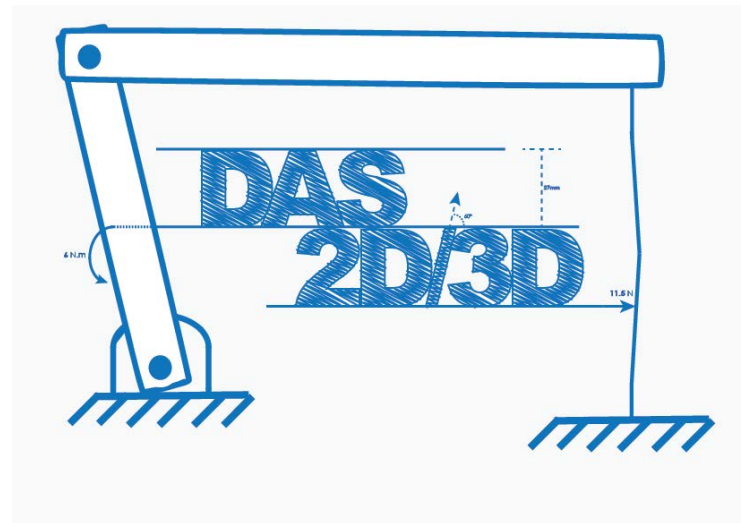


Design, Analysis and Synthesis (DAS) - 2D: A Design Tool for Compliant Mechanisms



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



- 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.



- Design, Analysis and Synthesis (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.



- 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



- 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




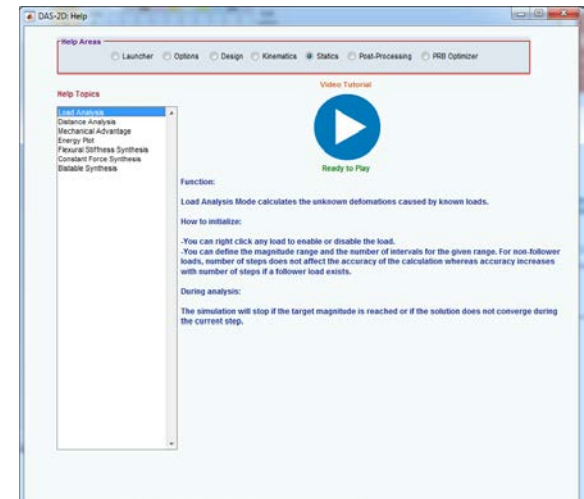
Through the launcher you can:

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



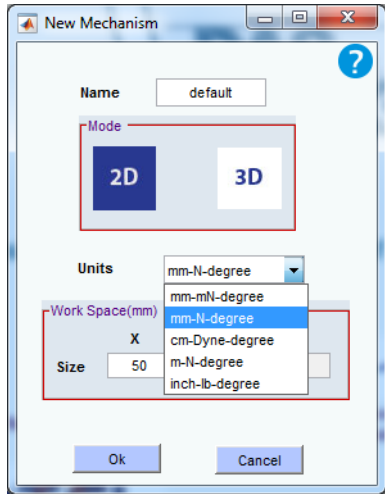
DAS 2D Launcher

- 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

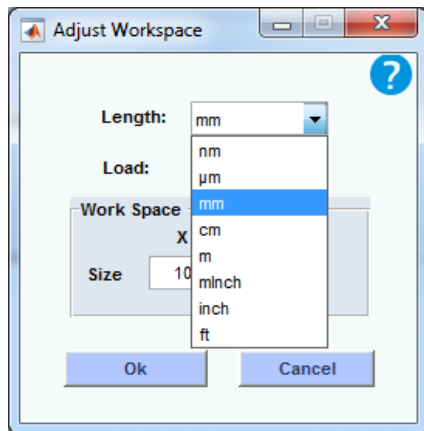




Before you create a new mechanism, you need to select:

