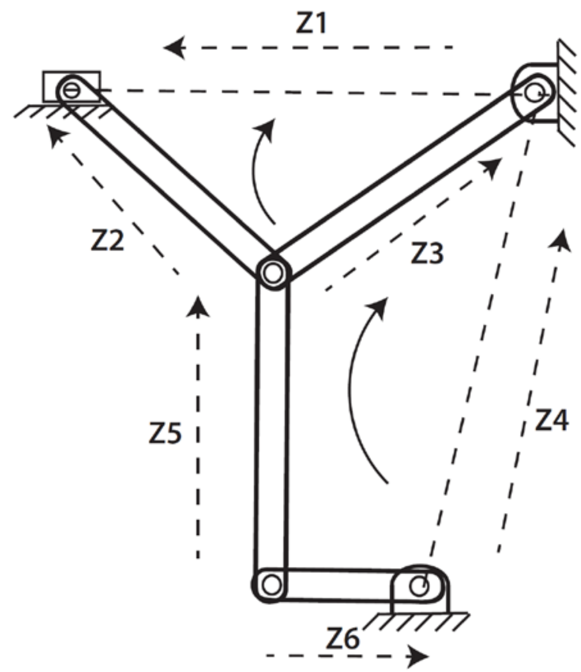
Solver– Kinematic Analysis

- Kinematic equations are used for:
  - Kinematic Analysis
  - Kinematic Constraints for Kinetostatic Analysis
- Independent loops are found using graph theory (base cycles).
- A link with any combination of joints will have the following kinematic equation:

$$\overrightarrow{Z_i} = \overrightarrow{Z_{i0}} + \lambda_i \times \vec{f}(x_i)$$

- For n independent loops, the kinematic equations for a mechanism with l links can be formed as:

$$\sum_{i=1}^{n} C_{ki}\overrightarrow{Z_i} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}, \qquad 1 \le k \le l$$

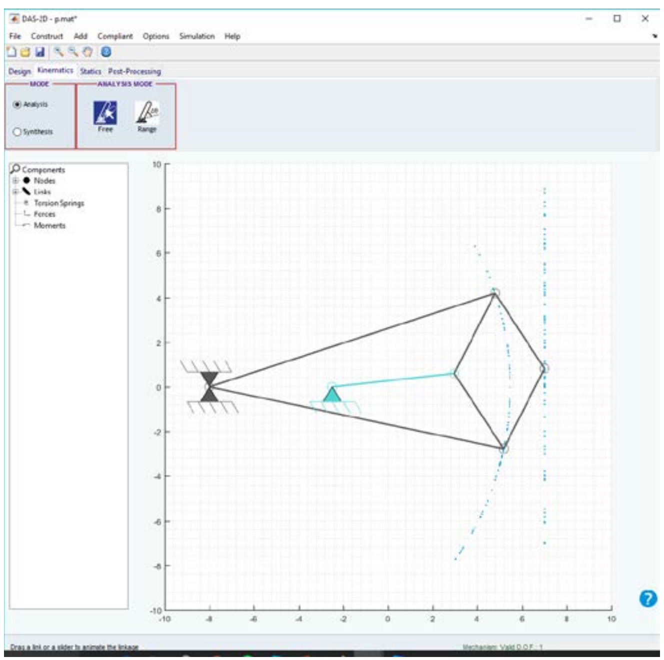


$$\overrightarrow{Z_2} - \overrightarrow{Z_1} - \overrightarrow{Z_3} = 0$$

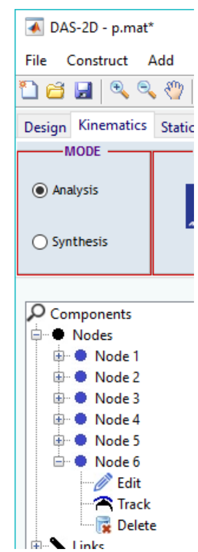
$$-\overrightarrow{Z_6} + \overrightarrow{Z_5} + \overrightarrow{Z_3} - \overrightarrow{Z_4} = 0$$

Interactive Kinematics

- Interactive Kinematics enable users to interactively simulate the mechanism.
- Click and drag a link to drive a mechanism.
- If a mechanism can not move, the driver will be painted red.
- You can track a coupler node by expanding the node and clicking “Track” from the component tree.
- Mechanisms with compliant members can be simulated.



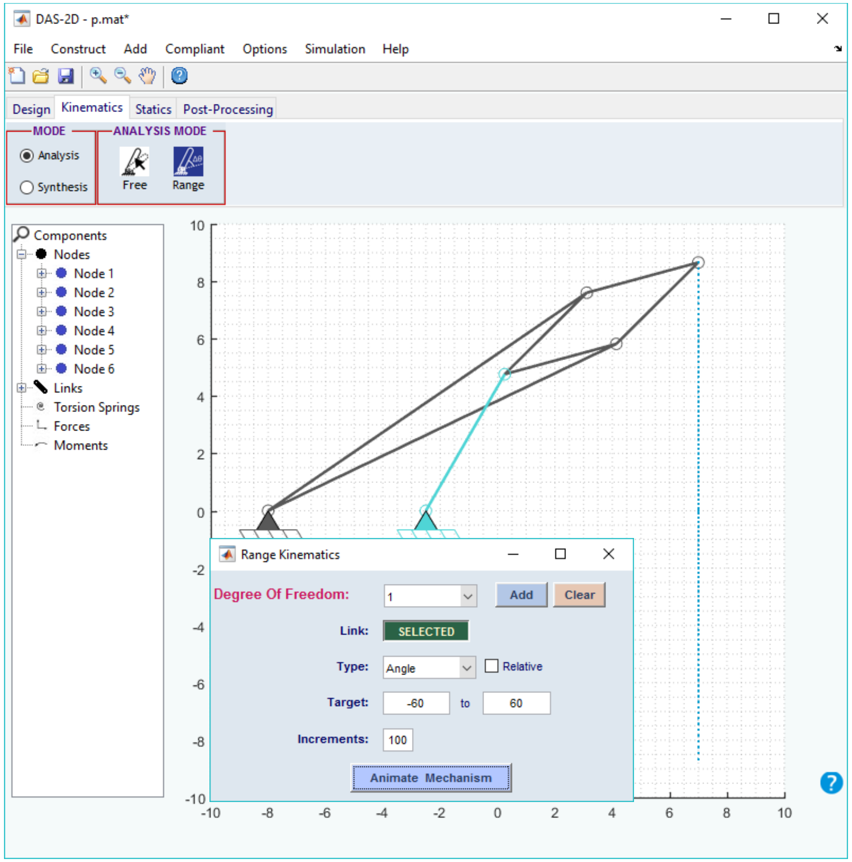
Free Mode



Track a Node

Kinematics for a Range of Motion

- Range Kinematics enable users to explicitly specify the degrees of freedom of the mechanism.
- Select a degree of freedom
- Select a link
- Angle or Length (if possible) of the link can be specified as a range.
- If relative option is selected, the target values will be added to the current value.
- Click Add to finalize a degree of freedom.
- At least first degree of freedom must be added to simulate the mechanism.
- You can track a coupler node by expanding the node and clicking “Track” from the component tree.
- Mechanisms with compliant members can not be simulated.



Range Mode

Kinematic Analysis Demonstration
Ex: KinematicAnalysis

Solver– Kinetostatic Analysis

Kinetostatic analysis is performed by minimizing the total energy of the system.

The total work done on a mechanism

$$W = W_I + W_E = U + V$$

U is the negative of the work done on springs:

$$U = -\sum_{i=0}^{n} \frac{1}{2} k_{ei}(x_i - x_{i0})^2 - \sum_{j=0}^{m} \frac{1}{2} k_{\theta j}\left(\theta_j - \theta_{j0}\right)^2$$

V is the total work done by loads:

$$V = \sum_{i=0}^{n} \int_{ri0}^{ri} \overrightarrow{F_i} \,.\, d\vec{r} + \sum_{j=0}^{m} \int_{\theta_{j0}}^{\theta_j} M_j \, d\theta$$

Optimization of the total energy:

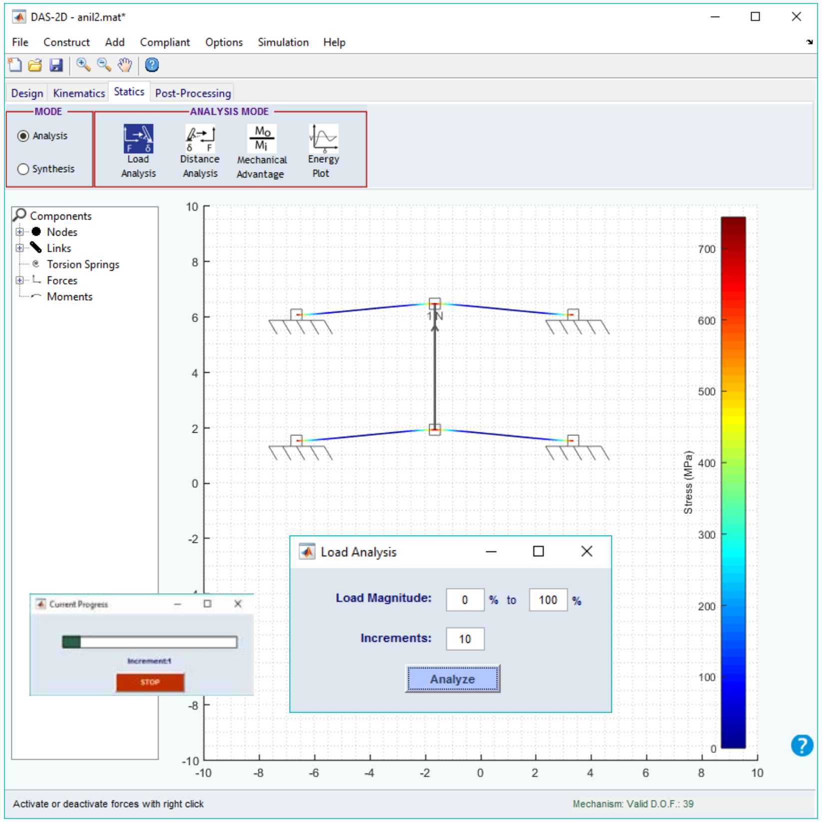
$$\min_{\Psi} f = \left(\sum_{k=1}^{n} E_k(\Psi) - \sum_{i=0}^{n} \int_{ri0}^{ri} \overrightarrow{F_i} \,.\, d\vec{r} - \sum_{j=0}^{m} \int_{\theta_{j0}}^{\theta_j} M_j \, d\theta \right)$$

$$Subject\ to\!:\ \ g = \sum_{i=1}^{n} C_{ki} \overrightarrow{Z_i} = 0$$

Where $E_k(\Psi)$ is the energy stored in a complaint beam, in a torsion spring or a linear spring.

Load Analysis

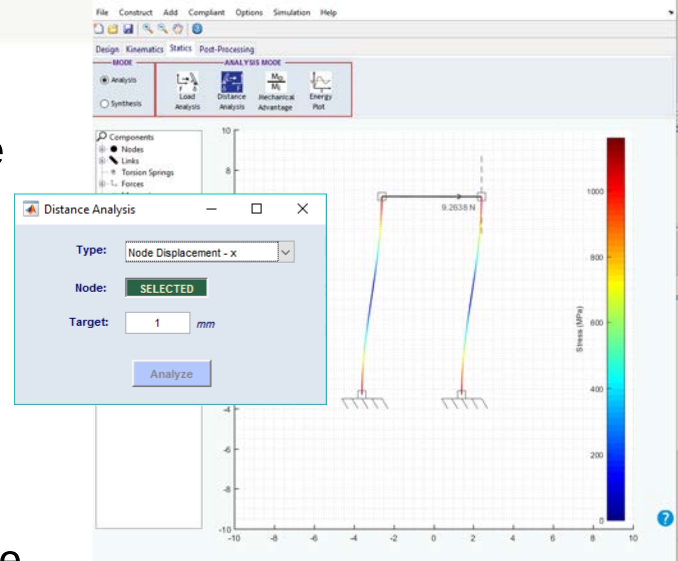
- Load analysis performs the kinetostatic analysis.
- Load can be incrementally increased.
- Lower range for the load must be bigger than or equal to 0%.
- Upper range can be set to any value as long as it is greater than the lower value.
- During the analysis, a progress bar displaying the current increment will show up.
- You can stop the analysis by clicking Stop button. And the analysis will stop at the beginning of next iteration.



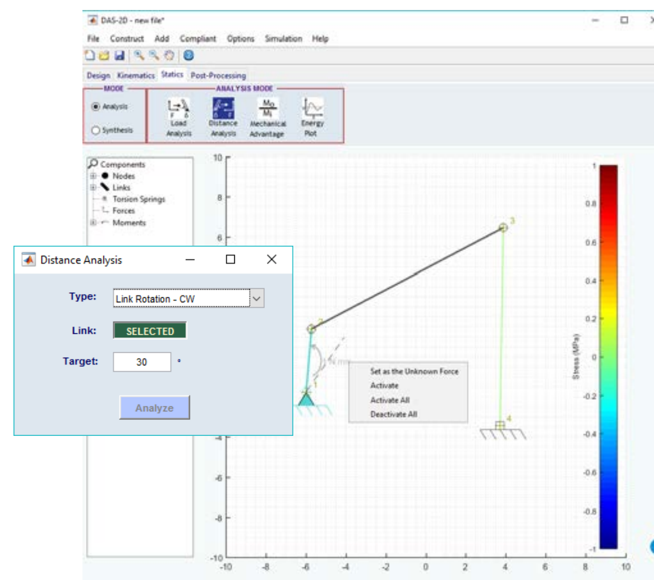
Load Analysis

Distance Analysis

- Distance analysis solves the opposite problem of the load analysis.
- The desired displacement is known, but the magnitudes of the loads are unknown.
- The workspace must contain a force or a moment.
- There are four different displacement types:
    - Rigid Link rotation (CW or CCW)
    - Node displacement (x or y)
- Right click a force or a moment to set the magnitude as unknown.
- After you set the target displacement and the unknown load, you can run the analysis.
- The analysis will determine the load magnitude that results in the target displacement.
- The load magnitude will be saved after the analysis.



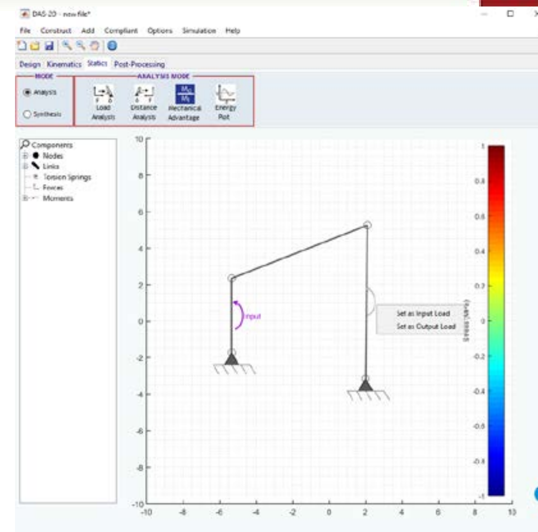
Distance Analysis – Node Displacement



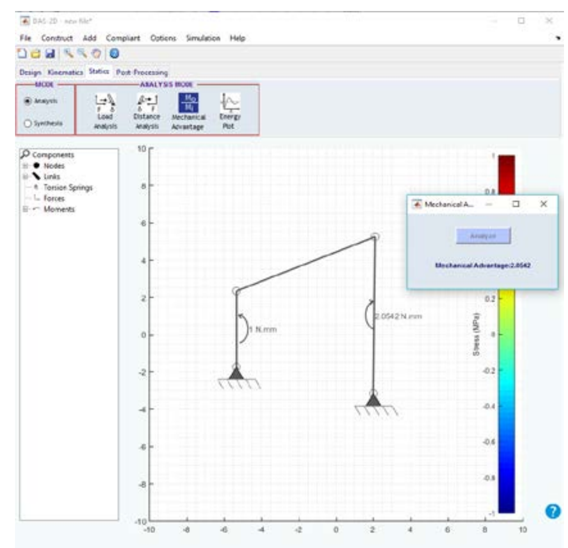
Distance Analysis – Link Rotation

Mechanical Advantage

- Mechanical advantage finds the balancing output load for a unit input load.
- The workspace must contain at least two loads (can be of different type).
- Set the input load by right clicking an existing load.
- Set the output load by right clicking an existing load.
- Click Analyze to start analysis.
- Input load will be assumed as an unit load.
- The balancing output load magnitude will be determined.



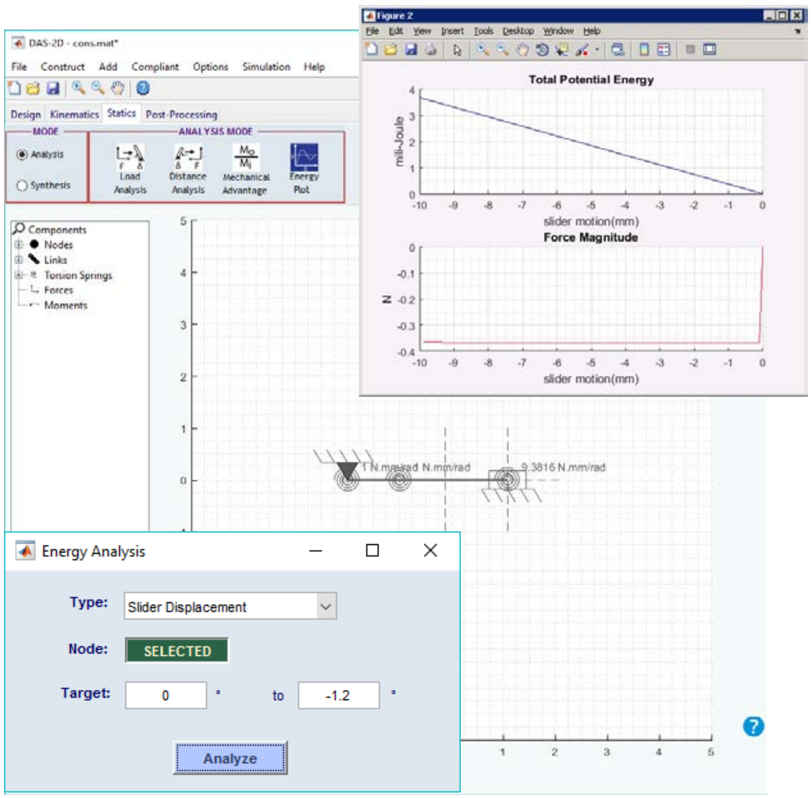
Mechanical Advantage – Select Loads



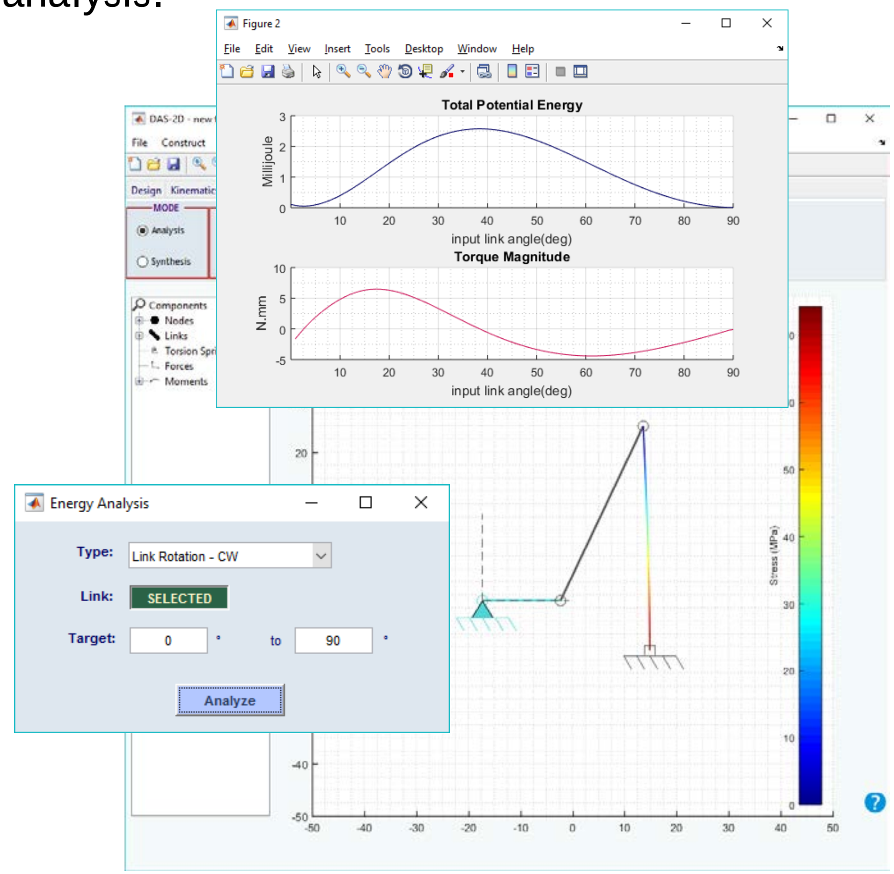
Mechanical Advantage – After Analysis

Energy Plot

- Energy Plot plots the strain energy and the driving load over a distance.
- There are three different displacement types:
    - Rigid Link rotation (CW or CCW)
    - Slider displacement
- After setting the target, you can run the analysis.



Energy Plot – Slider Displacement



Energy Plot – Link Rotation

Kinetostatic Analysis Demonstration

Ex_LoadAnalysis.mat

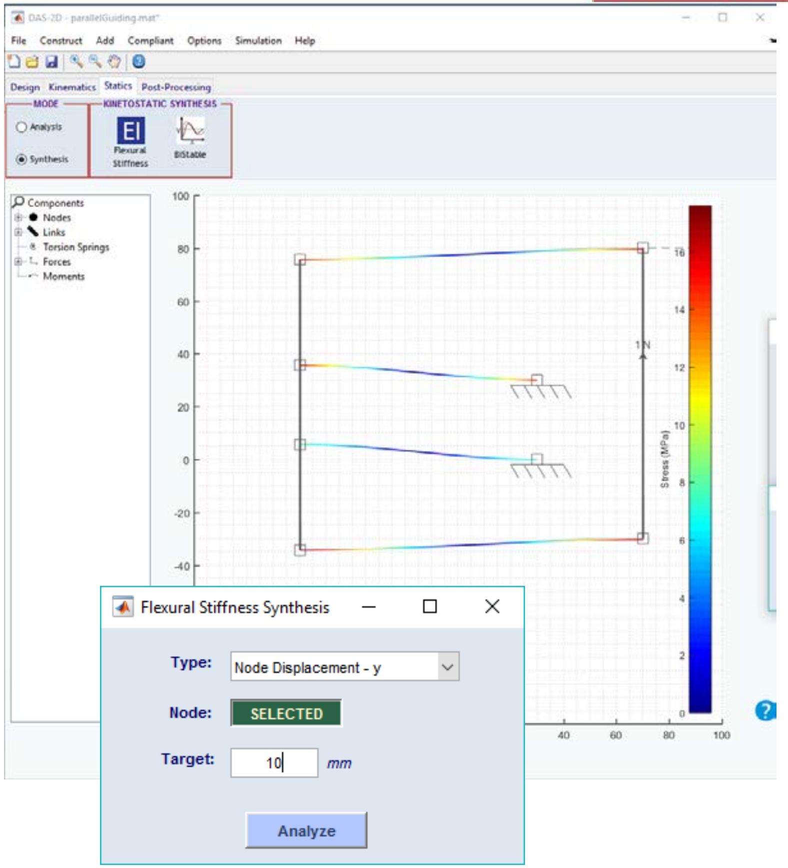
Ex_DistanceAnalysis.mat

Ex_MA.mat

Ex_Energy.mat

Flexural Stiffness Synthesis

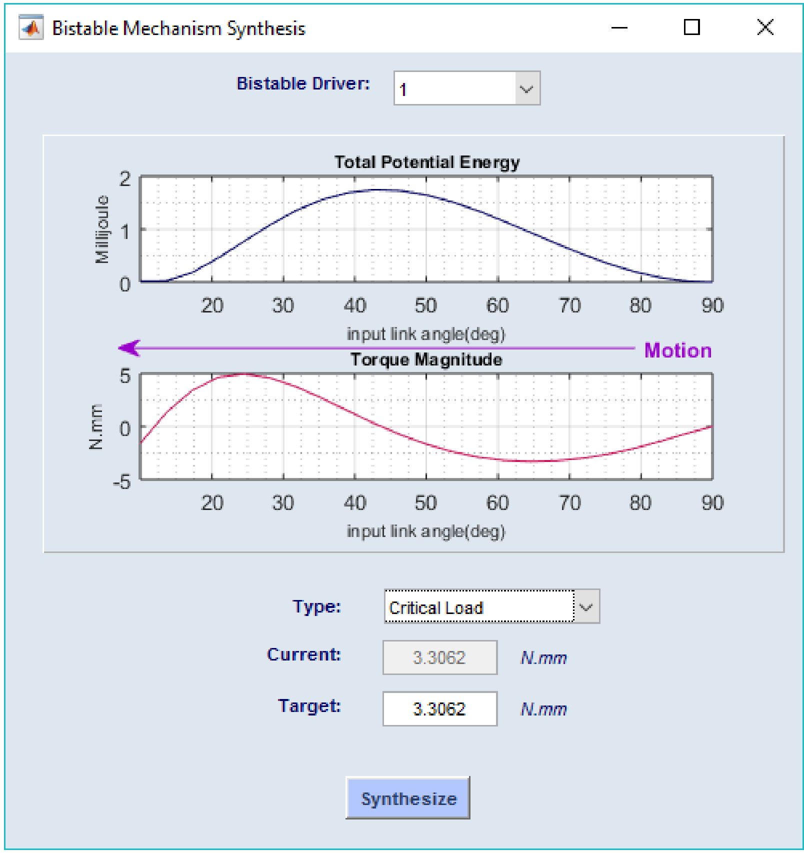
- Flexural stiffness synthesis is very similar to the distance analysis.
- The desired displacement and the magnitudes of the loads are unknown, but flexural stiffness (EI) of compliant members are unknown.
- The workspace must contain a compliant member.
- There are four different displacement types:
    - Rigid Link rotation (CW or CCW)
    - Node displacement (x or y)
- Right click a compliant member to make its **EI** variable.
- After you set the target displacement, you can run the analysis.
- The new EIs will be saved after the analysis.