- One of four unit systems
- Workspace size

New Mechanism Screen



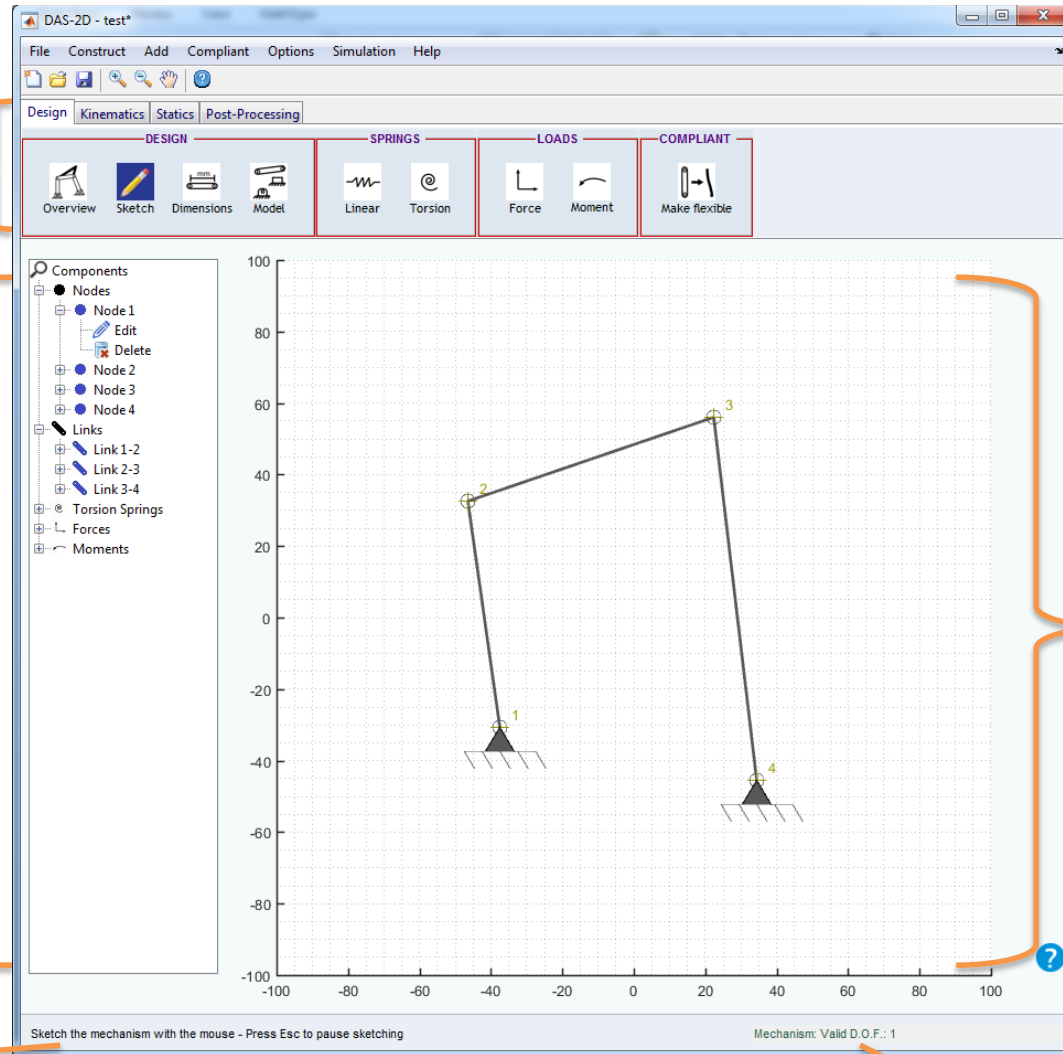
More unit types are available at:
Menu->Options->Workspace

Options



Different Modules:

- Design
- Kinematics
- Statics
- Post-Processing



Component Tree

Workspace

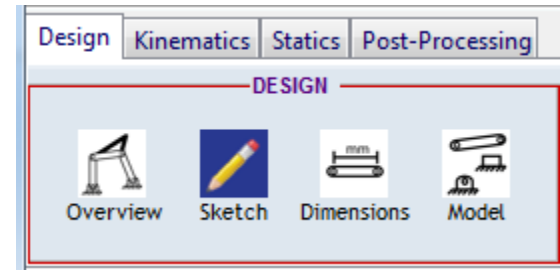
Help System

Current Module Tips

- Mechanism Status
- Total Degrees of Freedom

The Design Module has four tools

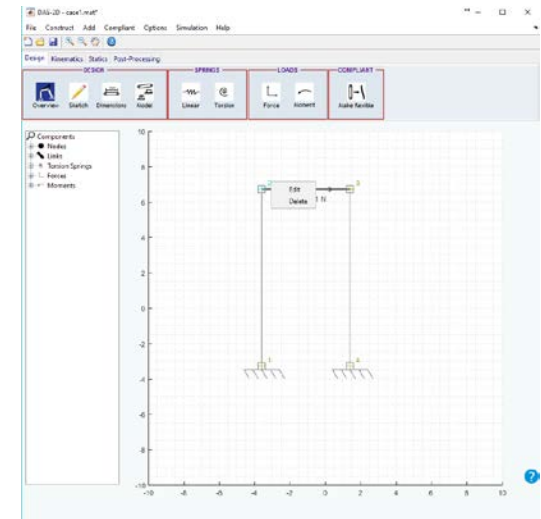
- **Overview**: add, edit and delete a node or link or joint
- **Sketch**: sketching schematic view of a mechanism
- **Dimension**: sizing link length, angle, coordinate etc.
- **Model**: change joint type etc.



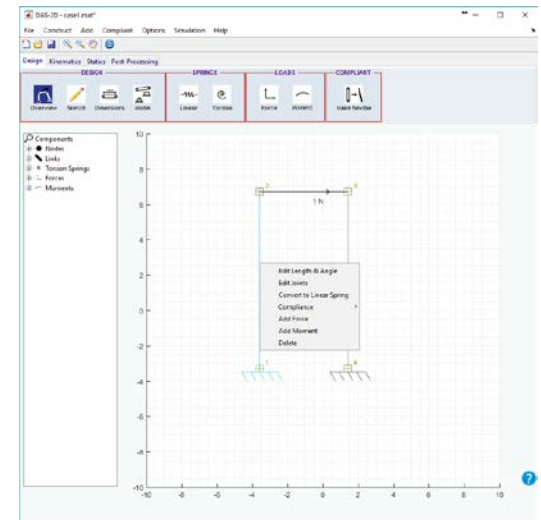
Design Module

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



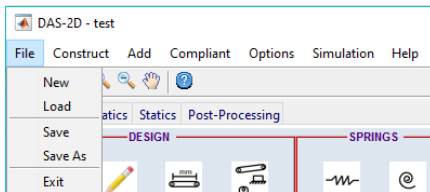
Edit a Link

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

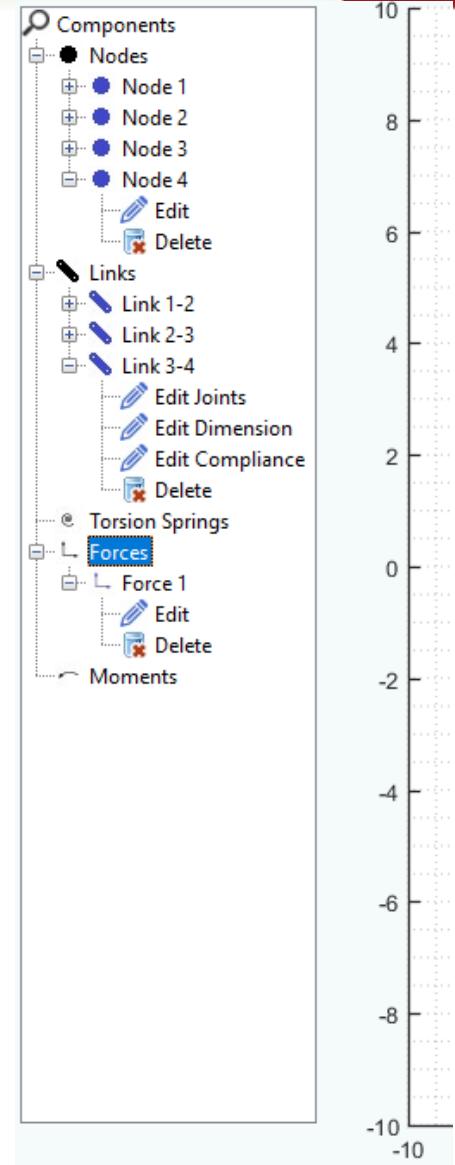
- The toolbar and the menu can be used to open, load or save the mechanism.
- Any module can be accessed via the menu.
- The workspace can be adjusted (zoom in-out or pan) using the toolbar.
- The exact workspace size can be set under Options->Workspace



Menu



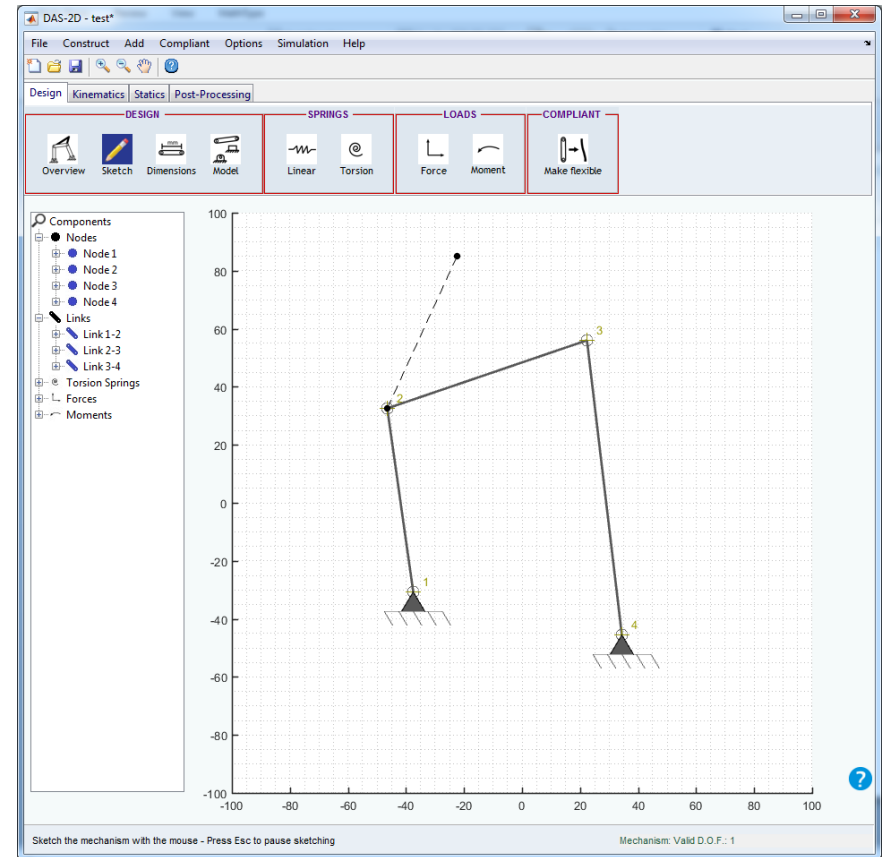
Toolbar



Component Tree

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

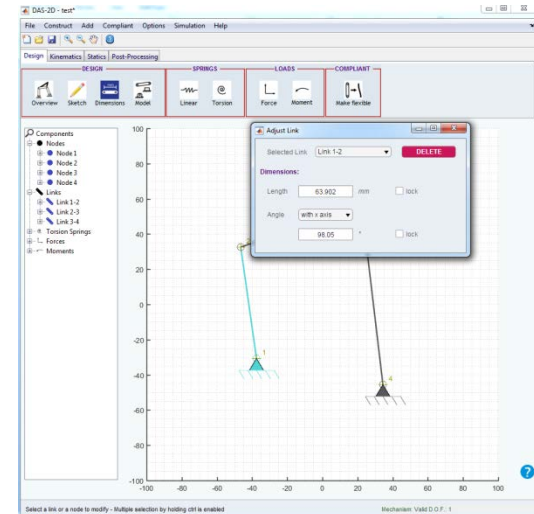


Sketching Tool



Edit a Link

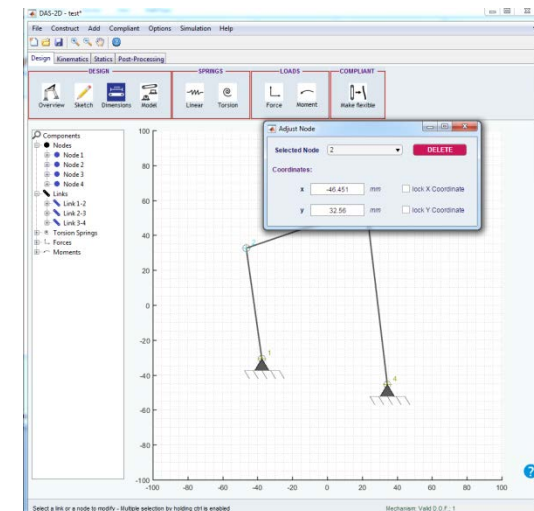
- Click a link to edit length or angle
- Enter a valid length and click lock or hit enter key
- You can specify the angle with the x or y axis