Flexural Stiffness Synthesis

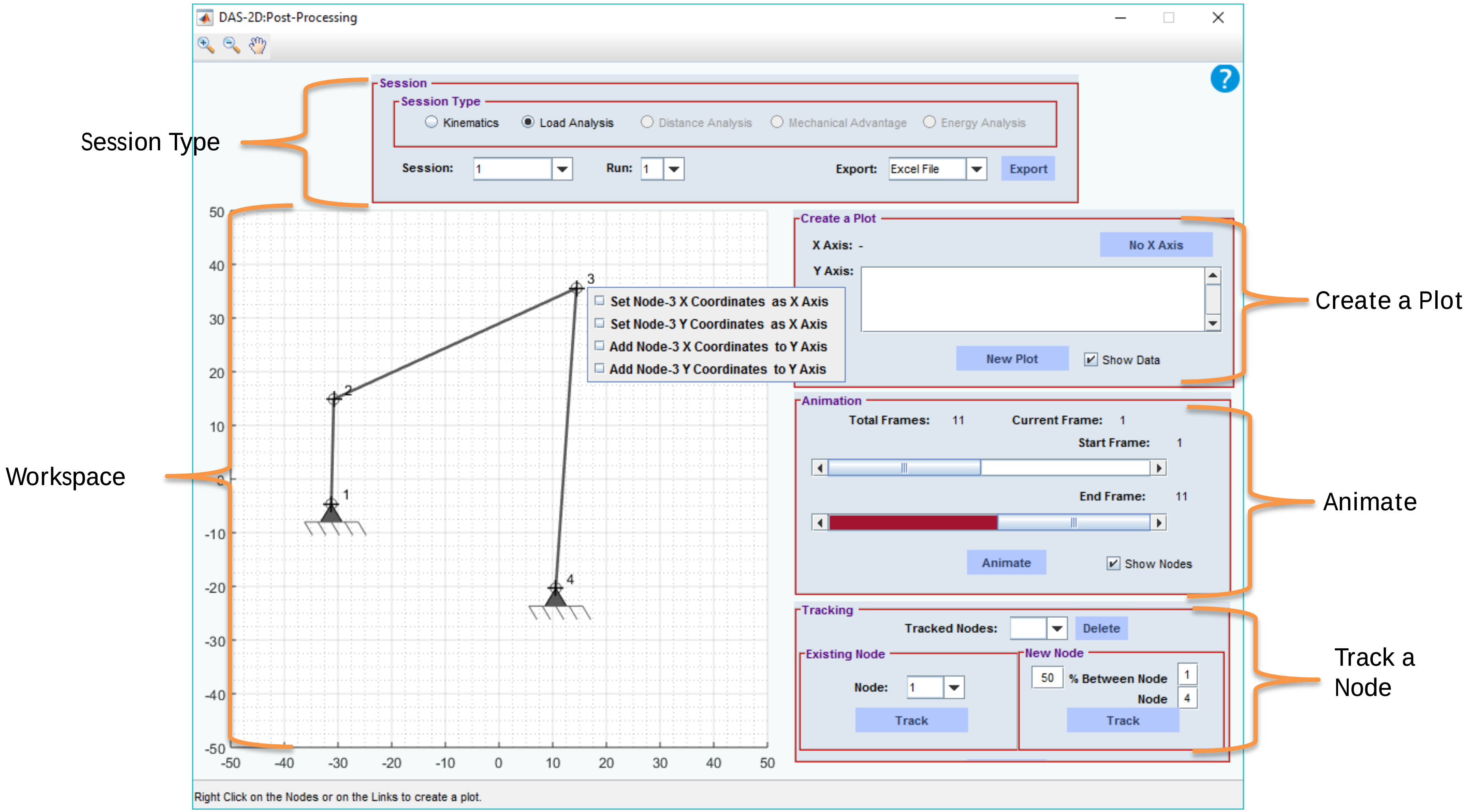
Bistable Mechanism Synthesis

- This synthesis will alter the critical load or locations of (un)stable positions.
- First, the module will check if the mechanism have a driver that results in bistable behavior.
- If there is more than one driver that results in bistable behavior, you can select the desired driver.
- There are three synthesis cases:
    - Critical Load Synthesis
    - Bistable Position Synthesis
    - Stable Position Synthesis
- Bistable and stable position synthesis cases are kinematic synthesis problems. Link lengths will be altered until synthesis is satisfied.
- Critical load synthesis will determine the required flexural stiffness of the compliant members that results in desired critical load.

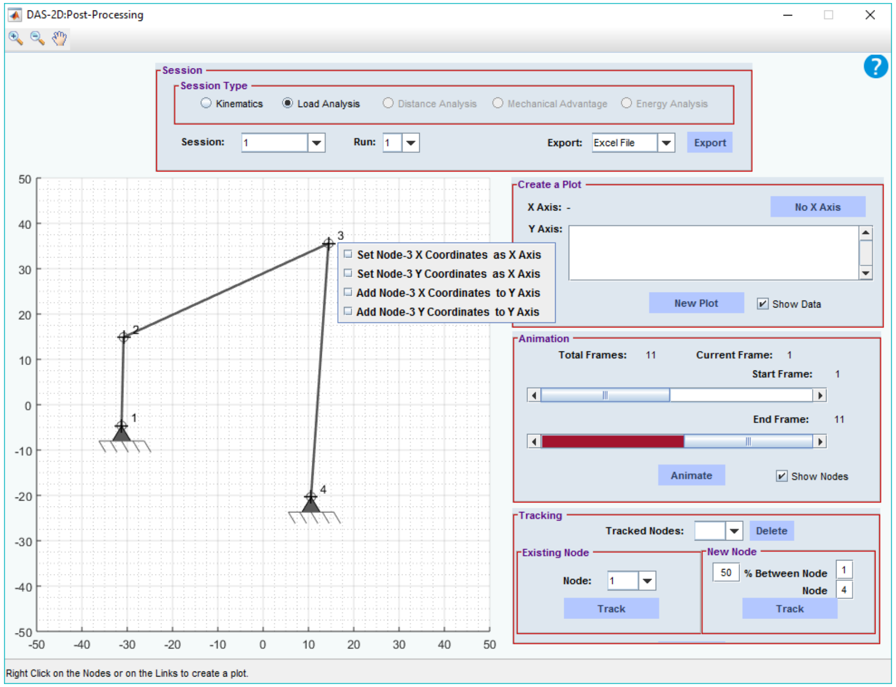


Bistable Mechanism Synthesis

Post-Processing



- All analysis operations are automatically saved.
- Even if mechanism dimensions change, previously saved analysis are not deleted.
- Save files contain the analysis before the save operation.
- You can switch between different analysis types in the top panel.
- The analysis can be animated frame by frame.
- Various plots can be created by right clicking any component.
- If No X Axis button is clicked, the x axis of a plot will be frames.
- All data can be exported as an Excel spreadsheet or a Matlab variable.



Post Processing Demonstration

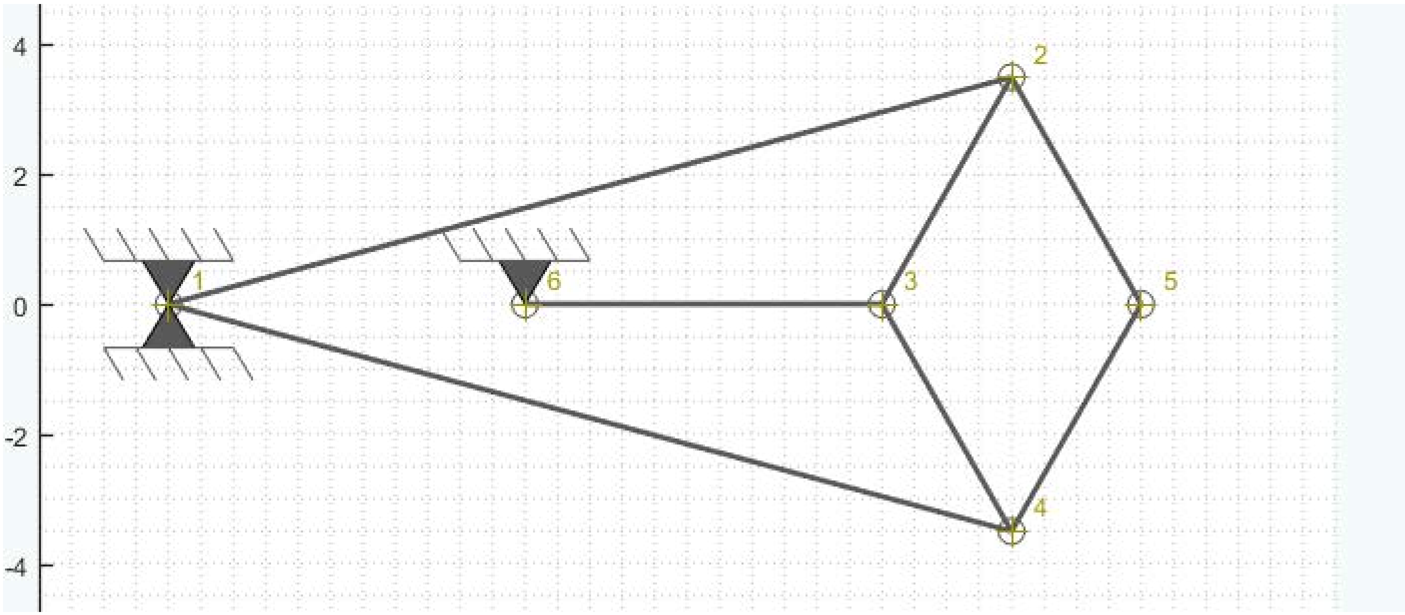
Ex_Post.mat

QUESTIONS?

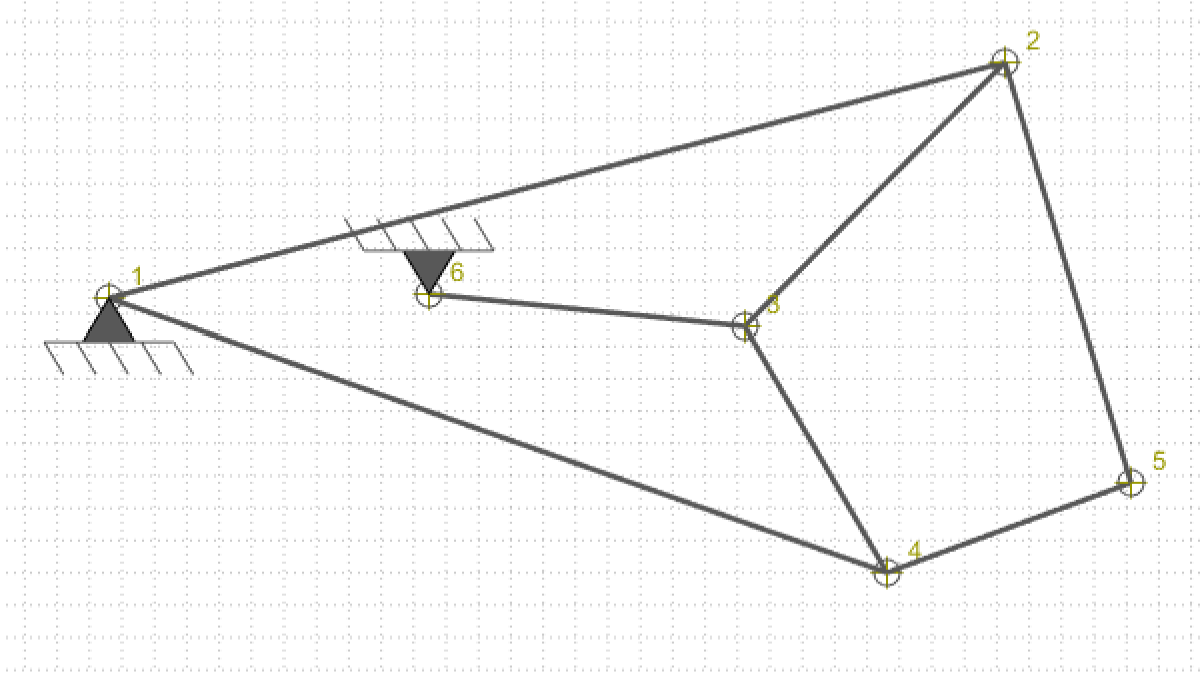
You can send comments and requests for DAS 2D to turkkan.1@osu.edu

Ex.1 Peaucellier-Lipkin Mechanism

The first true straight line mechanism
Start by setting workspace size 10 by 10 from Options-> Workspace

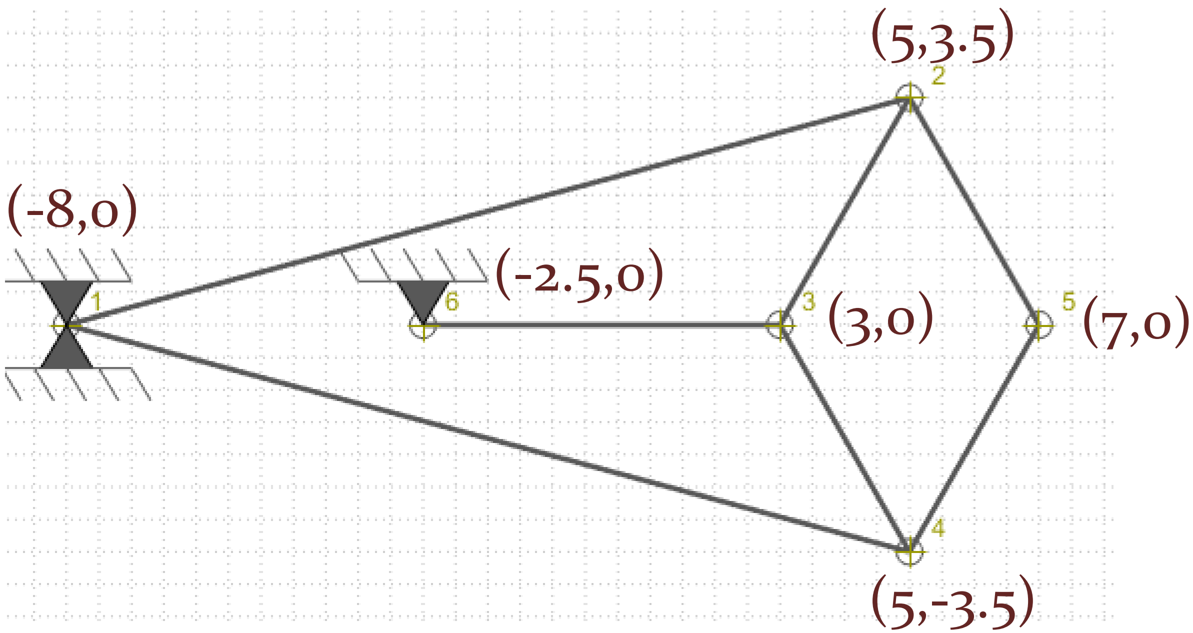


Step 1. Go to sketch mode and draw the mechanism. You will need to press Esc time to time.
Step 2. Go to Model Module and click the links that have different joint types compared to the mechanism below.

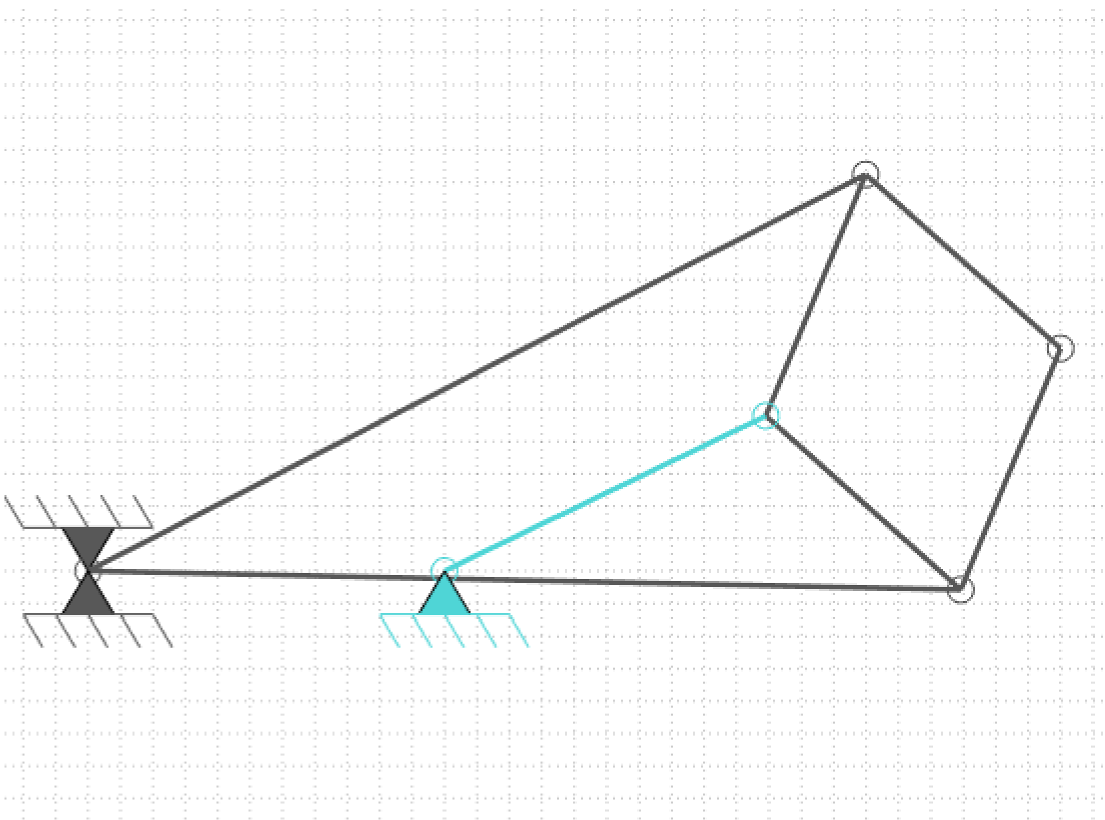


Ex.1 Peaucellier-Lipkin Mechanism

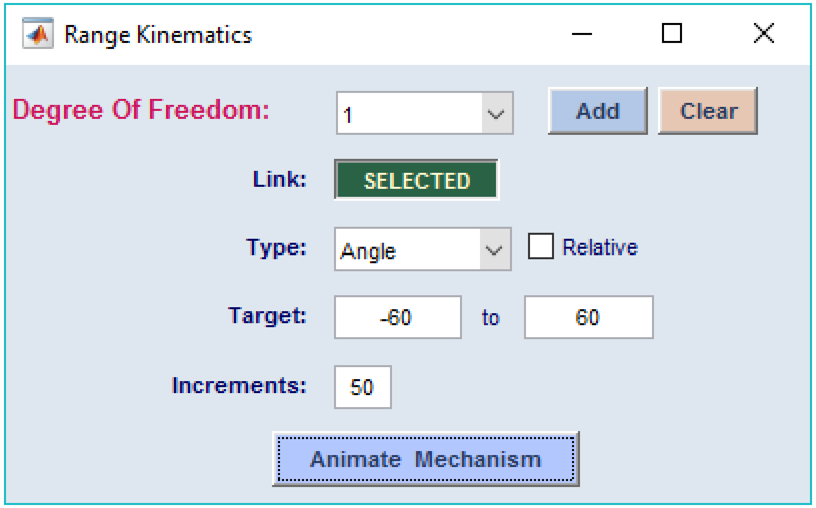
Step 3. Either go to Overview mode and right click nodes and select Edit or go to dimensions and click nodes one by one. Enter the coordinates below.



Step 4. Go to Free Kinematics Mode and experiment by dragging the short crank link

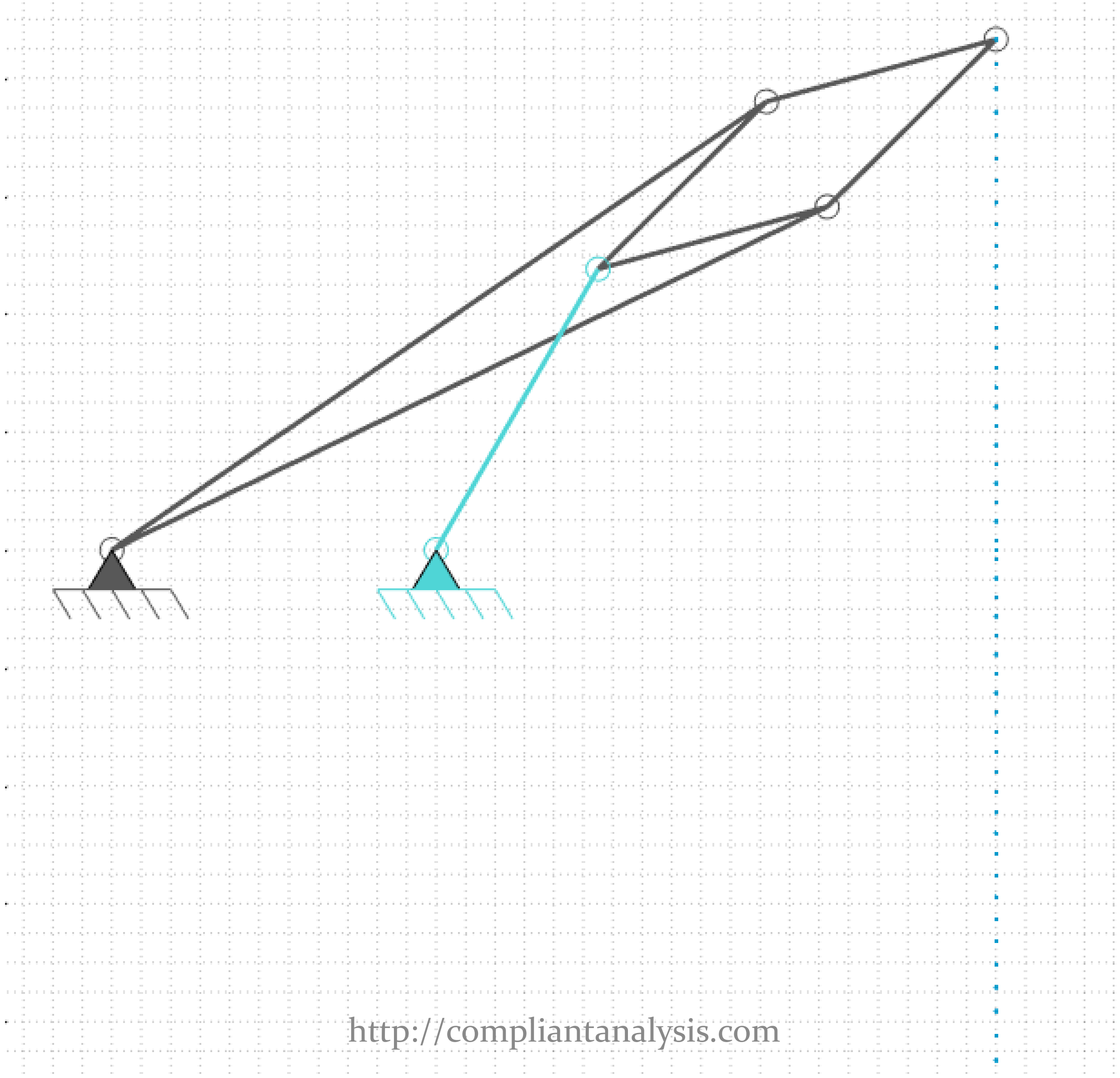


Step 5. Go to Range Kinematics Mode and Click to the short crank and fill the details as shown below.



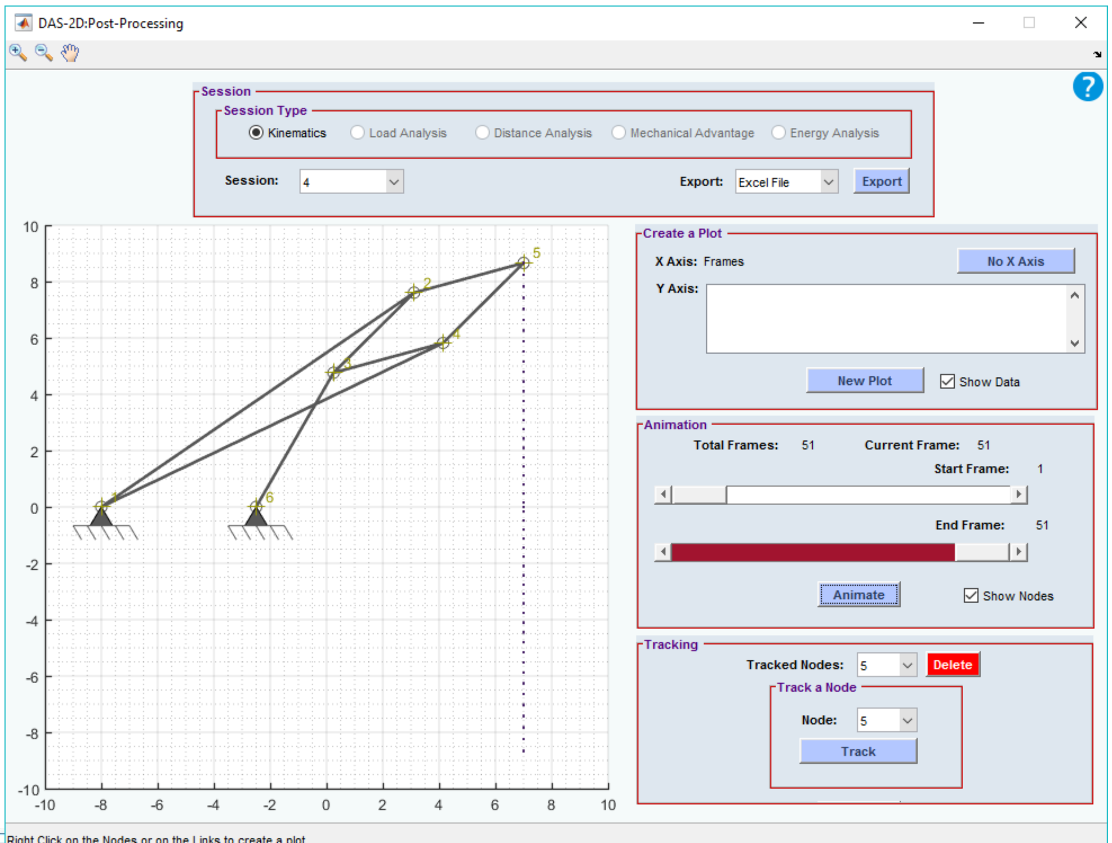
Step 6. Press Add and before clicking animate go to component tree and find the rightmost node and click Track.
Step 7. Press Animate Mechanism.

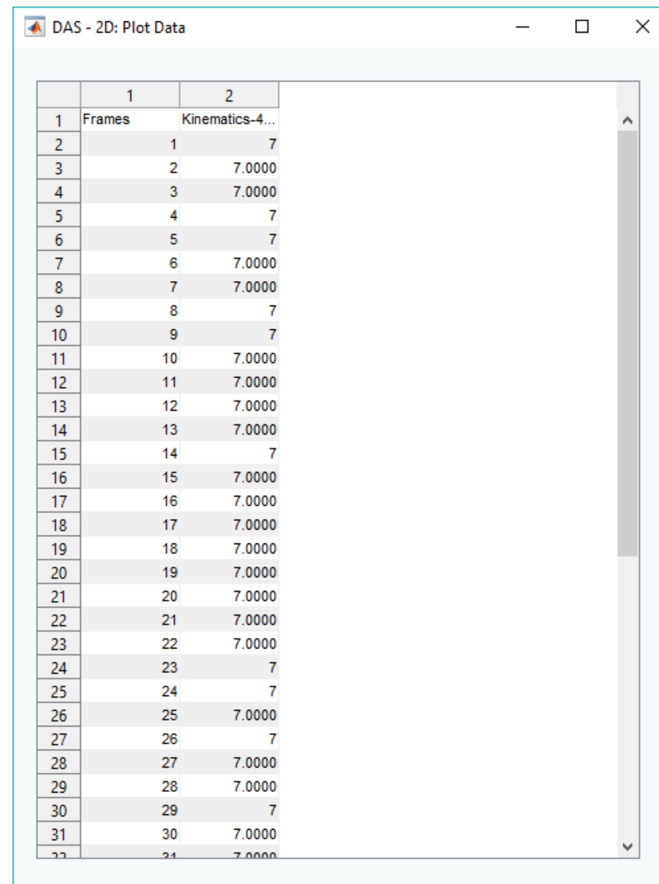
Ex.1 Peaucellier-Lipkin Mechanism



Ex.1 Peaucellier-Lipkin Mechanism

Step 8. Go to Post-Processing Module
Step 9. From down right corner select the correct node to track
Step 10. Hit Animate
Step 11. Right Click Right Most Node and select “Add Node X Coordinates to the Y axis. Click new Plot and observe that X coordinates remain same

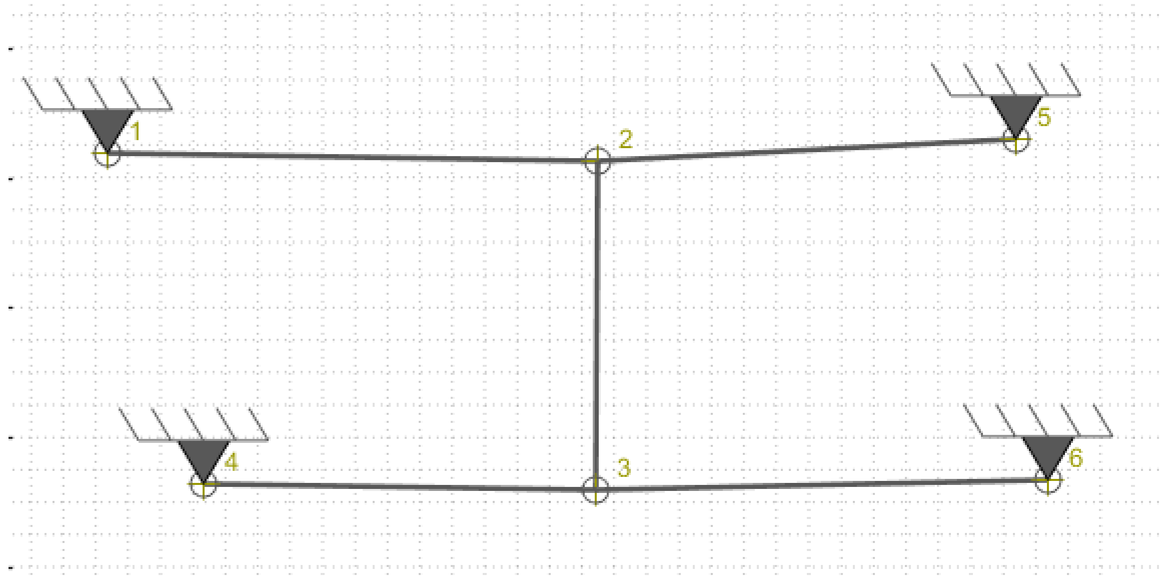




	1	2
1	Frames	Kinematics-4...
2	1	7
3	2	7.0000
4	3	7.0000
5	4	7
6	5	7
7	6	7.0000
8	7	7.0000
9	8	7
10	9	7
11	10	7.0000
12	11	7.0000
13	12	7.0000
14	13	7.0000
15	14	7
16	15	7.0000
17	16	7.0000
18	17	7.0000
19	18	7.0000
20	19	7.0000
21	20	7.0000
22	21	7.0000
23	22	7.0000
24	23	7
25	24	7
26	25	7.0000
27	26	7
28	27	7.0000
29	28	7.0000
30	29	7
31	30	7.0000

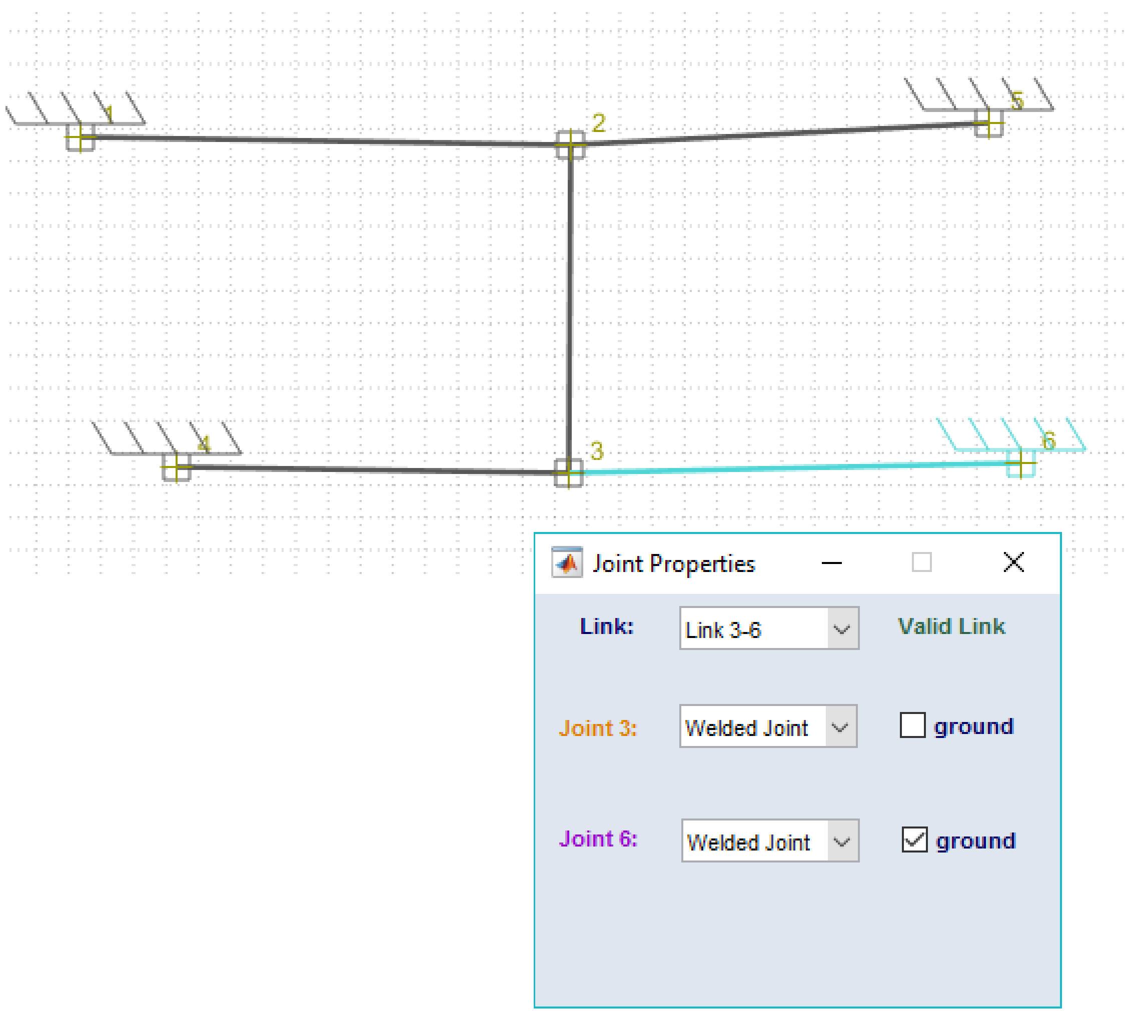
Ex.2 Parallelogram Mechanism

Start by setting workspace size 100 by 100 from Options-> Workspace
Step 1. Sketch the mechanism as shown below



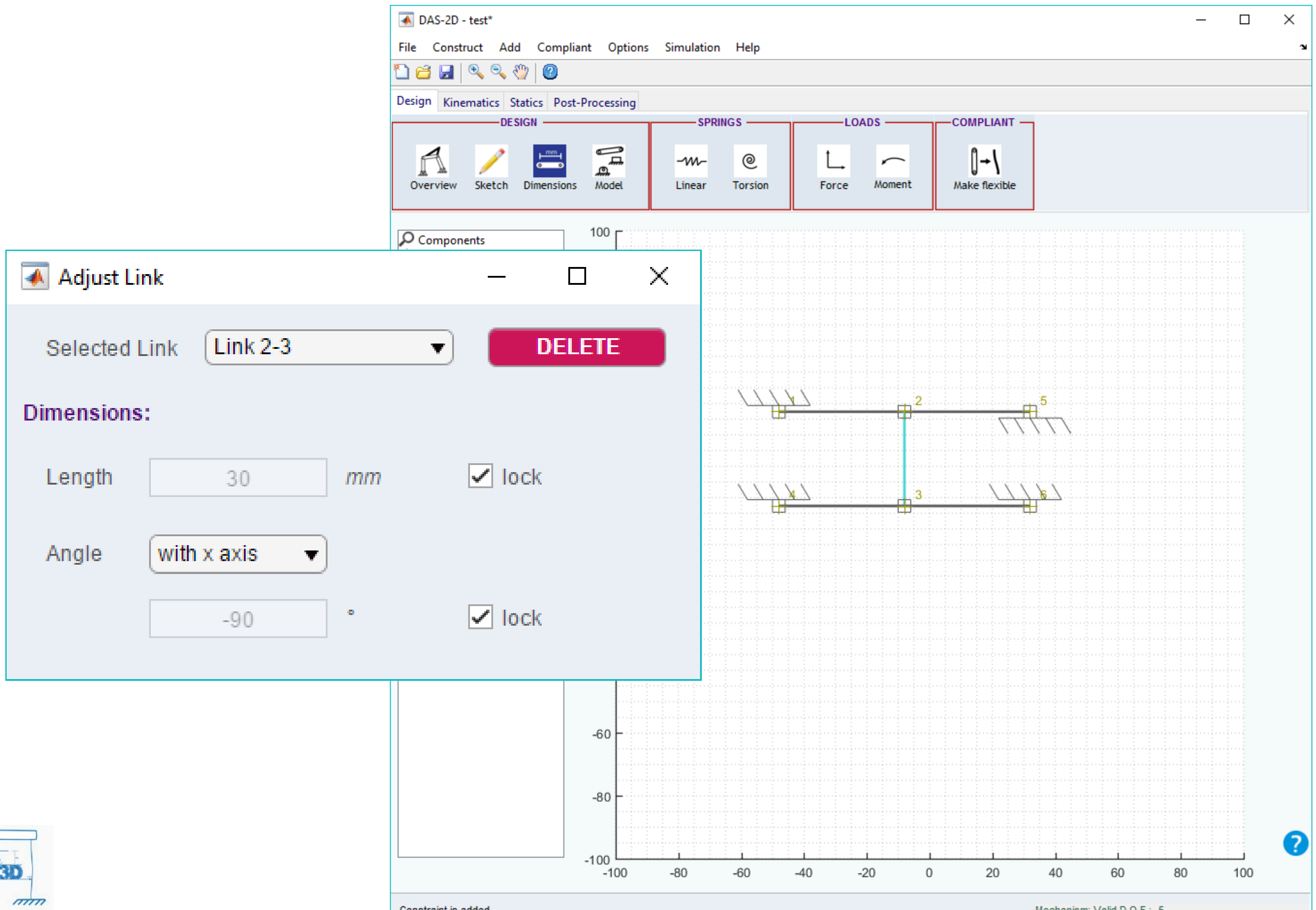
Ex.2 Parallelogram Mechanism

Step 2. Go to Model Module and click links one by one and set joints as welded joint.



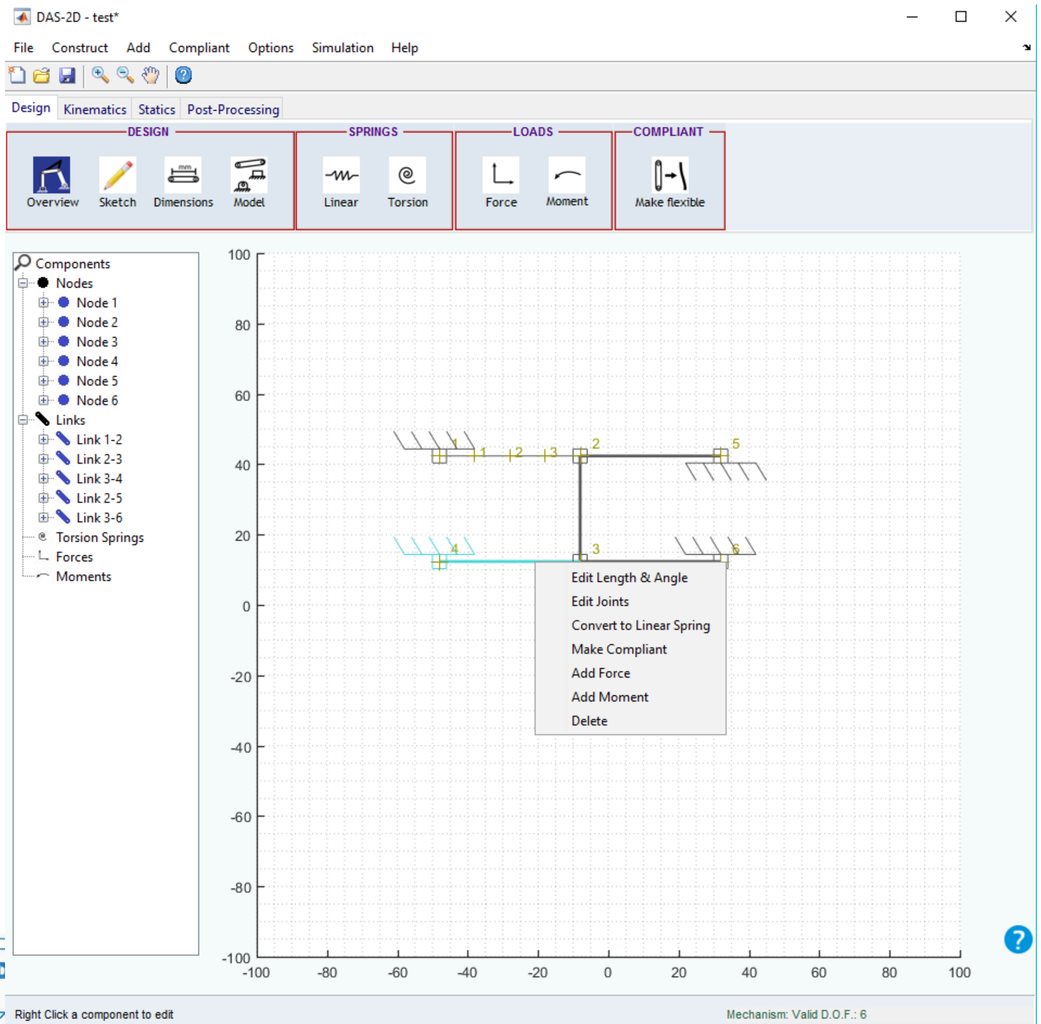
Ex.2 Parallelogram Mechanism

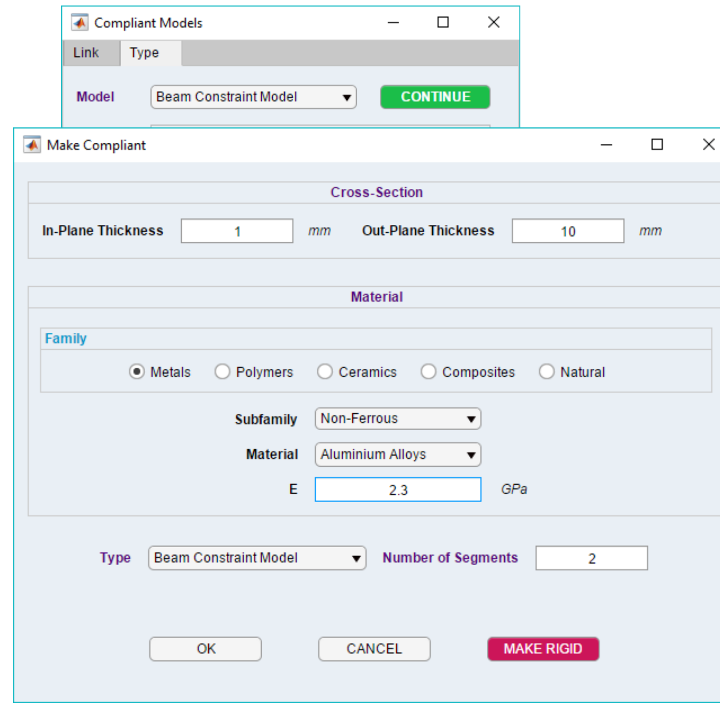
Step 3. Go to Dimensions Module and set the middle link's orientation to 90° (or -90 ° degrees) and length to 30 mm. Other links are parallel to the x axis and have length of 40 mm.



Ex.2 Parallelogram Mechanism

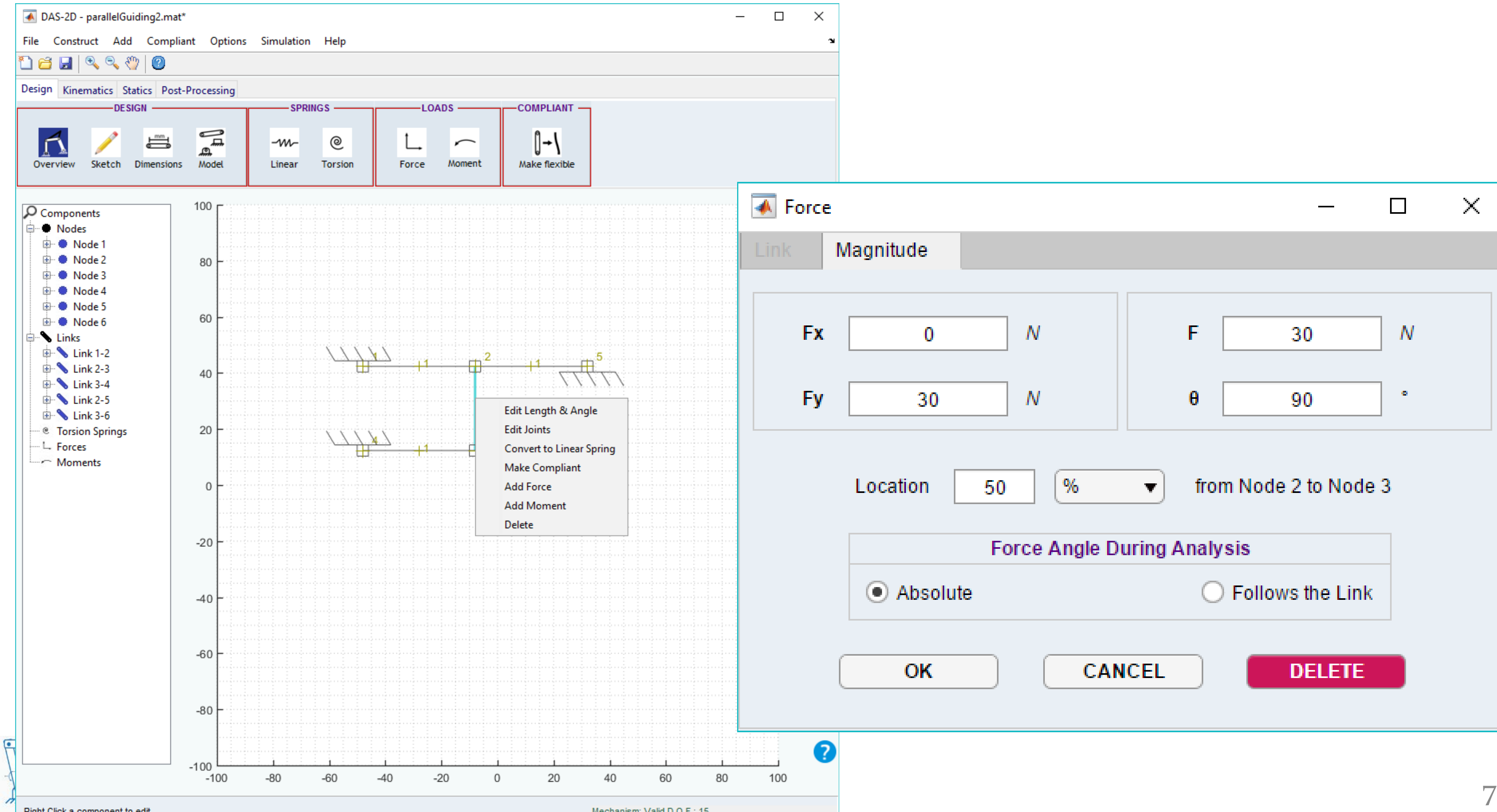
Step 4. Go to Overview Module. Right Click to all links except the middle one and select make compliant. Choose BCM and hit continue. In-Plane thickness is 1 mm and out-plane thickness is 10 mm. Set E to 2.3 Gpa. Number of segments can be 2. Hit Ok.