- Current or desired angle can be negative. Make sure to check the link direction



Edit a Link

Edit a Node

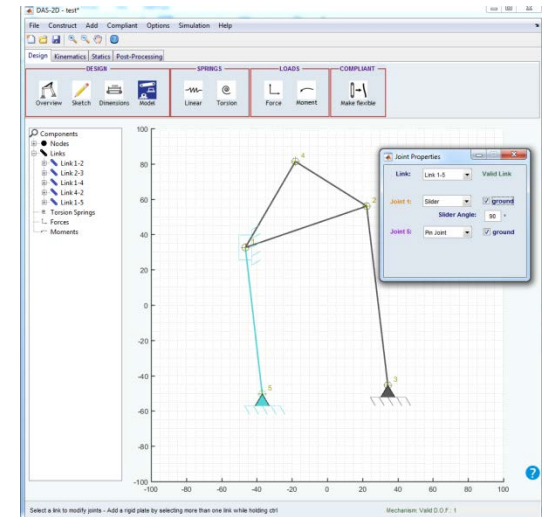
- Click a node to edit x or y coordinates
- Enter a valid coordinate and click lock or hit enter key
- Locked node coordinates will not change if link lengths or angles are altered



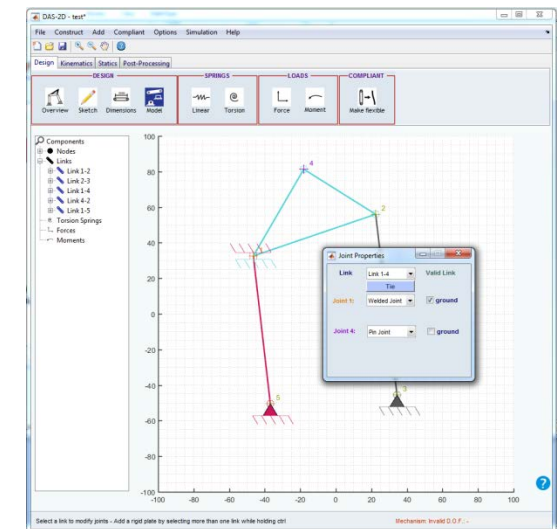
Edit a Node

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 draw in red color.