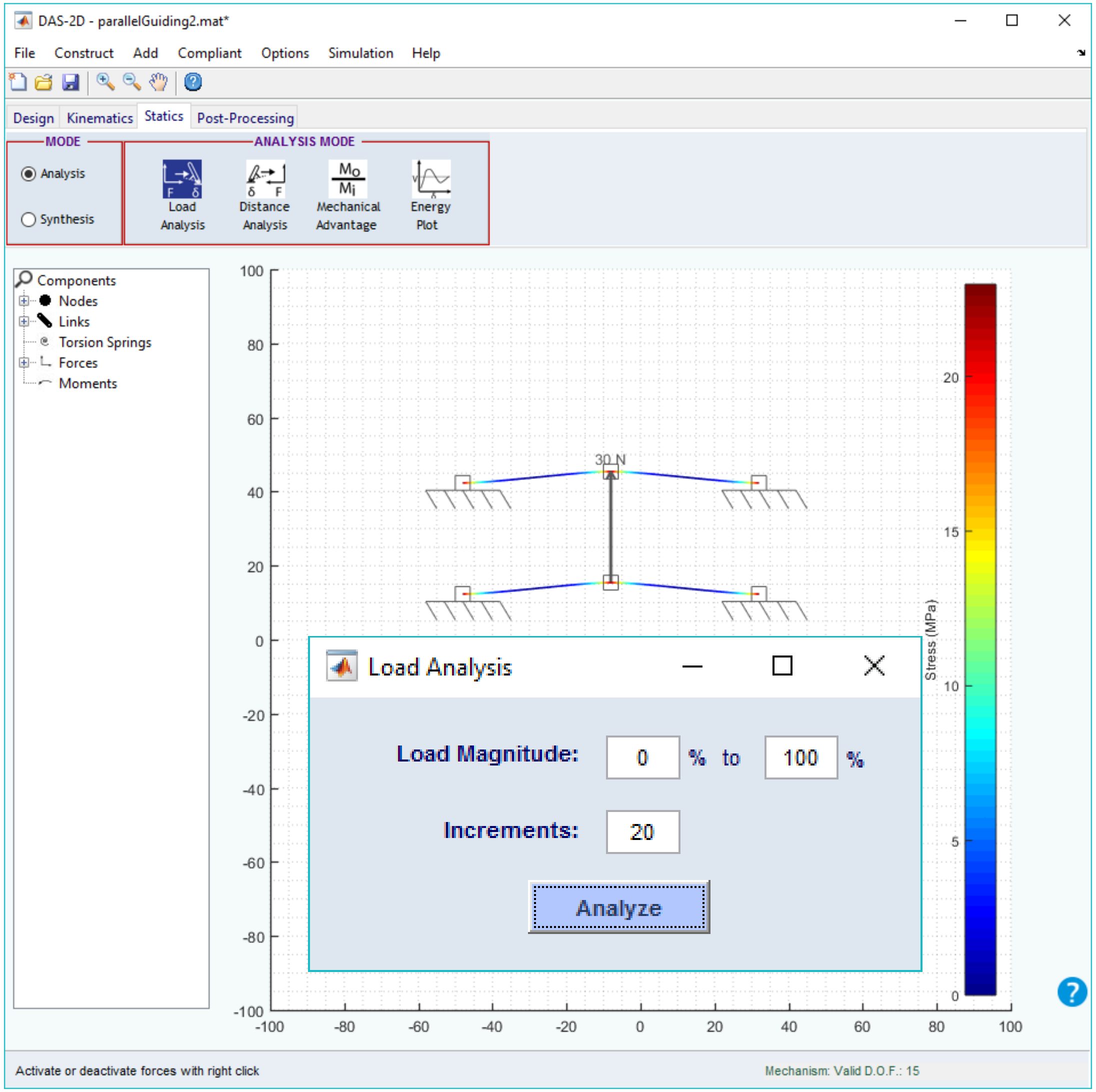


Ex.2 Parallelogram Mechanism

Step 5. Right Click to middle link and select add force. Fx=0 and Fy=30 N. Location=50%. Hit Ok.

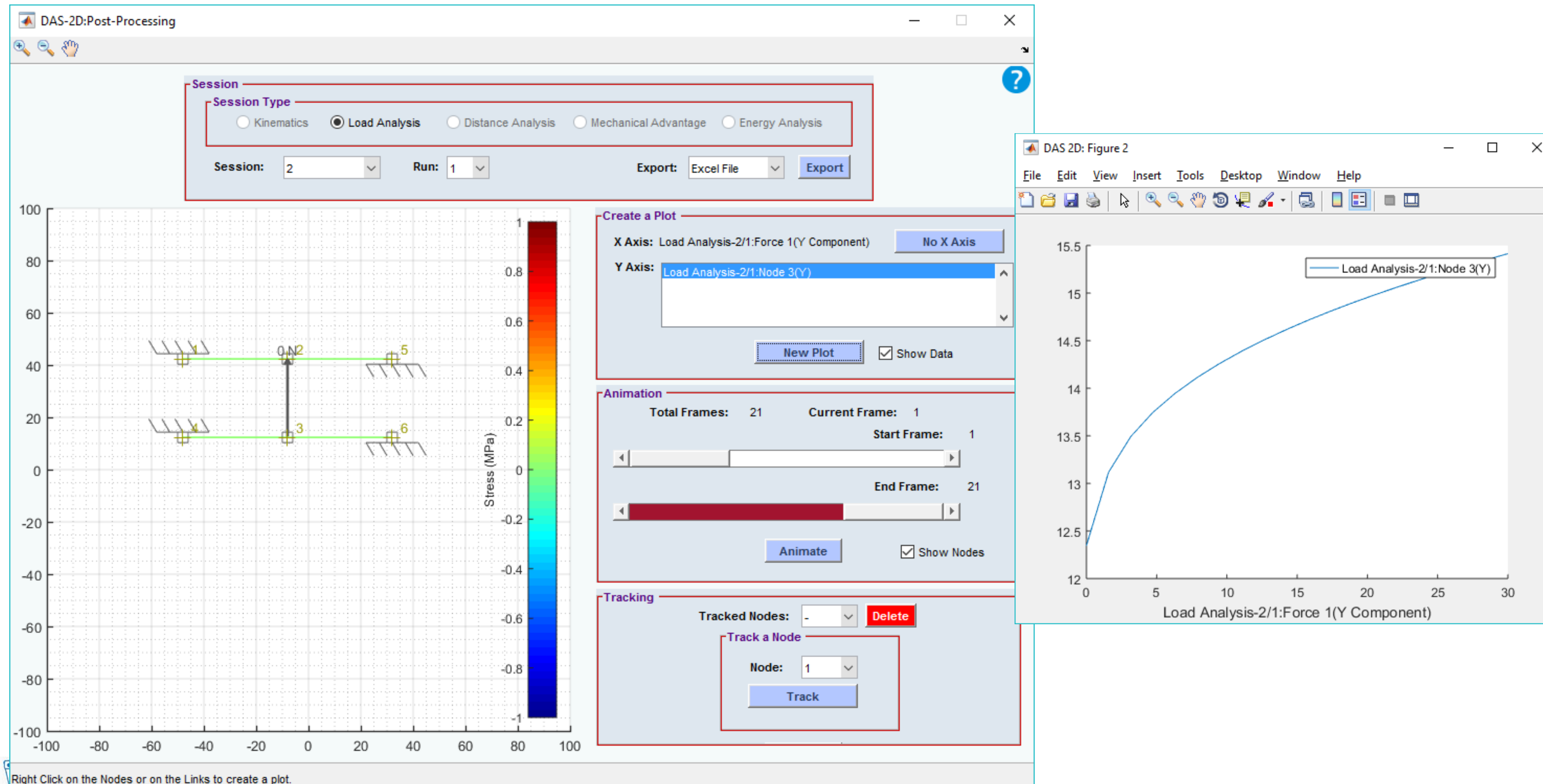


Step 6. Go to Statics-> Load Analysis. Set increments to 20.



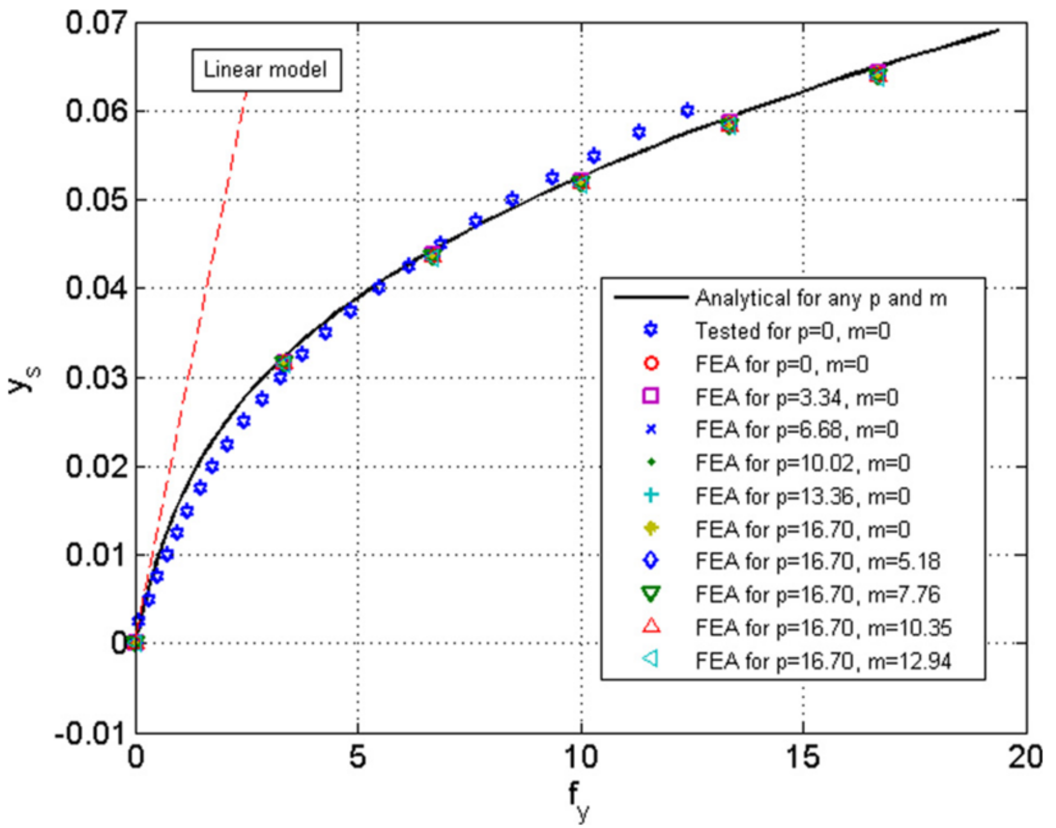
Ex.2 Parallelogram Mechanism

Step 7. Open Post-Processing. Right Click force and select Set Magnitude as the X Axis. Right Click one of the nodes on the middle link and select Add Y Coordinates to Y Axis. Hit New Plot.



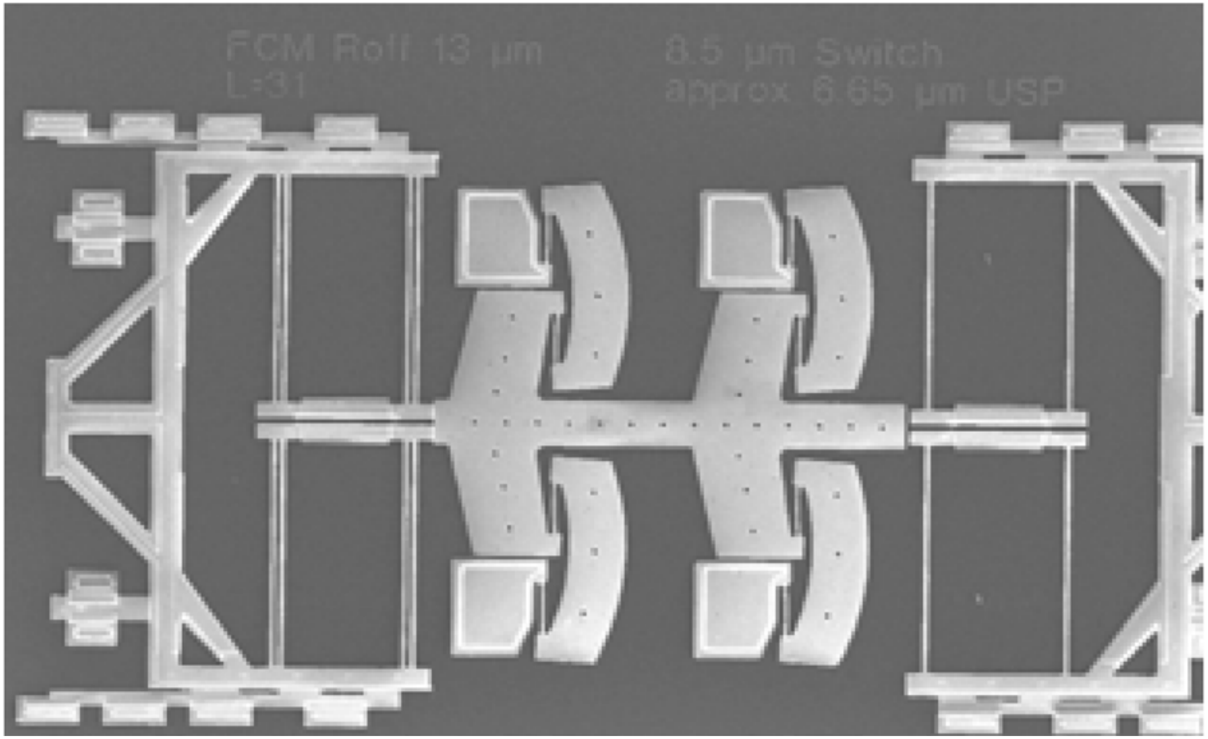
Ex.2 Parallelogram Mechanism

Observe that multiple BCM can capture increasing stiffness with Y.



Hao, Guangbo, and Haiyang Li. "Nonlinear Analytical Modeling and Characteristic Analysis of a Class of Compound Multibeam Parallelogram Mechanisms." *Journal of Mechanisms and Robotics* 7, no. 4 (November 1, 2015): 041016–041016. doi:10.1115/1.4029556.

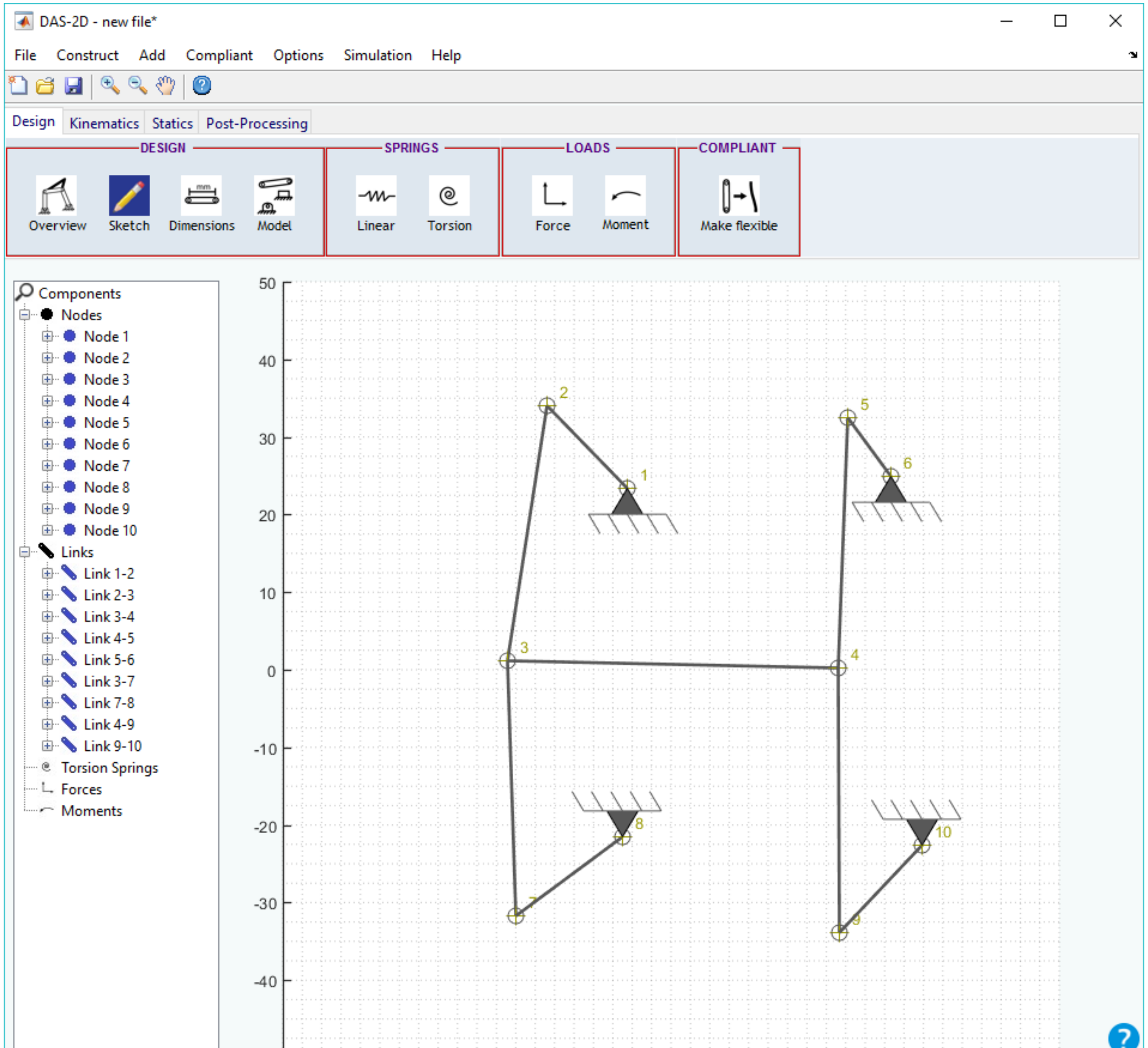
Start by setting workspace size 50 by 50 from Options-> Workspace



Masters, Nathan D., and Larry L. Howell. “A Three Degree-of-Freedom Model for Self-Retracting Fully Compliant Bistable Micromechanisms.” *Journal of Mechanical Design* 127, no. 4 (June 27, 2005): 739–44. doi:10.1115/1.1828463.

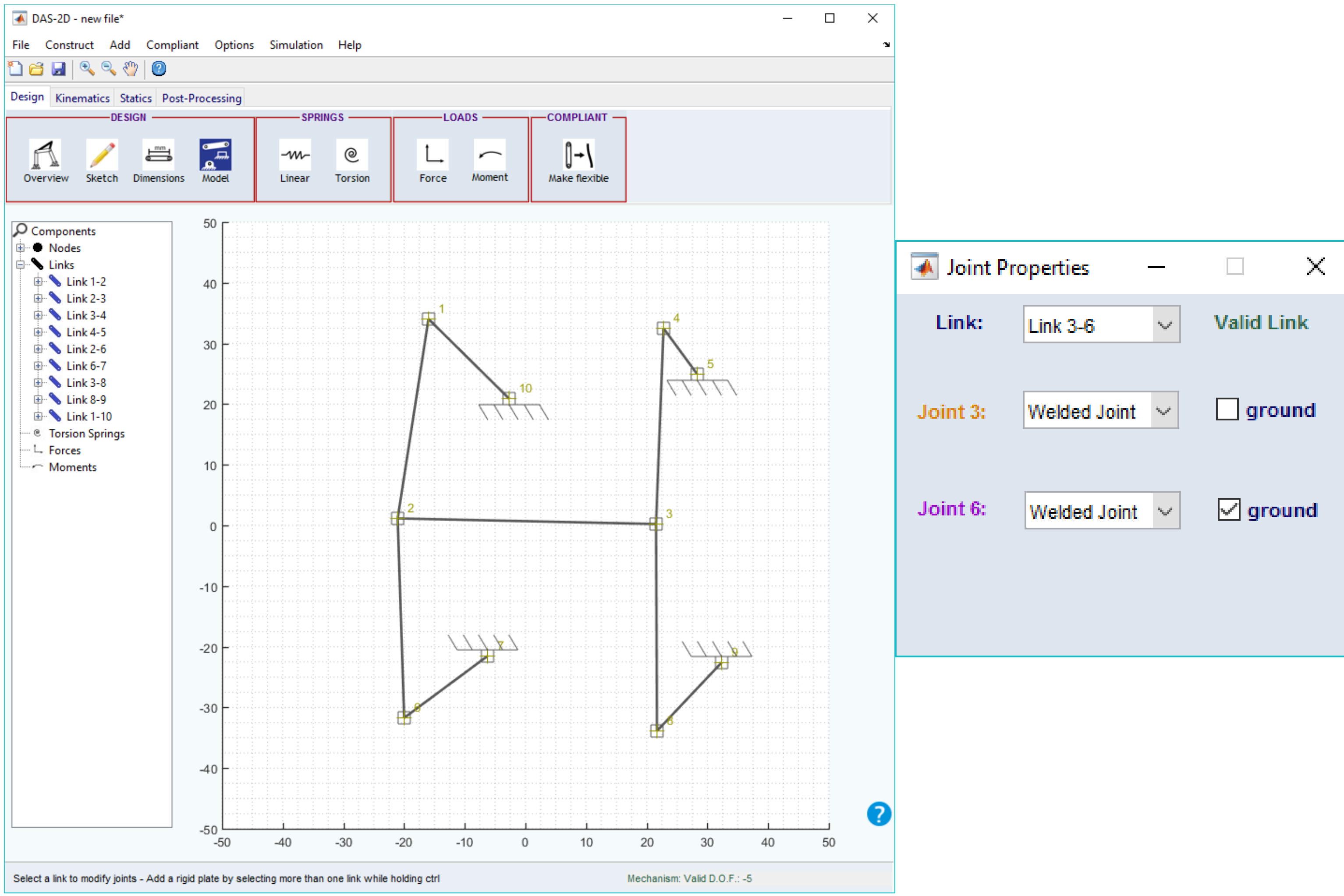
Ex.3 Bistable Switch Mechanism

Step 1. Sketch the mechanism as shown below



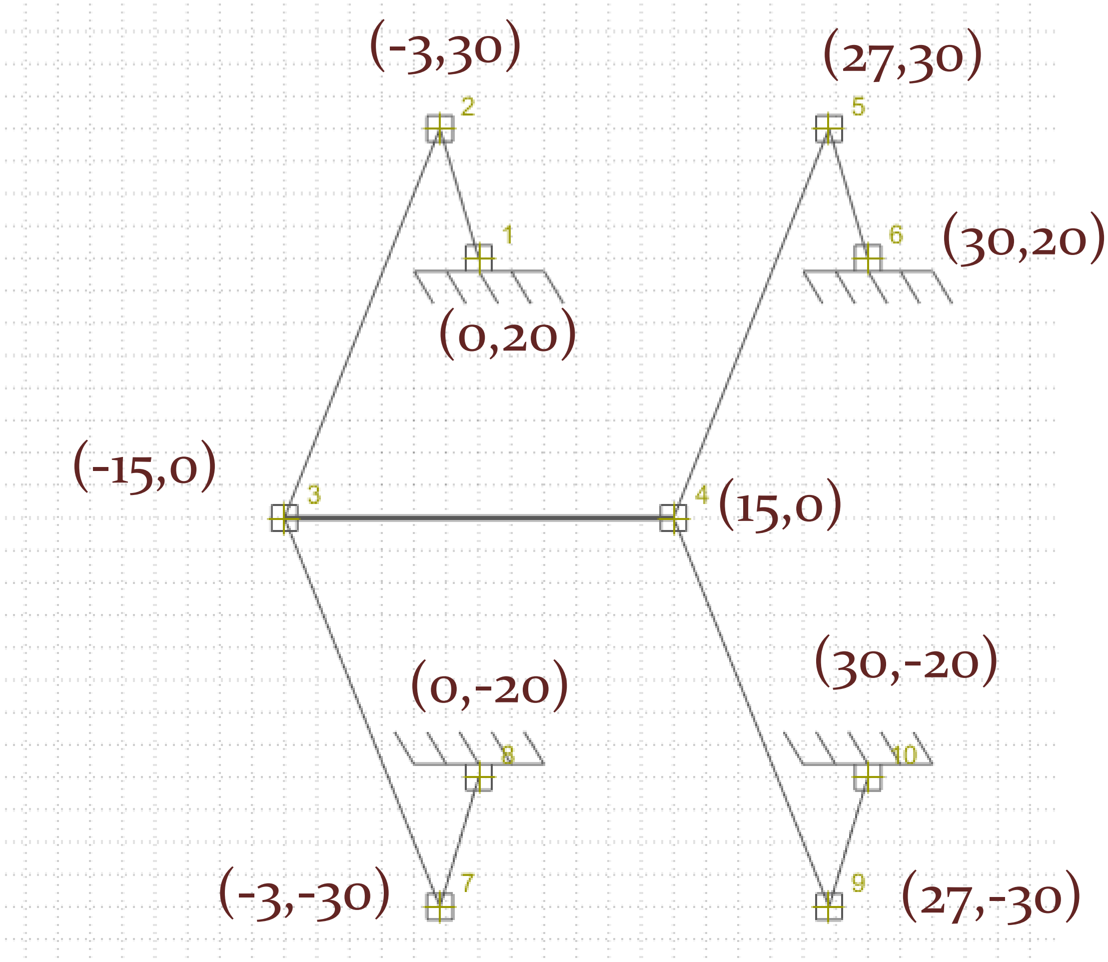
Ex.3 Bistable Switch Mechanism

Step 2. Go to Model Module and click links one by one and set joints as welded joint.



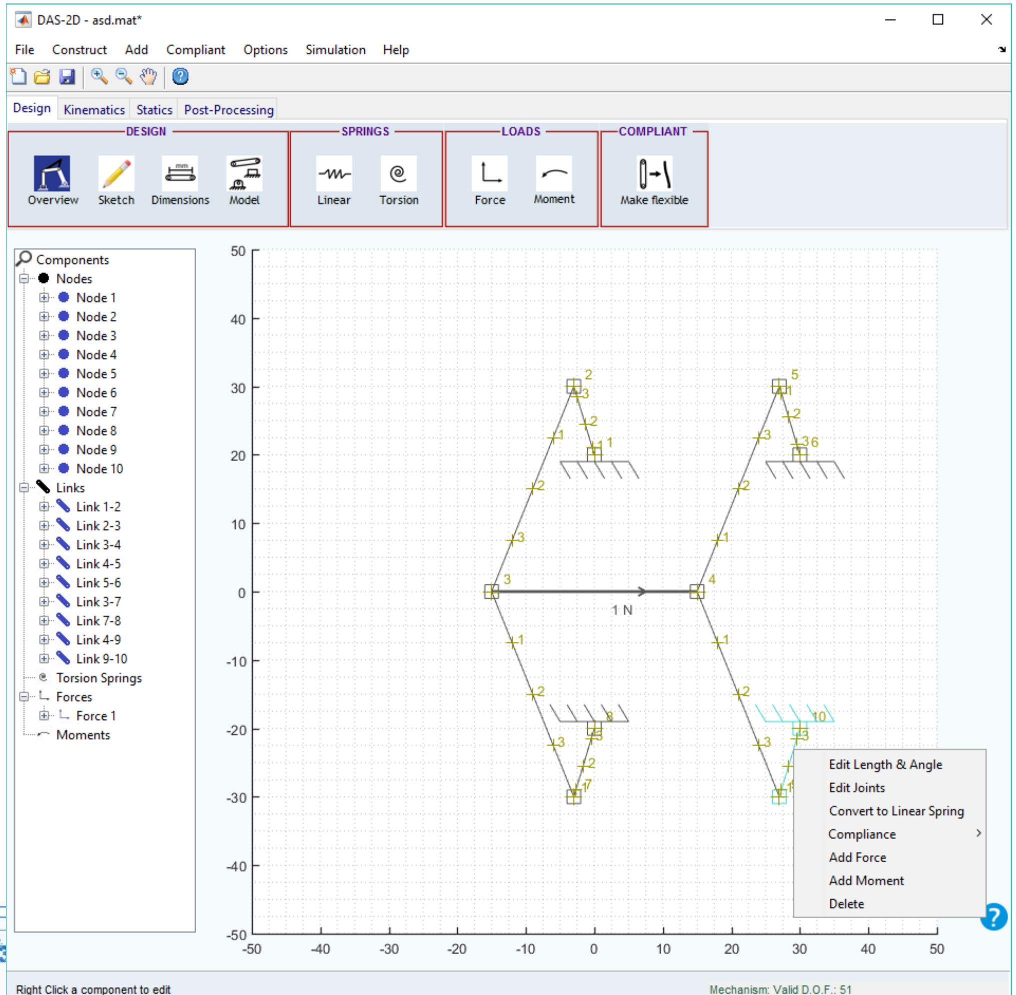
Ex.3 Bistable Switch Mechanism

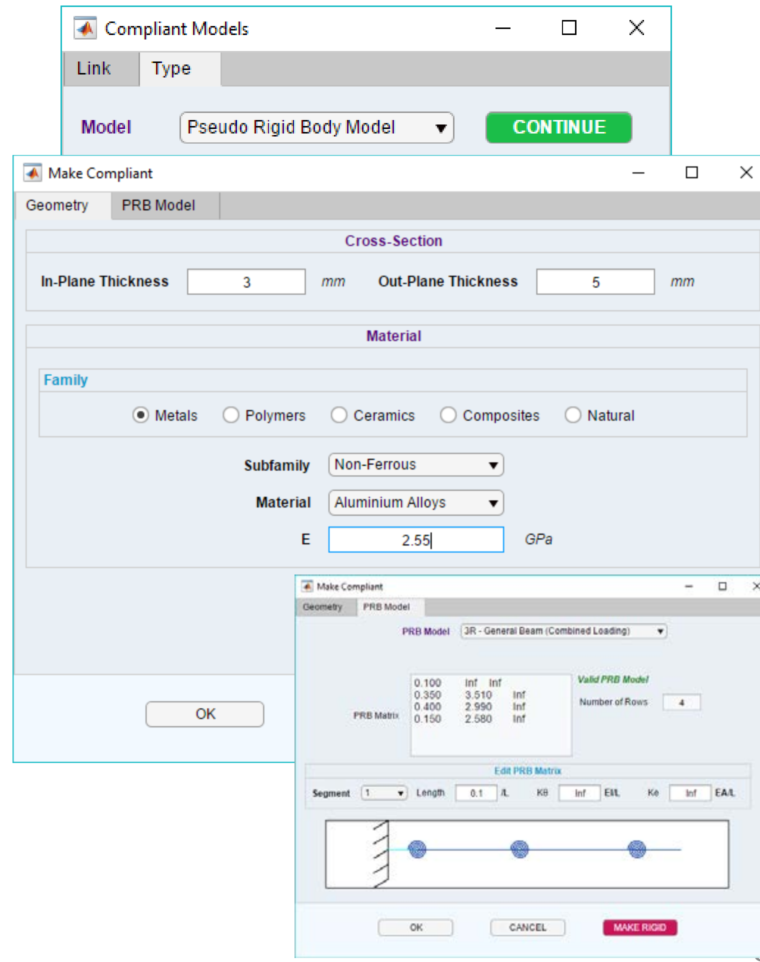
Step 3. Either go to Overview mode and right click nodes and select Edit or go to dimensions and click nodes one by one. Enter the coordinates below.



Ex.3 Bistable Switch Mechanism

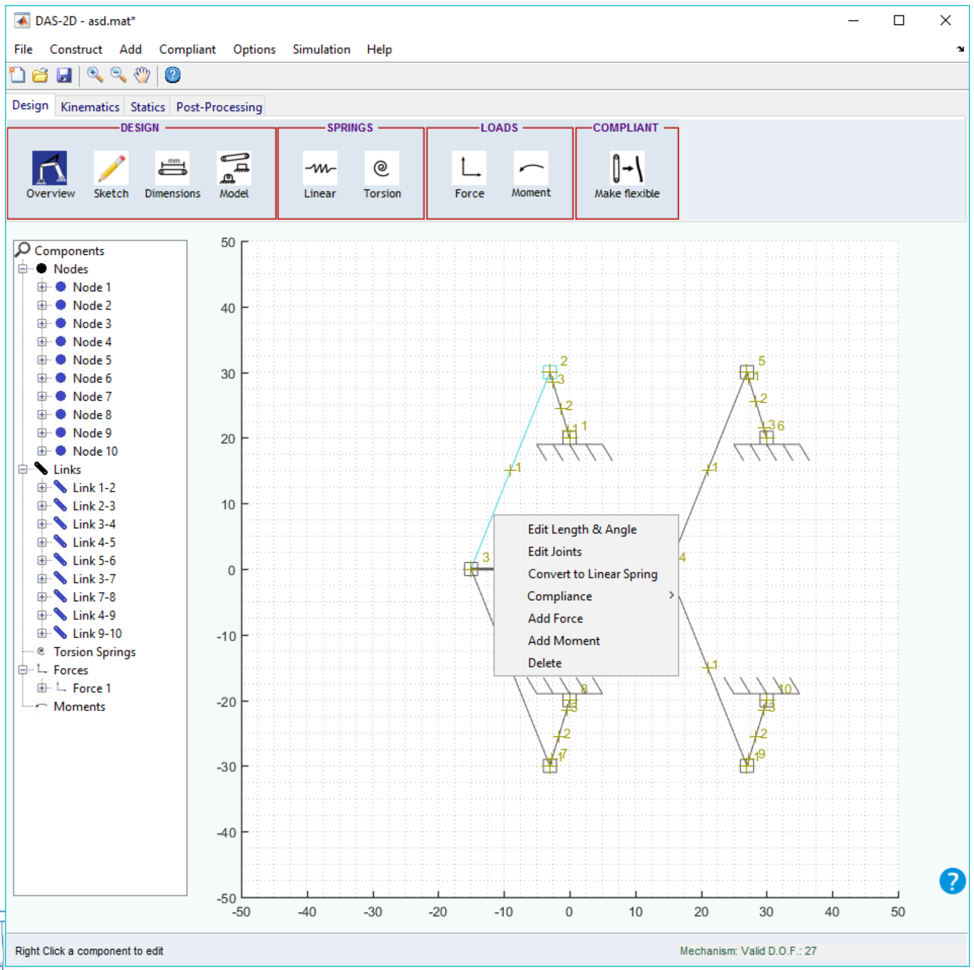
Step 4. Go to Overview Module. Right Click short links and select make compliant. Choose PRB and hit continue. PRB Model can be chosen as PRB-3R. In-Plane thickness is 3 mm and out-of-plane thickness is 5 mm. Set E to 2.55 Gpa. Hit Ok.

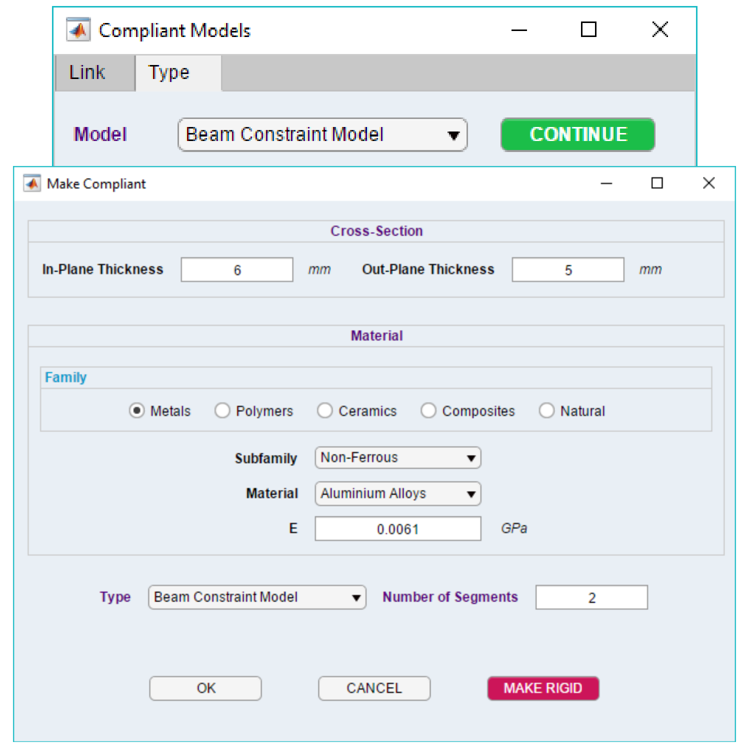




Ex.3 Bistable Switch Mechanism

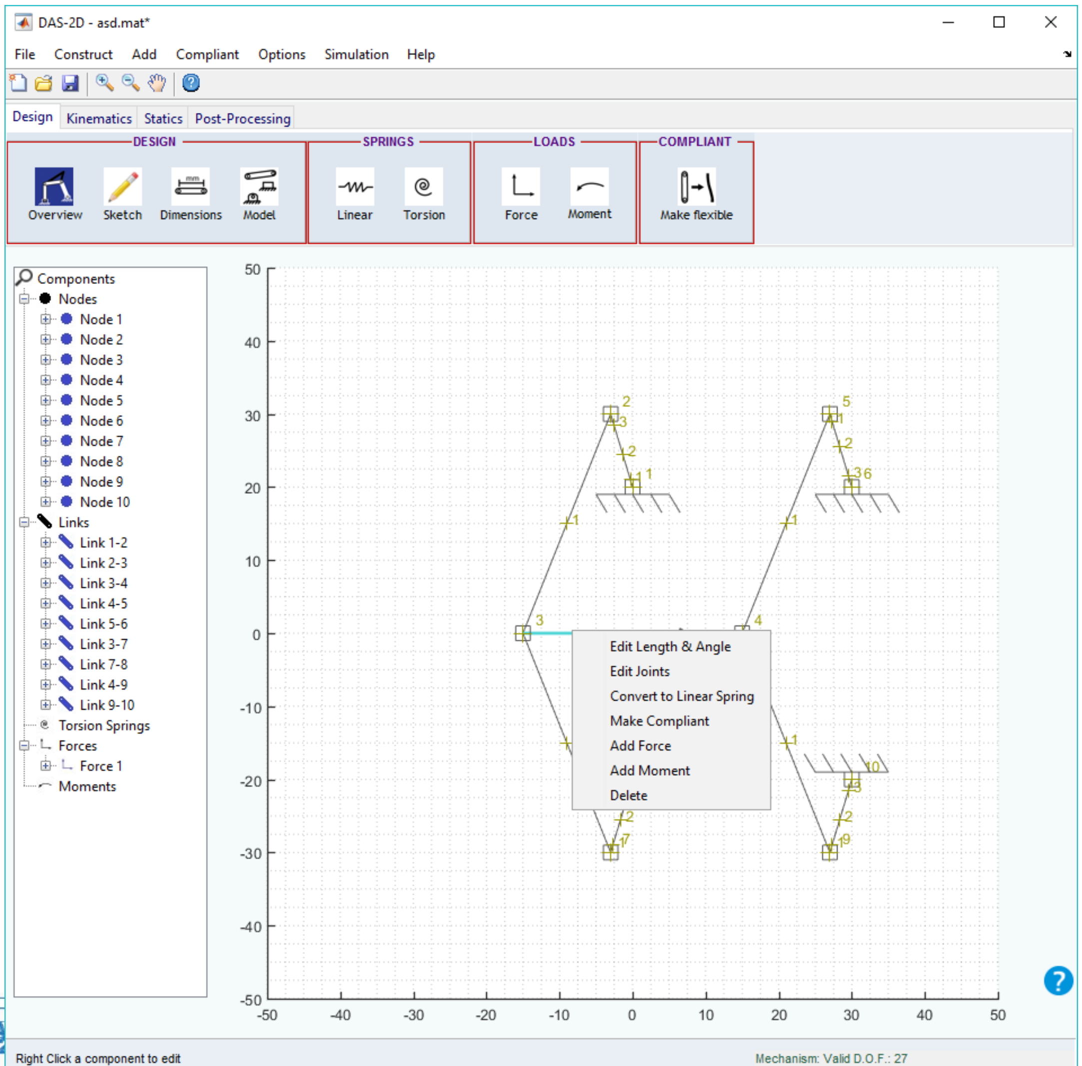
Step 5. Stay at Overview Module. Right Click long links and select make compliant. Choose BCM and hit continue. Number of segments can be 2. In-Plane thickness is 6 mm and out-of-plane thickness is 5 mm. Set E to 0.0061 Gpa. Hit Ok.

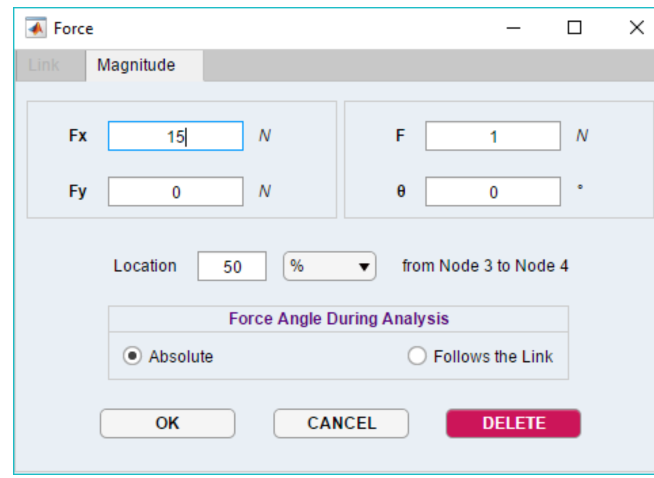




Ex.3 Bistable Switch Mechanism

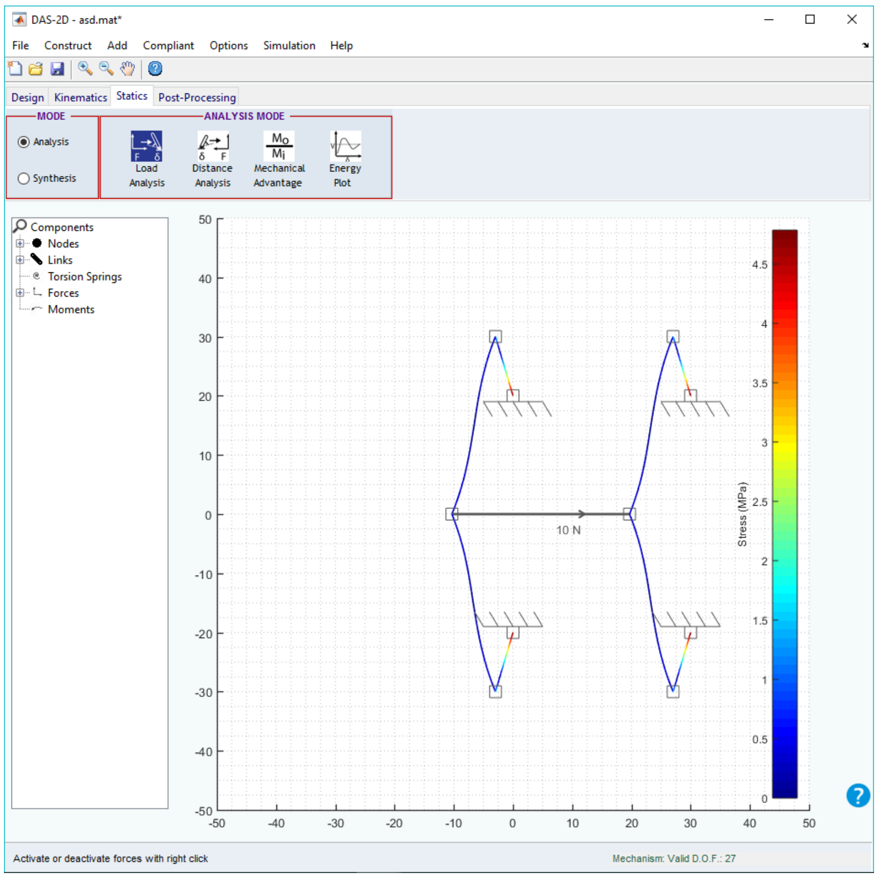
Step 6. Right Click to middle link and select add force. Fx=15 N and Fy=0 N. Location=50%. Hit Ok.

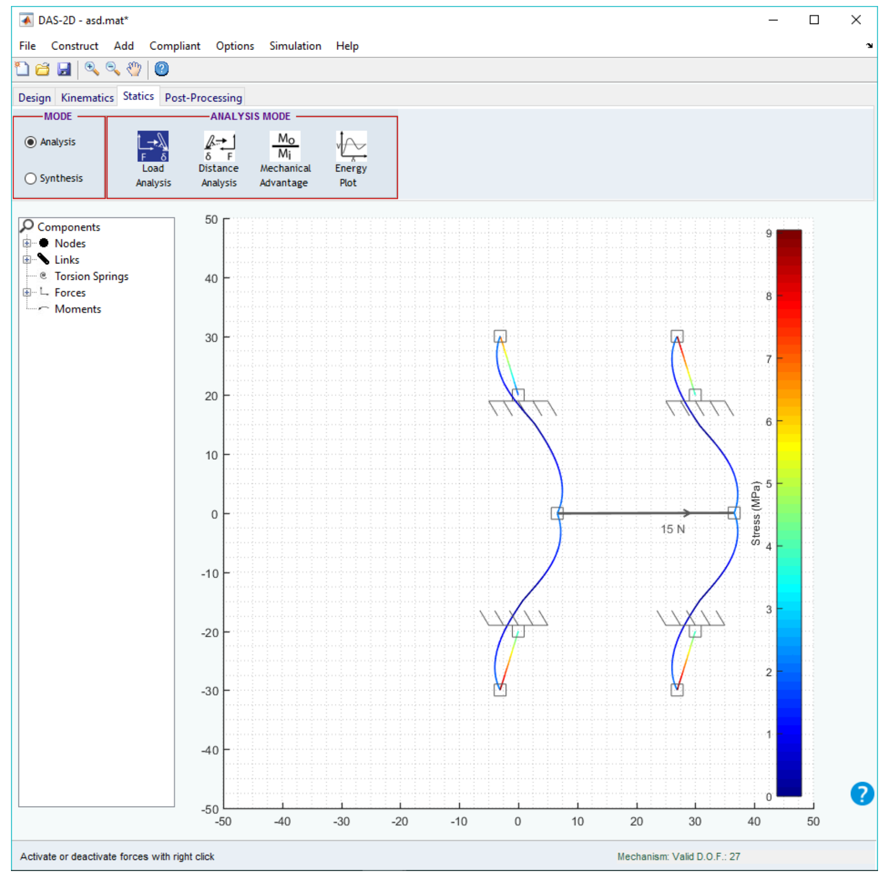




Ex.3 Bistable Switch Mechanism

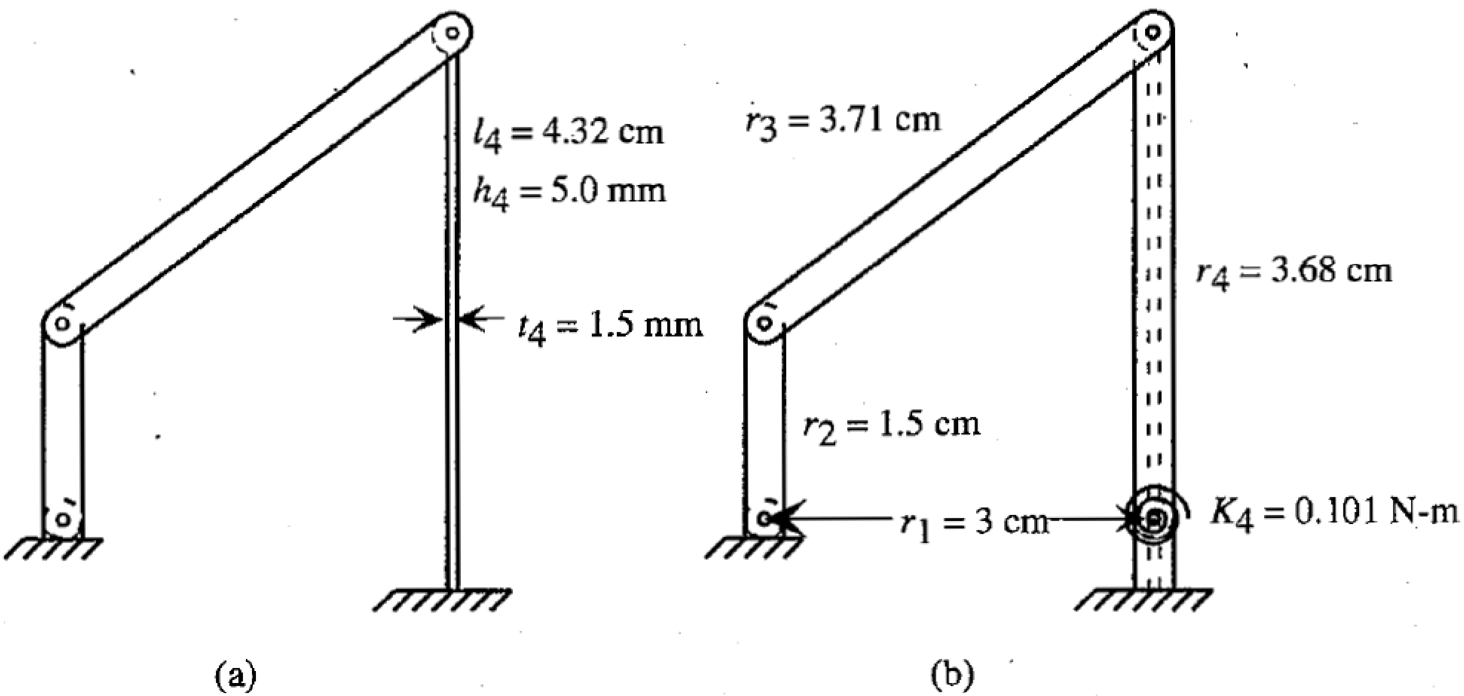
Step 7. Go to Statics-> Load Analysis. Click Ok. Observe the snapping motion after 10 N.





Ex.4 Bistable Four Bar Mechanism

Start by setting workspace size 40 by 40 from Options-> Workspace

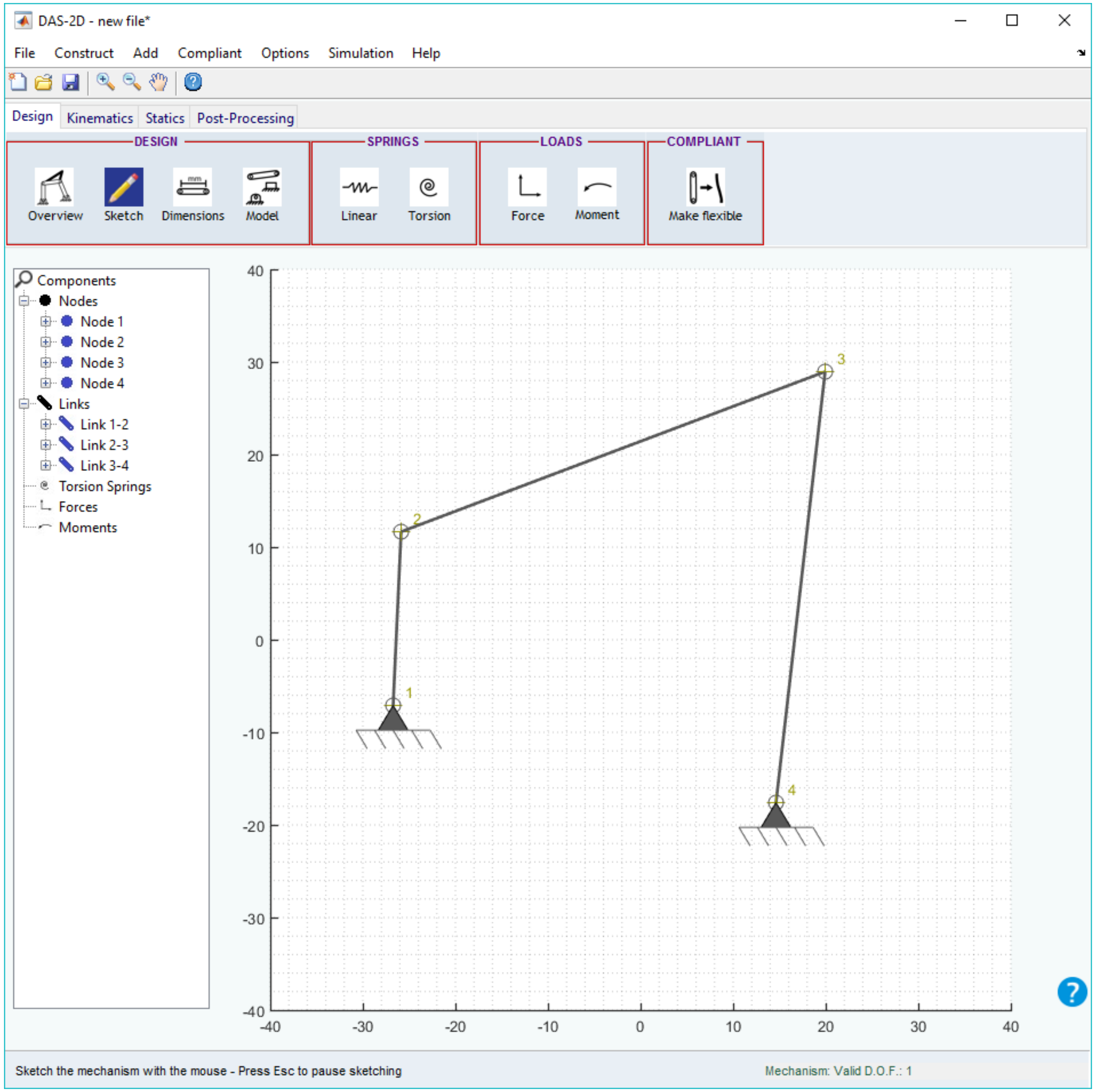


Compliant Mechanisms – Larry L. Howell
Page 363 – Figure 11.8

E=1.4GPa,
In-plane thickness t_4=1.5mm
Out-of-plane thickness 5mm

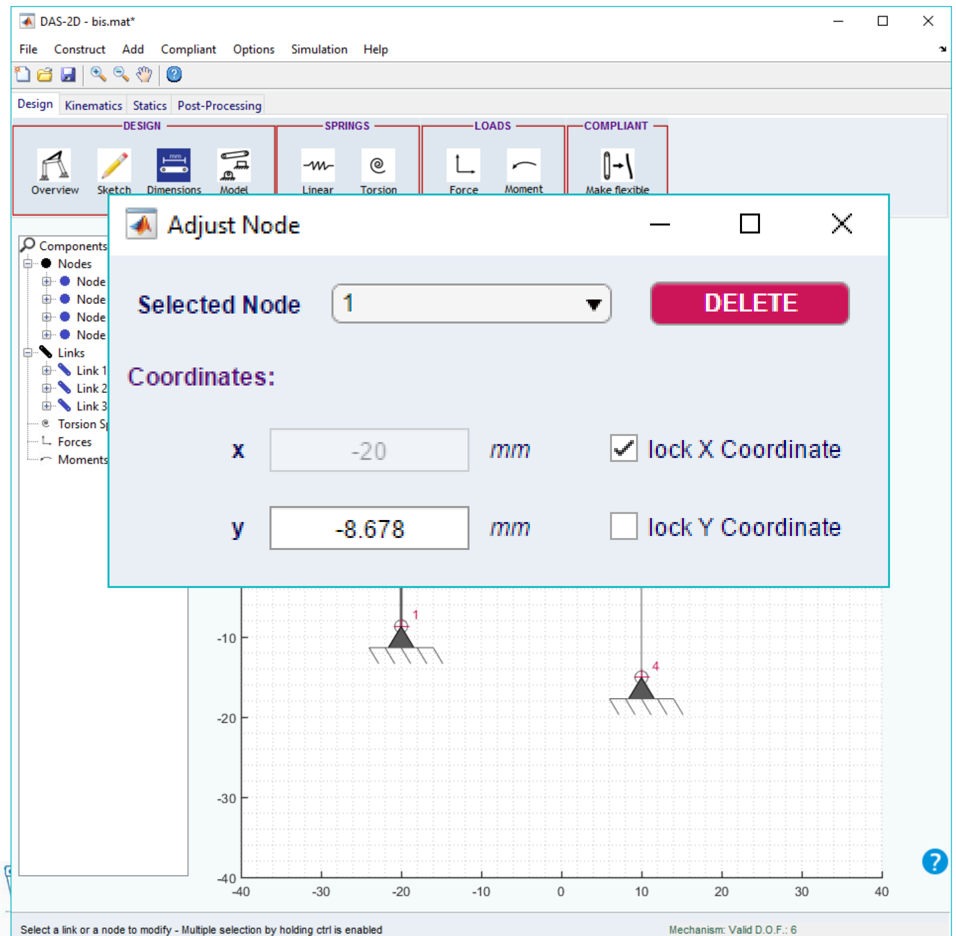
Ex.4 Bistable Four Bar Mechanism

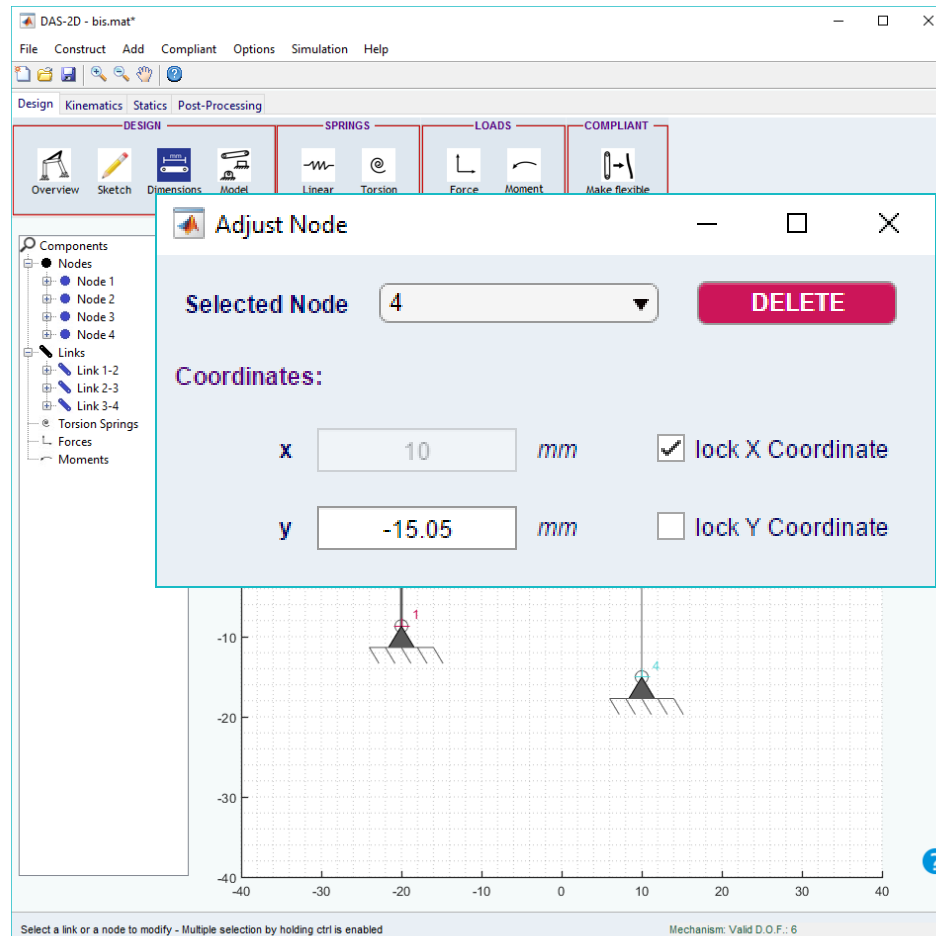
Step 1. Sketch the mechanism as shown below



Ex.4 Bistable Four Bar Mechanism

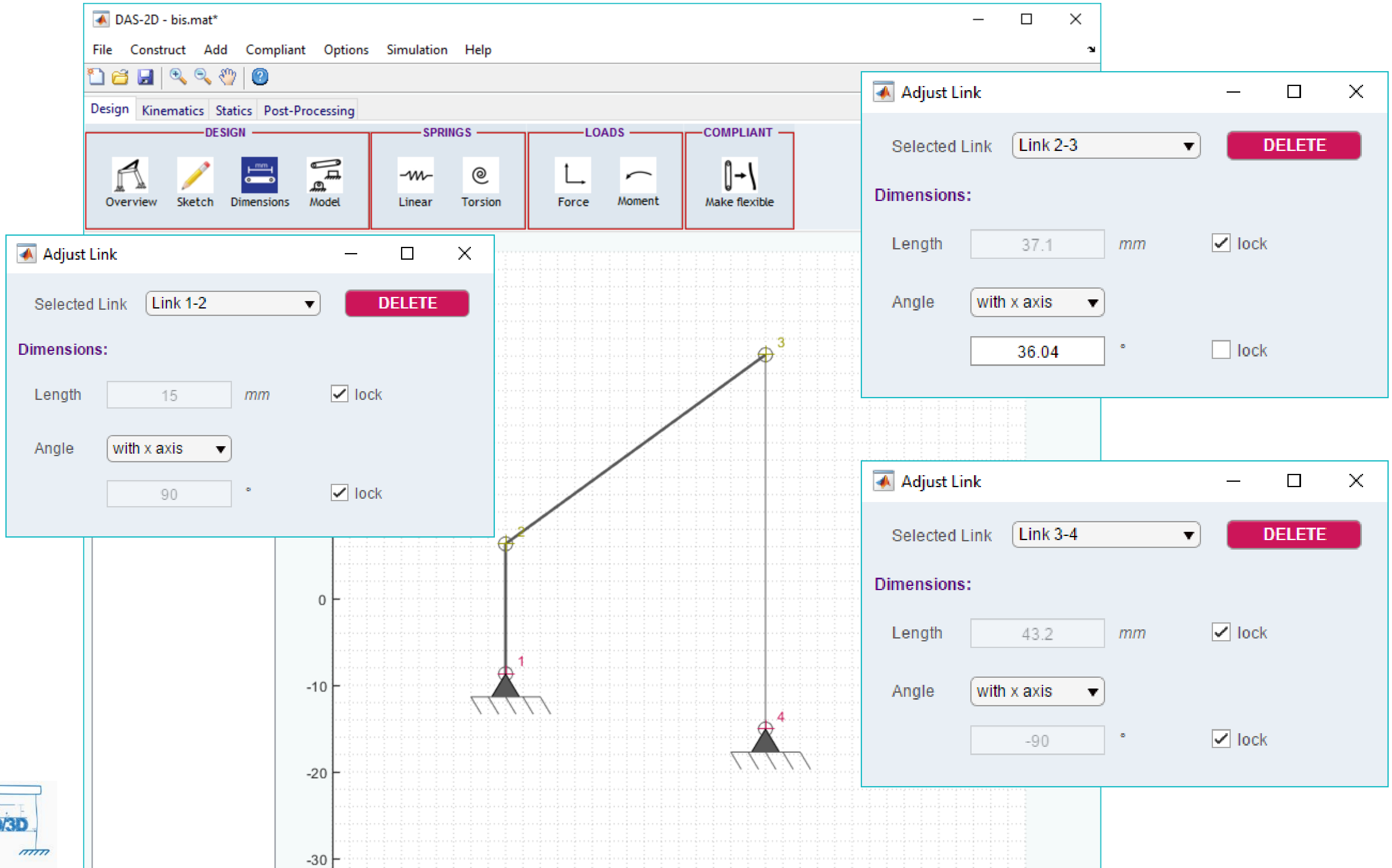
Step 2. Go to Dimensions Module and Click to Node 1. Lock x as -20 mm. Click to Node 4. Lock x as -20 mm.





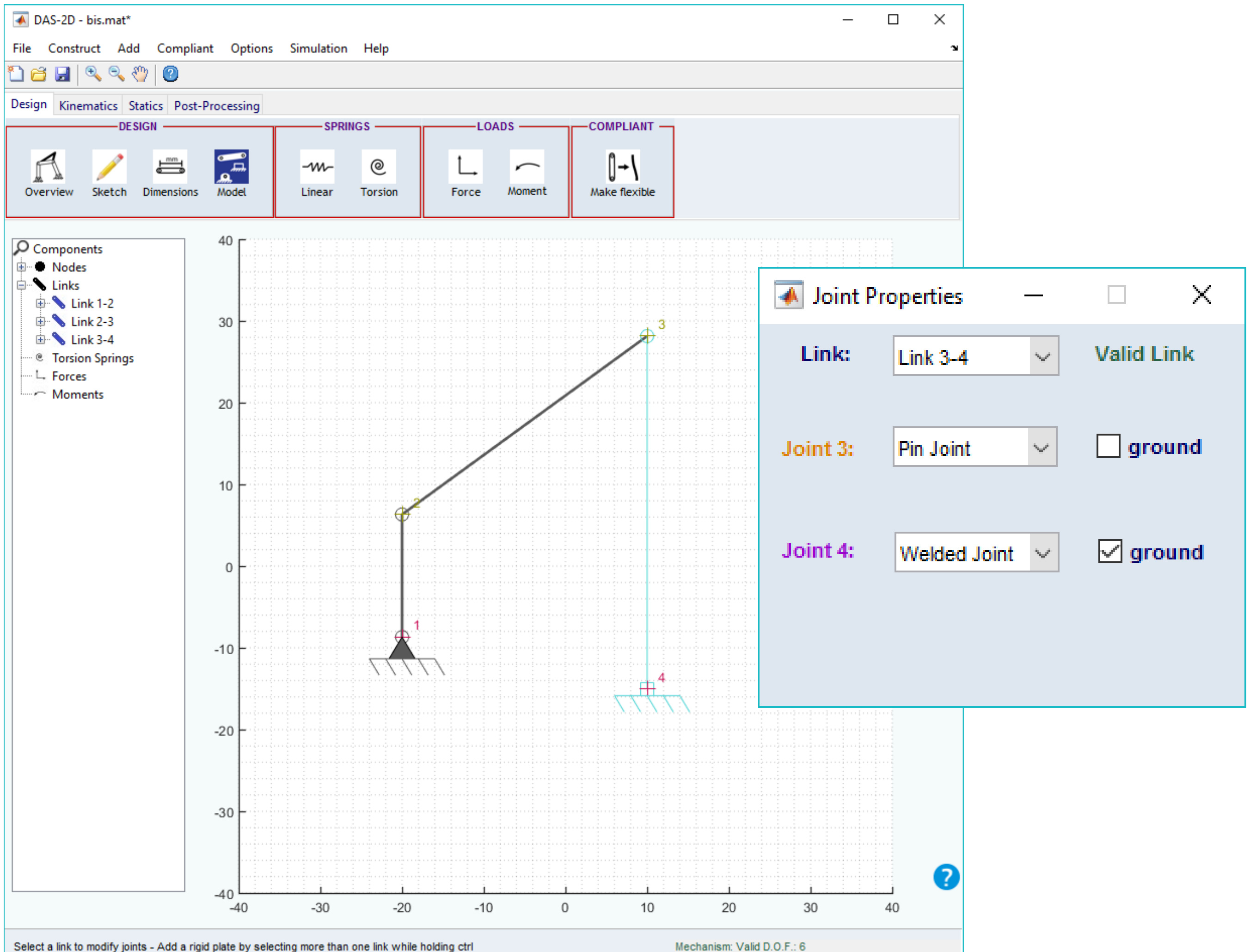
Ex.4 Bistable Four Bar Mechanism

Step 3. Click to Link 1-2: L=15 mm $\theta = 90°(-90°)$. Link 2-3: L=37.1 mm Link 3-4: L=43.2 mm $\theta = 90°(-90°)$



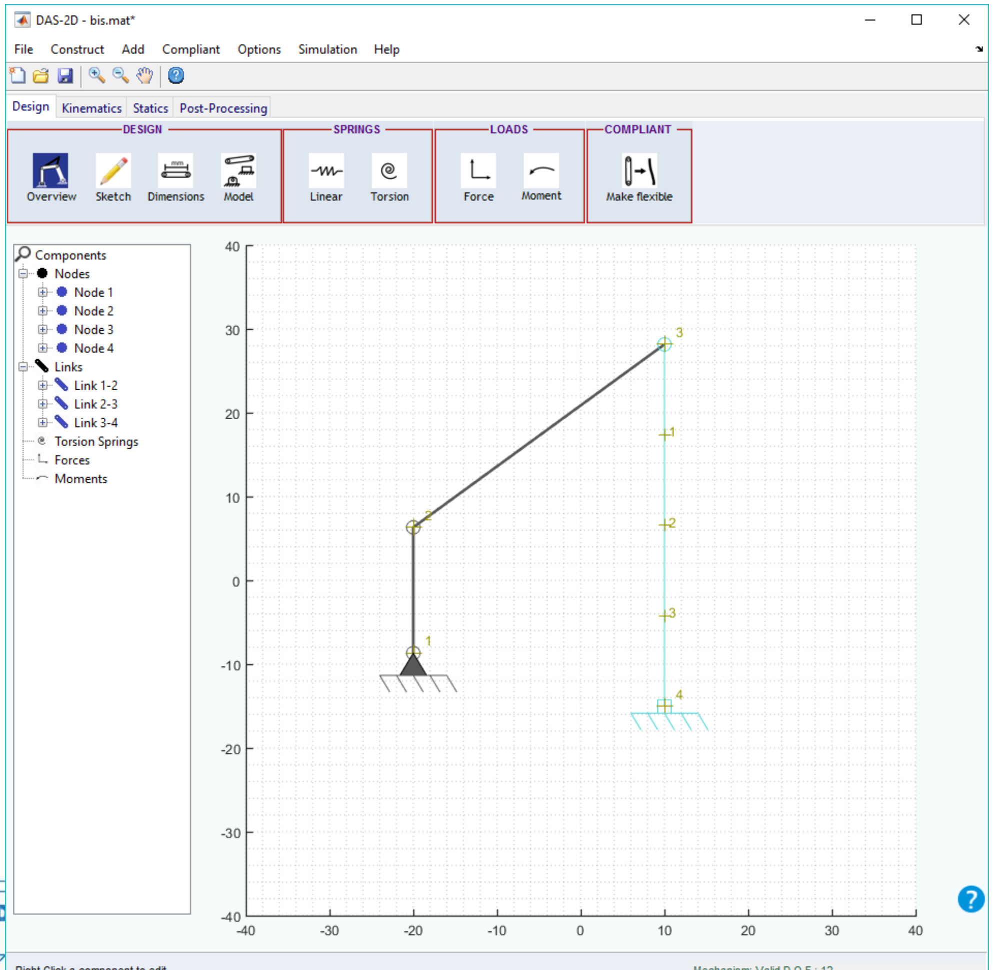
Ex.4 Bistable Four Bar Mechanism

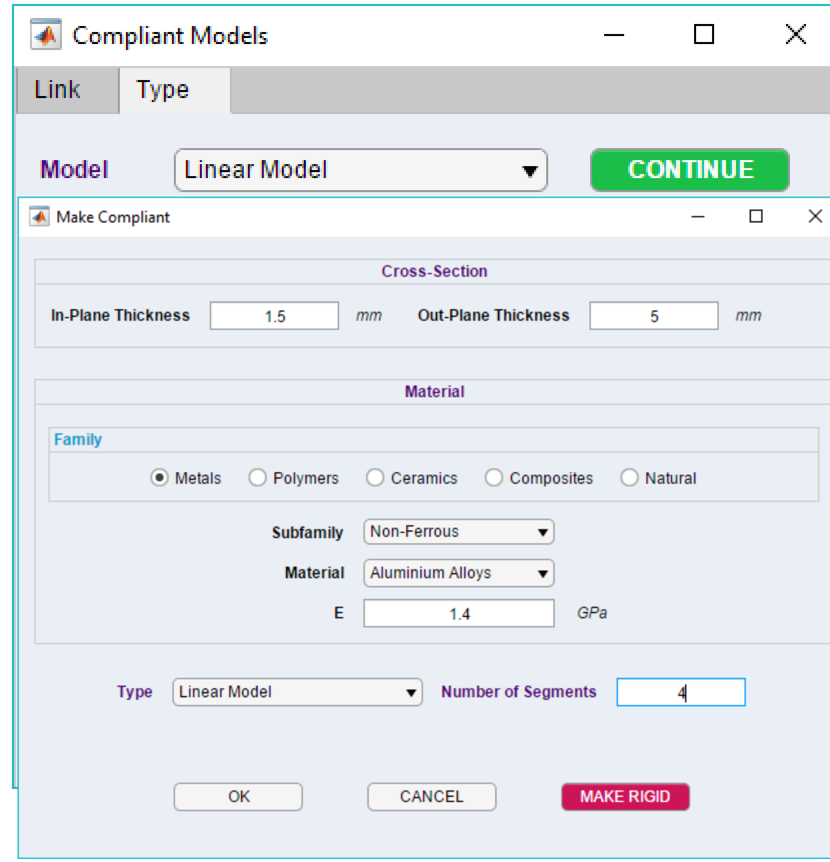
Step 4. Go to Model Module. Click Link 3-4 and set the ground node as a welded joint.



Ex.4 Bistable Four Bar Mechanism

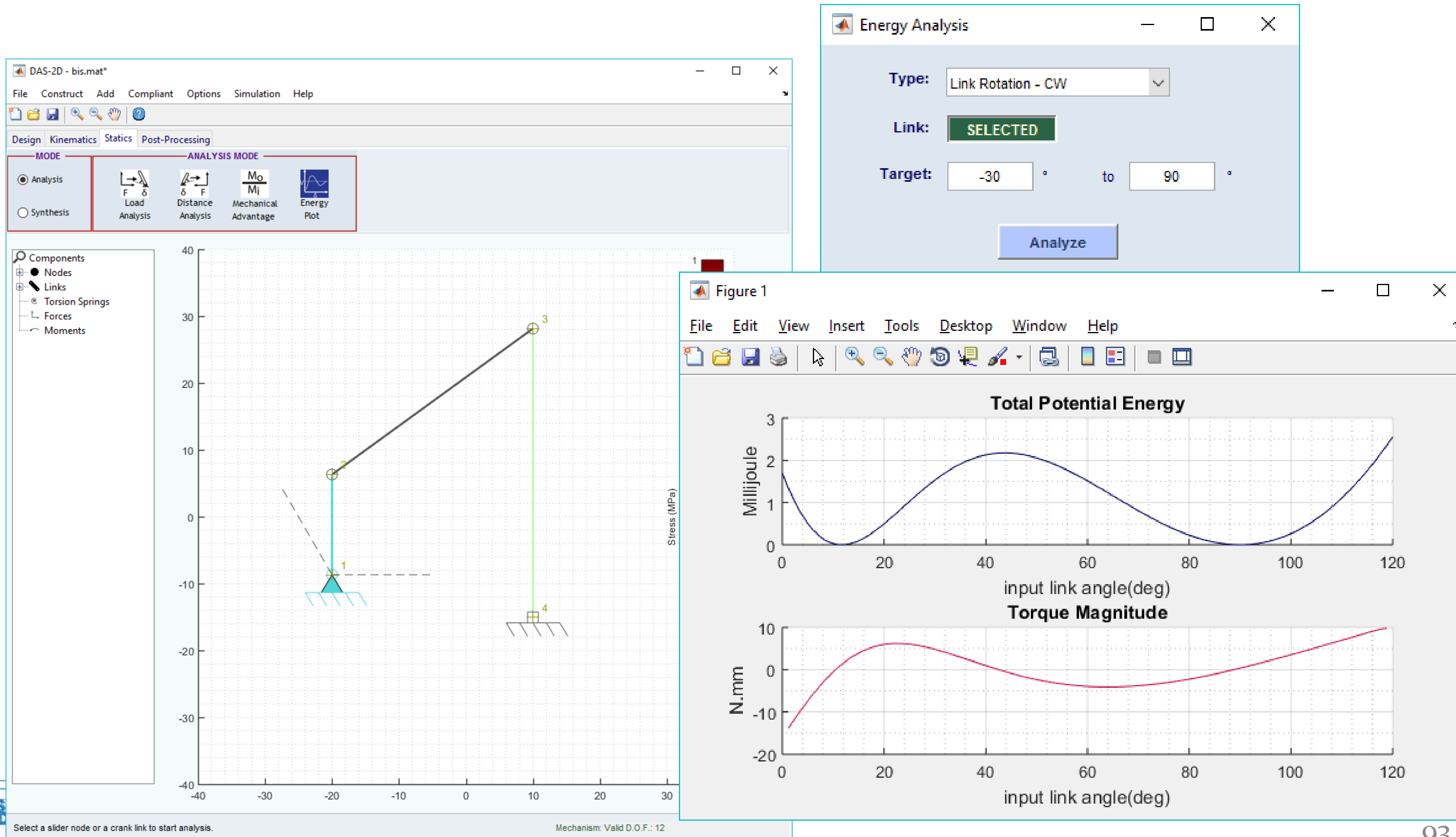
Step 5. Go to Overview Module. Right Click Link 3-4 and Make Compliant. Choose Linear Model. In Thickness = 1.5 mm
Out of Thickness =5 mm. E= 1.4 Gpa . Choose 4 Segments. Hit Ok.



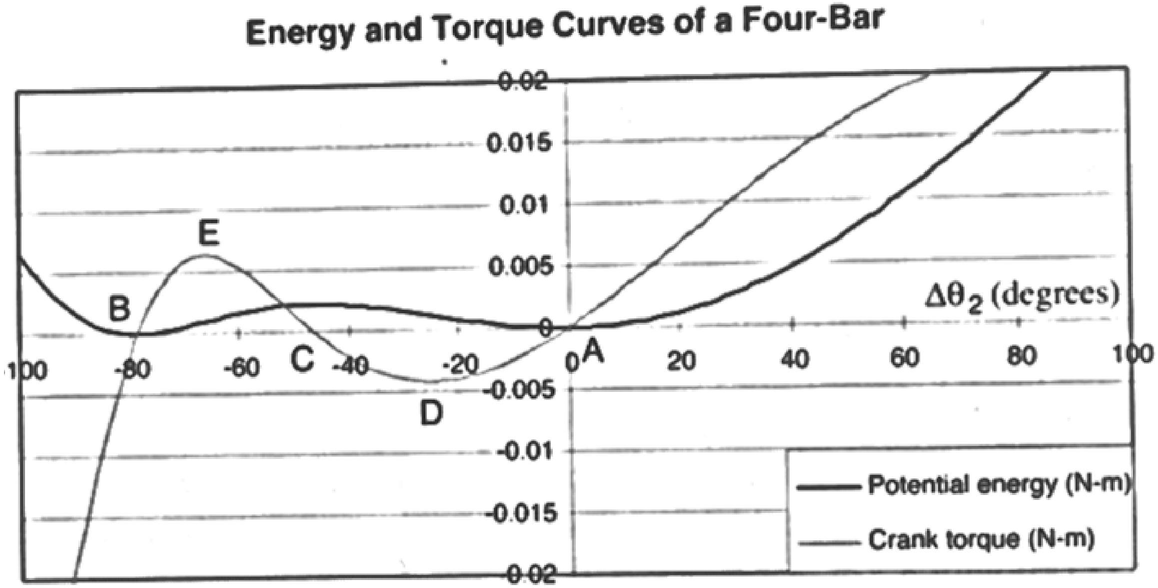


Ex.4 Bistable Four Bar Mechanism

Step 6. Go to Statics-> Energy Plot. Click the small Crank. Set target as -30° to 90 °. Click Analyze.

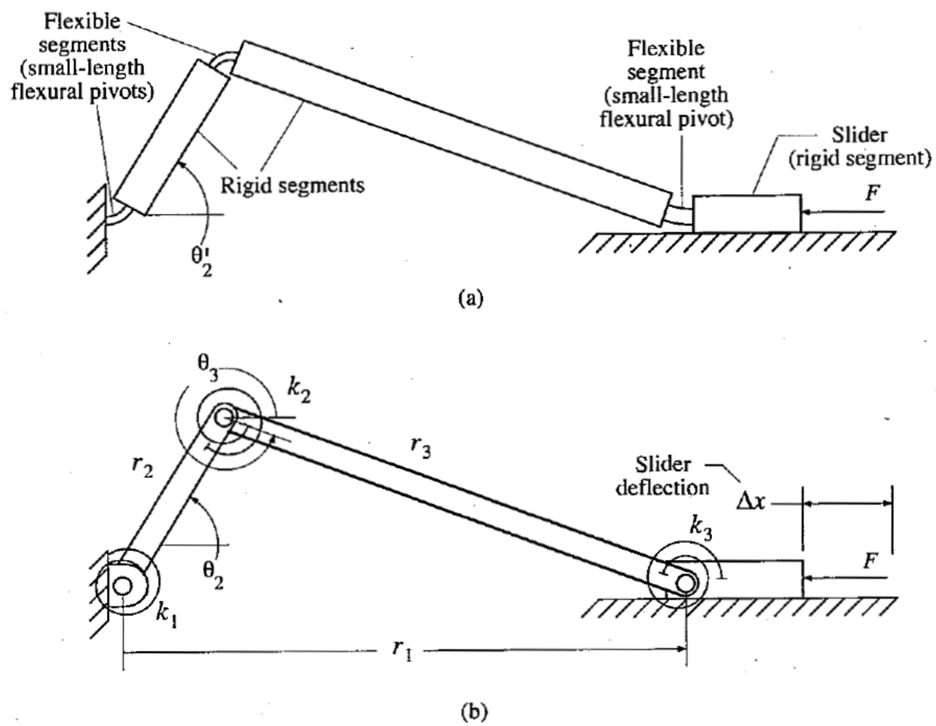


Compare your curve with curve below.



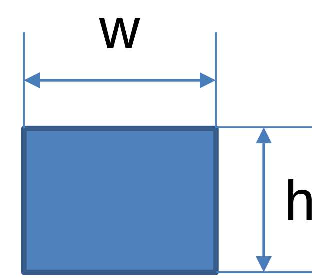
Compliant Mechanisms – Larry L. Howell
Page 364 – Figure 11.9

Start by setting workspace size 200 by 200 from Options-> Workspace



Compliant Mechanisms – Larry L. Howell
Figure 10.1 Page 338

r_2=40.2mm
r_3=107.1mm
E=1665 Mpa
h_2=h_1=0.76mm
h_3=1.78mm
l=5.08mm

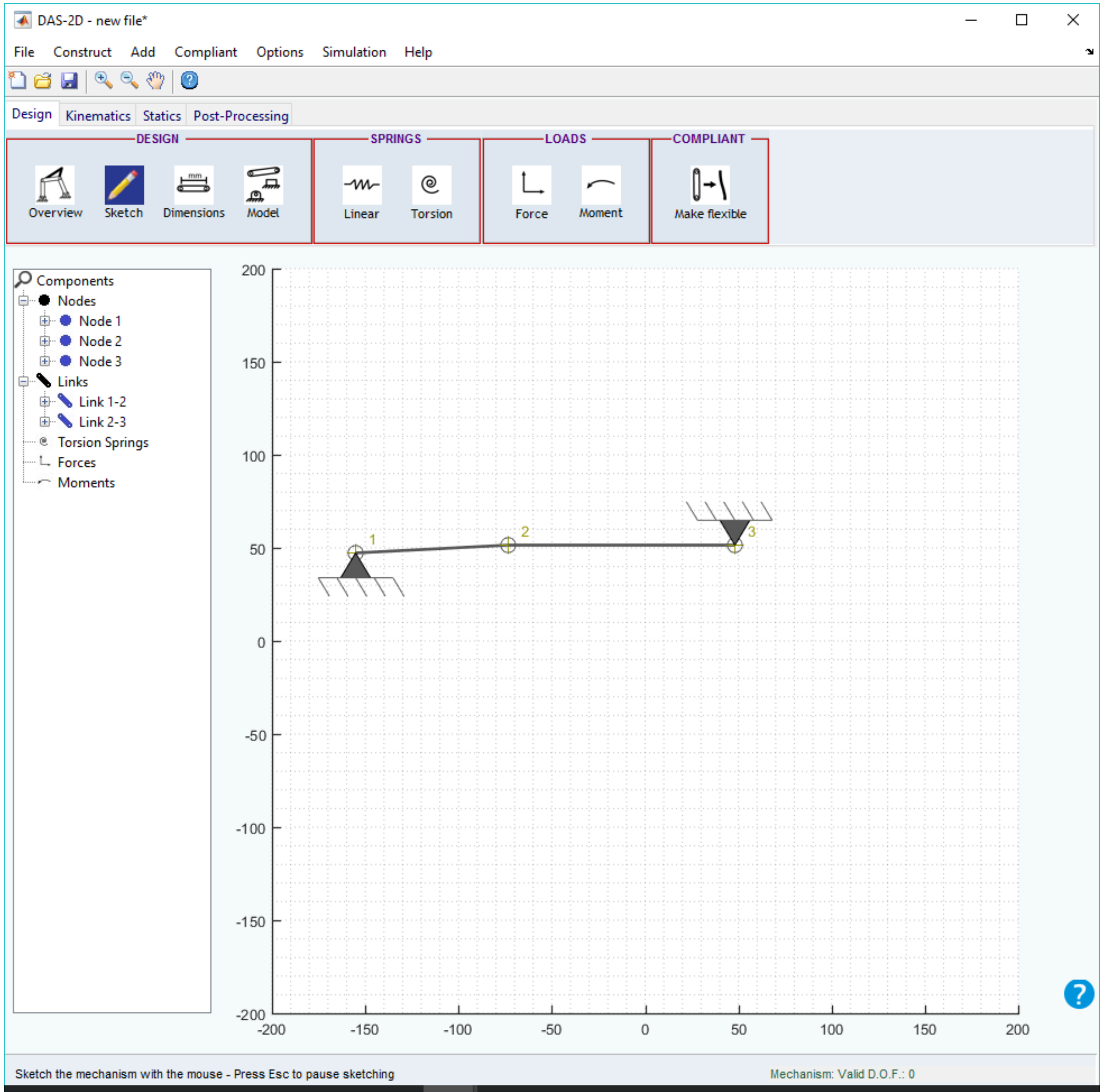


Rectangular cross-section

$$k_1 = \frac{EI}{l} = 151 \ \frac{\text{N-mm}}{\text{rad}} \quad \left(1.35 \ \frac{\text{in.-lb}}{\text{rad}}\right)$$

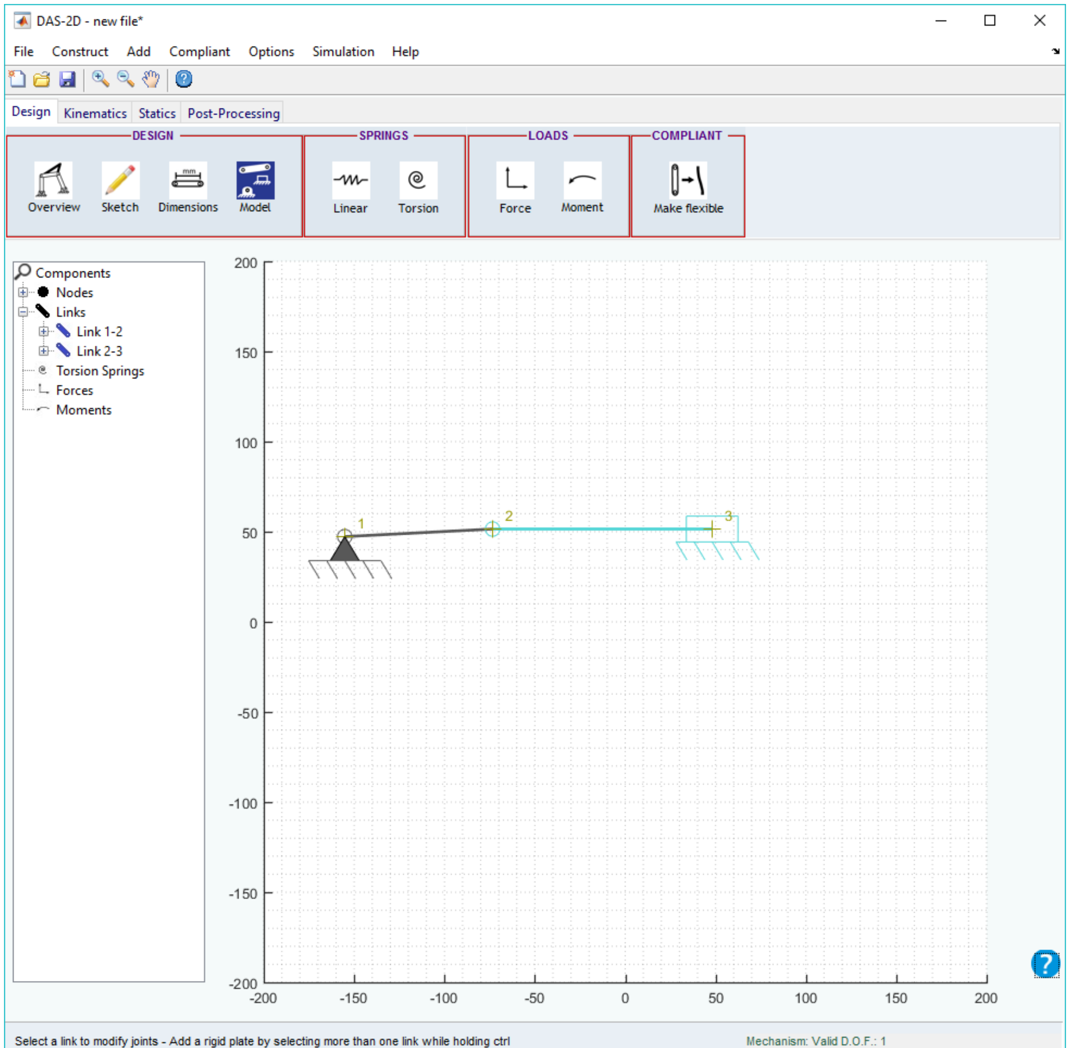
Ex.5 Constant Force Mechanism

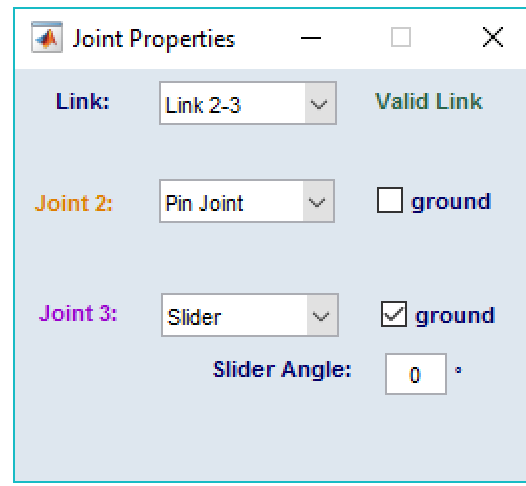
Step 1. Sketch the mechanism as shown below



Ex.5 Constant Force Mechanism

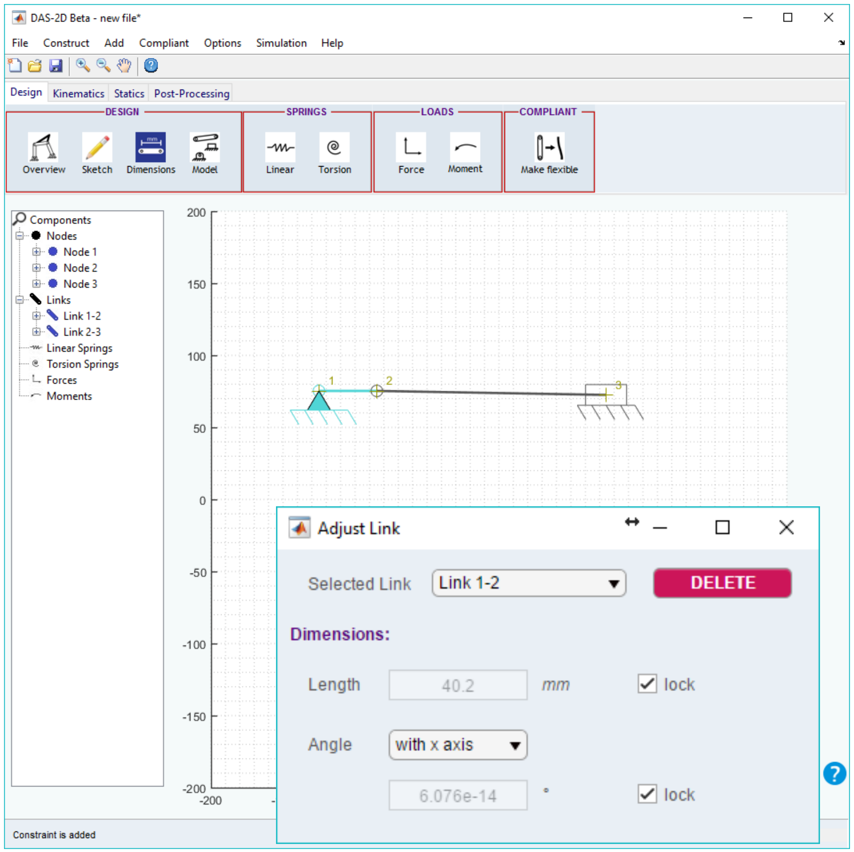
Step 2. Go to Model Module and add a slider.

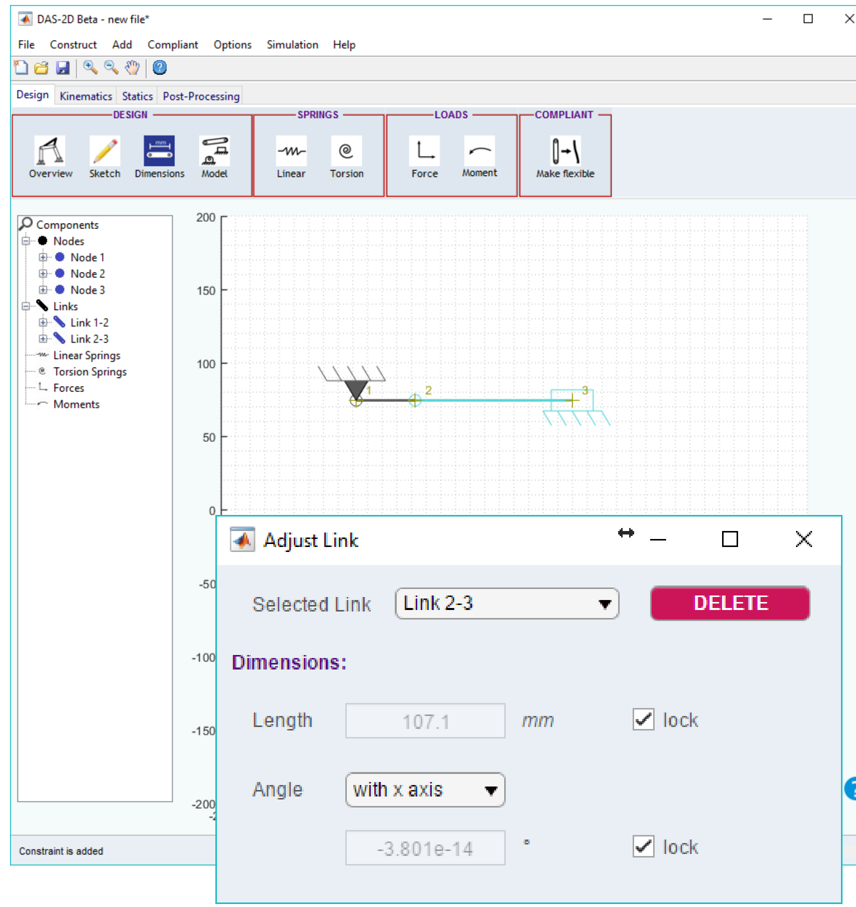




Ex.5 Constant Force Mechanism

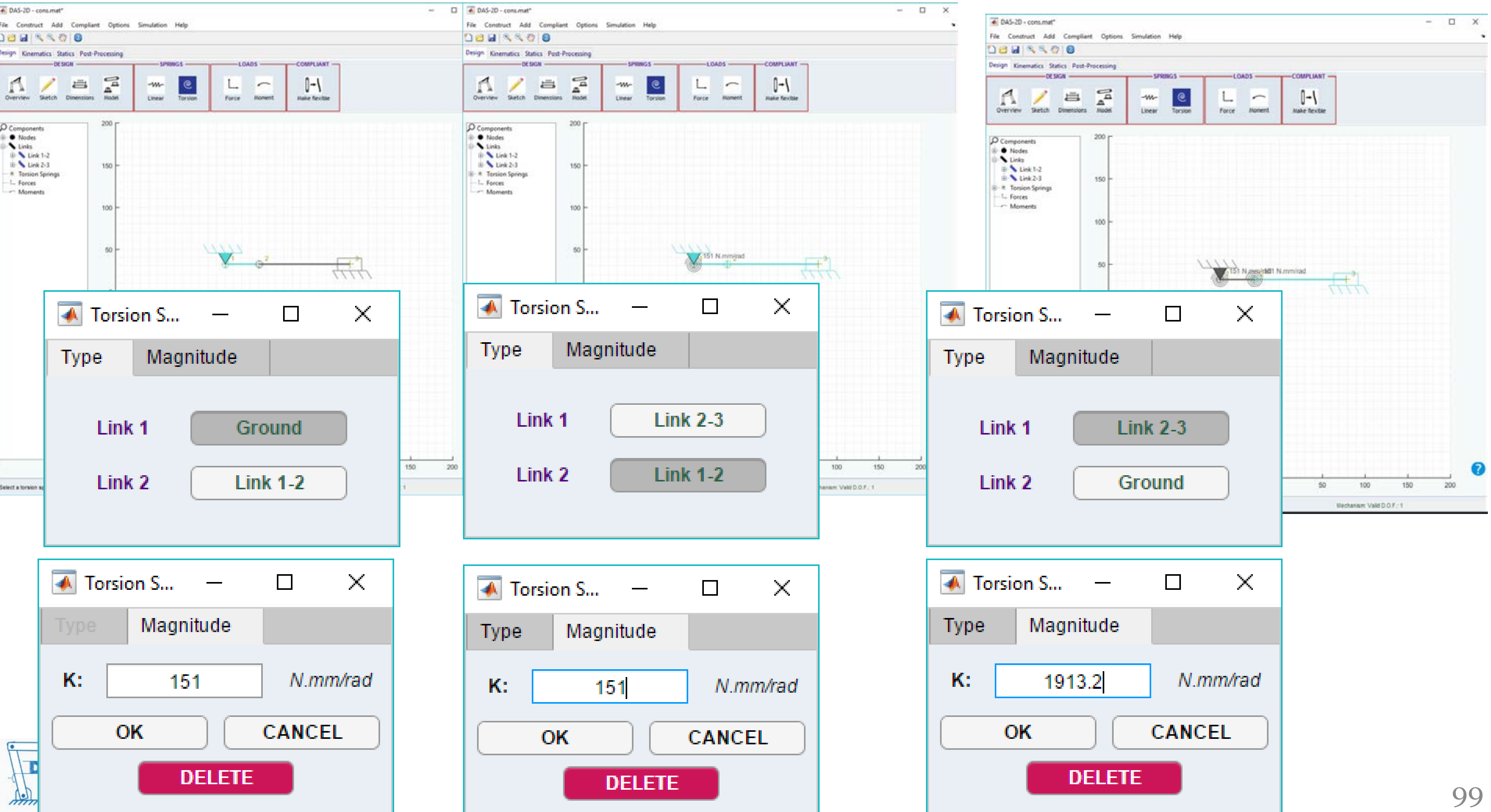
Step 3. Go to Dimension Module and enter following dimensions.





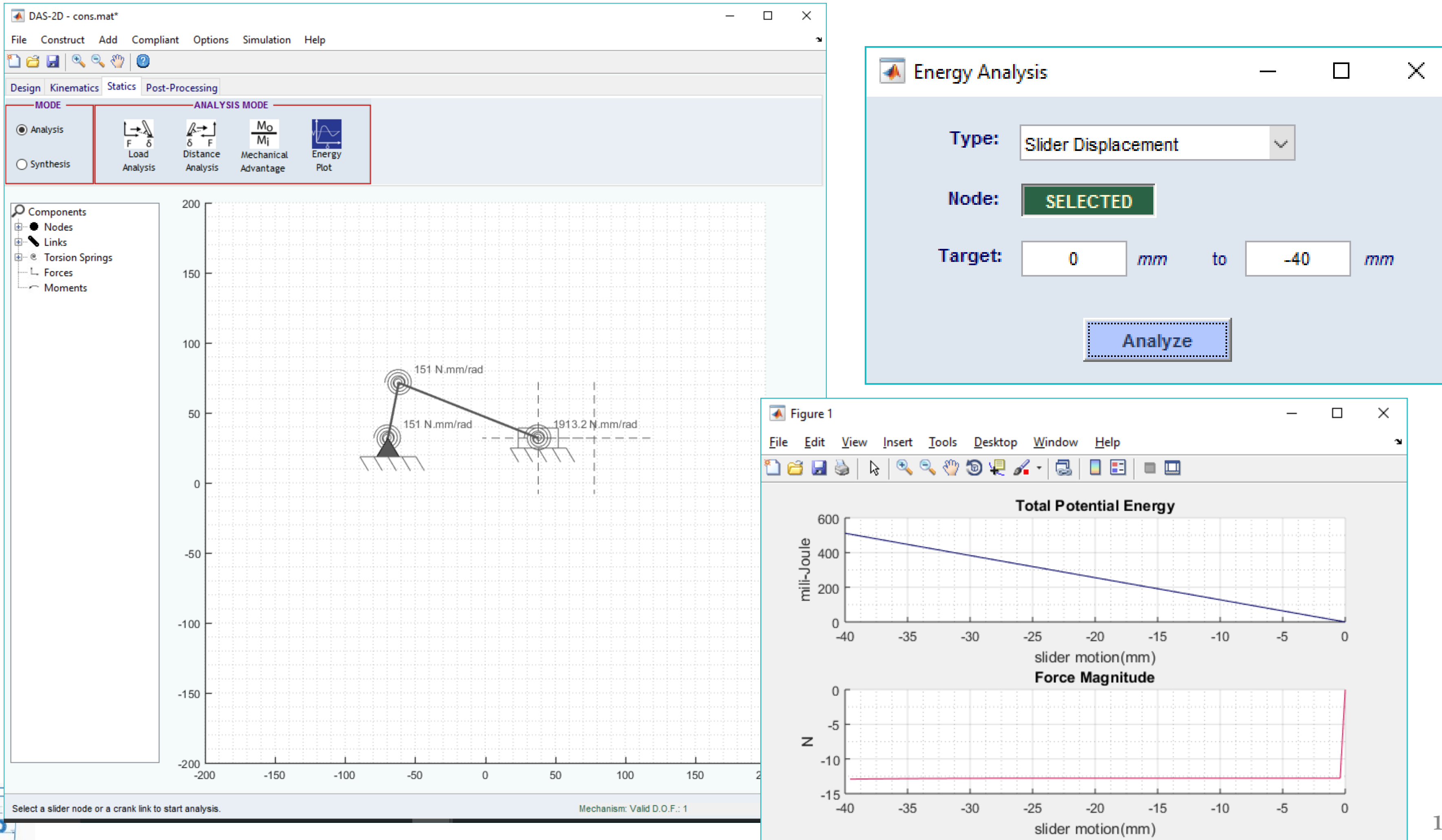
Ex.5 Constant Force Mechanism

Step 4. Go to Torsion Spring Module. Select Link 1-2 and Ground (Click empty space) Magnitude = 151
Go to Torsion Spring Module. Select Link 1-2 and Link 2-3. Magnitude = 151.
Go to Torsion Spring Module. Select Link 1-2 and Link 2-3. Magnitude = 1913.2.



Ex.5 Constant Force Mechanism

Step 6. Go to Statics-> Energy Plot. Select type as Slider Displacement. Click to Node 3. Target is 0 to -40 mm. Hit Analyze.



Ex.5 Constant Force Mechanism

Check if your force is around 13N

Example: Mechanism with Three Flexural Pivots. For a mechanism with three flexural pivots (Figure 10.2n) and a deflection of $\Delta x/(r_2 + r_3) = 0.16$, Table 10.1 suggests that $R = r_3/r_2 = 2.6633$, $K_1 = 1.0$, and $K_2 = 12.67$. If $r_2 = 40.2$ mm (1.583 in.), then $r_3 = Rr_2 = 107.1$ mm (4.216 in.). Recall that $k_i = EI_i/l_i$. If each flexural pivot has the same width (w), length (l), and modulus of elasticity (E), then $K_1 = k_2/k_1 = h_2^3/h_1^3$ and $K_2 = k_3/k_1 = h_3^3/h_1^3$. If $h_1 = 0.76$ mm (0.030 in.), then $h_2 = h_1$, and $h_3 = h_1 K^{1/3}{}_2 = 1.78$ mm (0.070 in.).

The nondimensionalized force term is $\Phi = 3.4016$ (Table 10.1). If the mechanism has a width $w = 12.7$ mm (0.50 in.), modulus of elasticity $E = 1655$ MPa (240,000 lb/in.^{2}), and the flexural pivots each have a length of $l = 5.08$ mm (0.20 in.), then

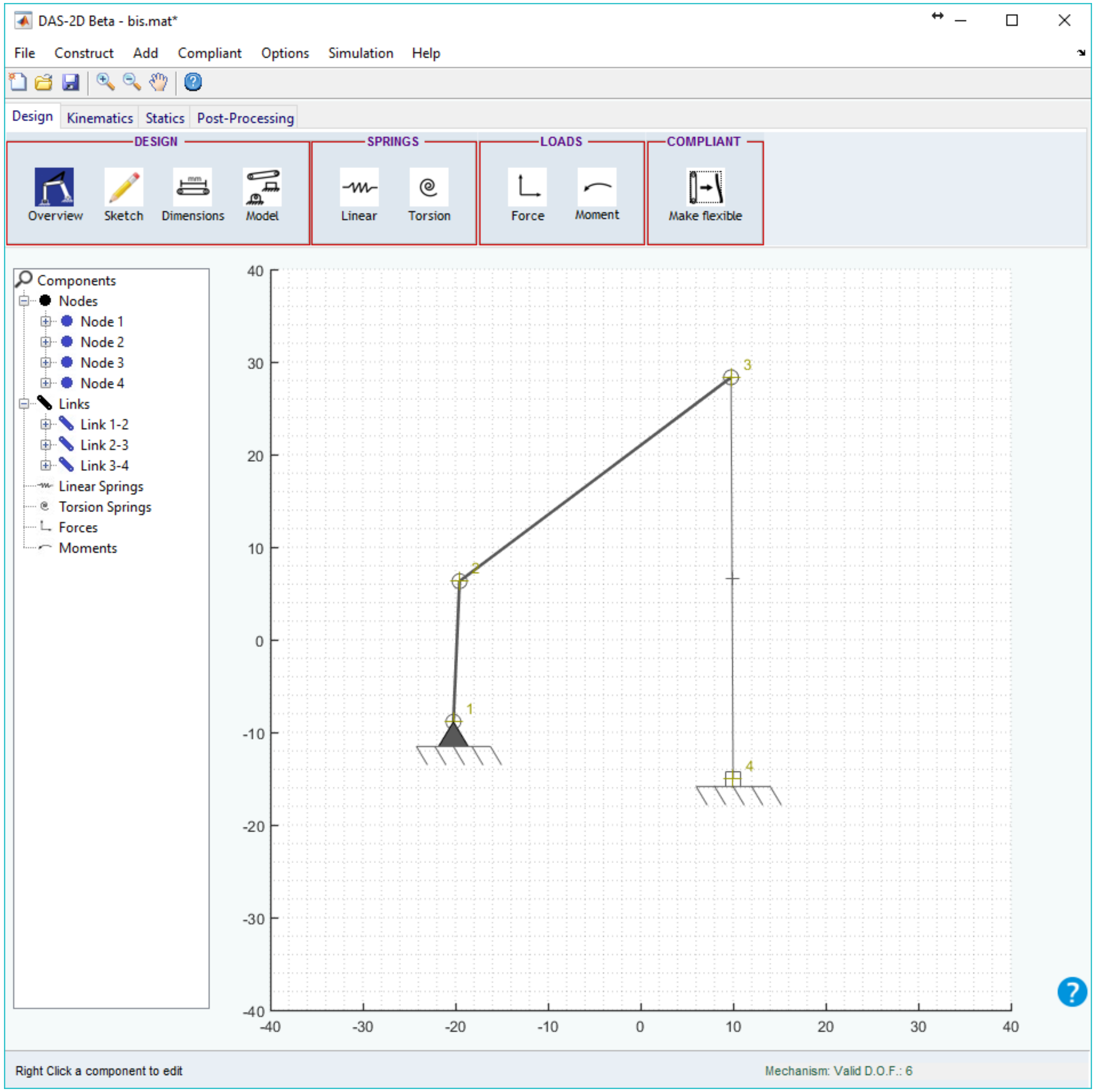
$$k_1 = \frac{EI}{l} = 151\ \frac{\text{N-mm}}{\text{rad}} \qquad \left(1.35\ \frac{\text{in.-lb}}{\text{rad}}\right) \tag{10.19}$$

The resulting force is

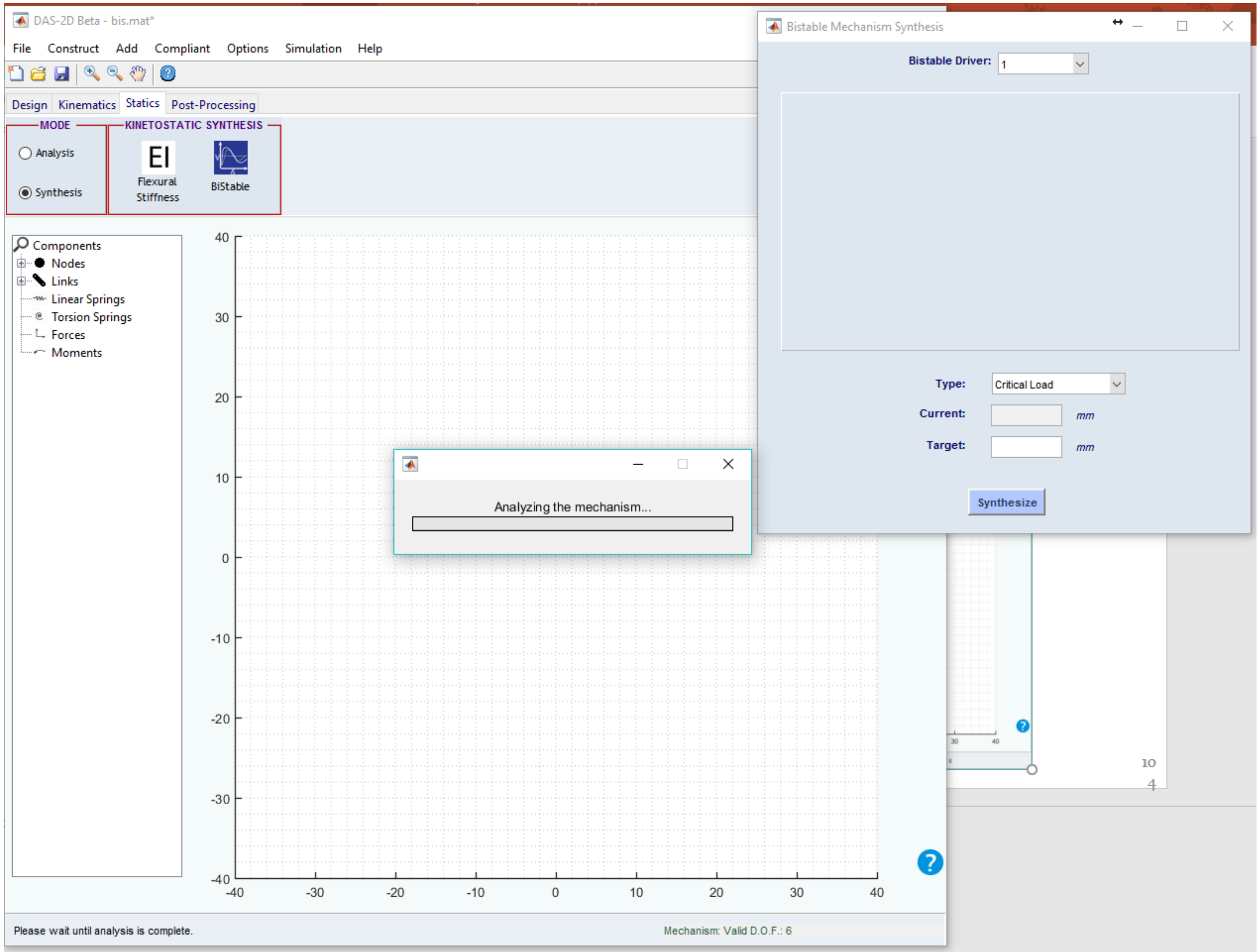
$$F = \frac{k_1}{r_2}\Phi = 13\ \text{N} \qquad (2.9\ \text{lb}) \tag{10.20}$$

Compliant Mechanisms – Larry L. Howell
Example: Mechanisms with Three Flexural Pivots – Page 344

Step 1. Create or load Ex.4

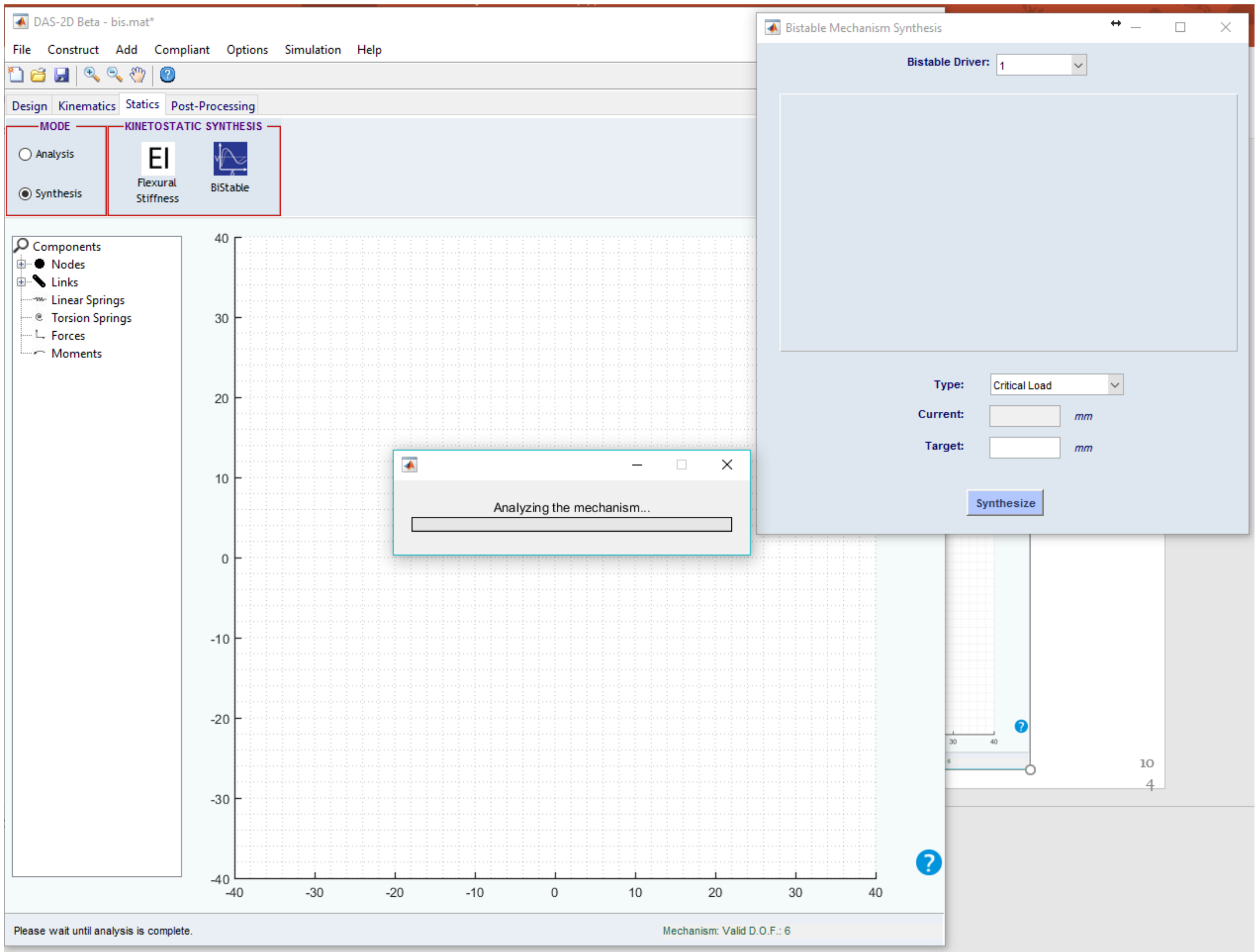


Step 2. Go to Statics-> Synthesis -> Bistable. The mechanism will be analyzed and two bistable drivers will be found (same driver CW and CCW)

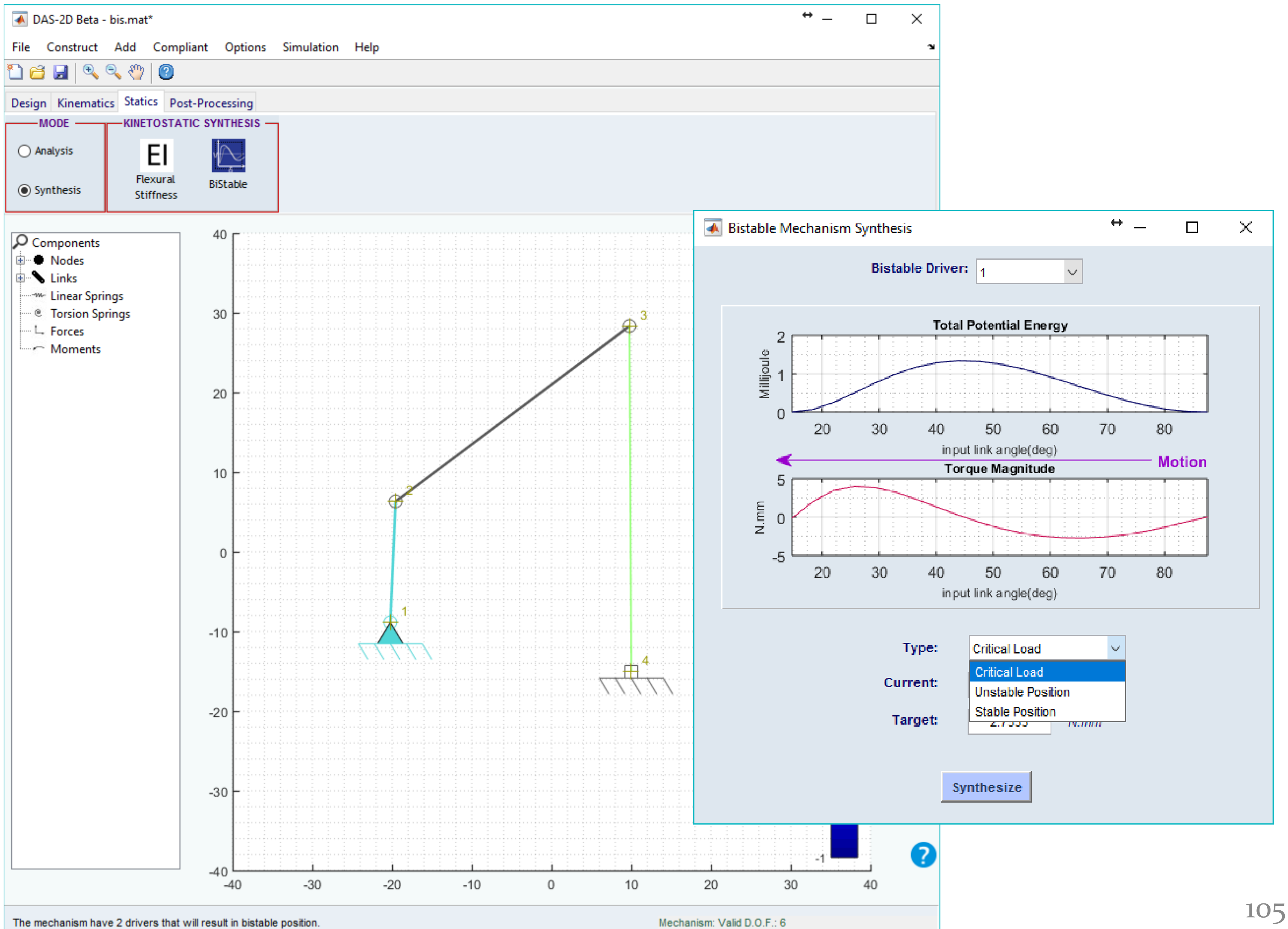


Ex.6 Bistable Four Bar Mechanism Synthesis

Step 2. Go to Statics-> Synthesis -> Bistable. The mechanism will be analyzed and two bistable drivers will be found (same driver CW and CCW)



Ex.6 Bistable Four Bar Mechanism Synthesis



Ex.6 Bistable Four Bar Mechanism Synthesis

Step 3. Go to Statics-> Synthesis -> Bistable. The mechanism will be analyzed and two bistable drivers will be found (crank CW and CCW). Enter 5 as the critical load and flexural stiffness of the compliant member will be optimized.