Edit Joints



Add a Rigid Plate

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



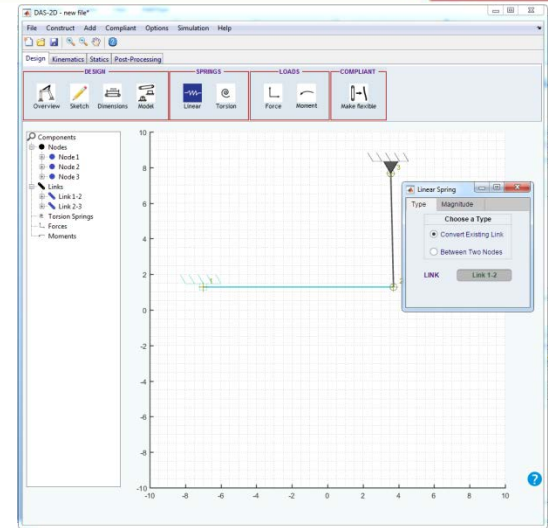


Design Module Demonstration



Linear Spring - Convert a Link

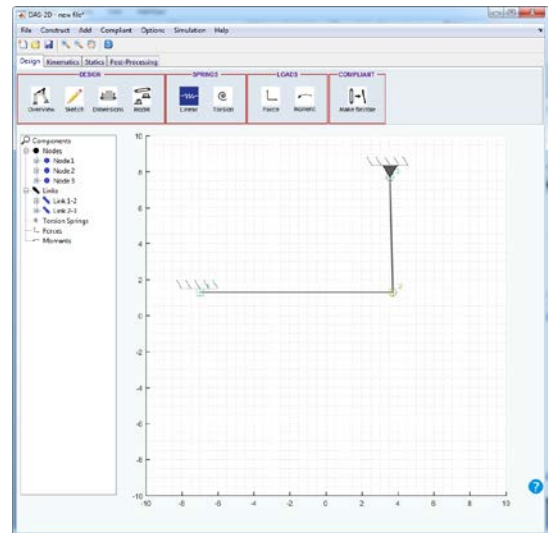
- A link can be converted to a linear spring.
- Click a link that you want to convert
- Enter the magnitude



Convert a link

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.



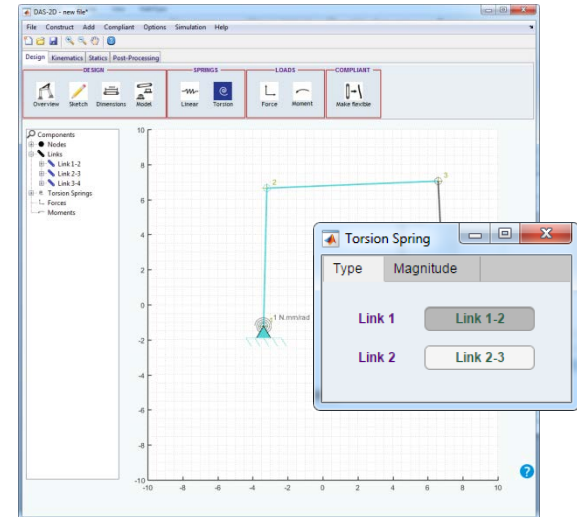
Add a new Link

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



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.

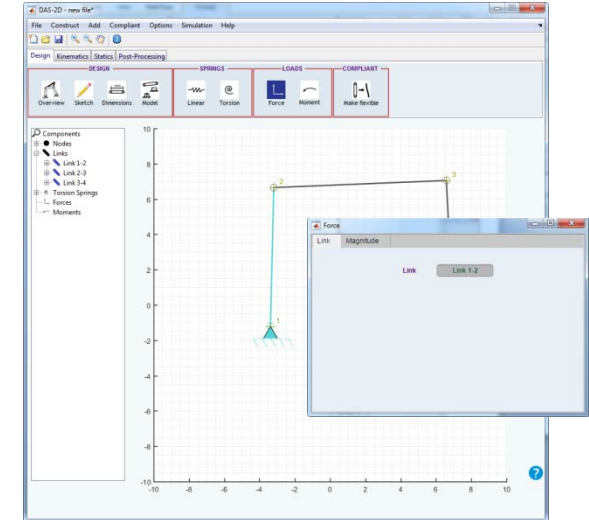


Add a Torsion Spring

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

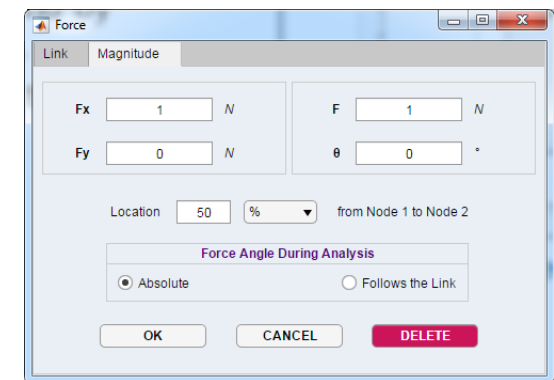
Add a Force

- 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.

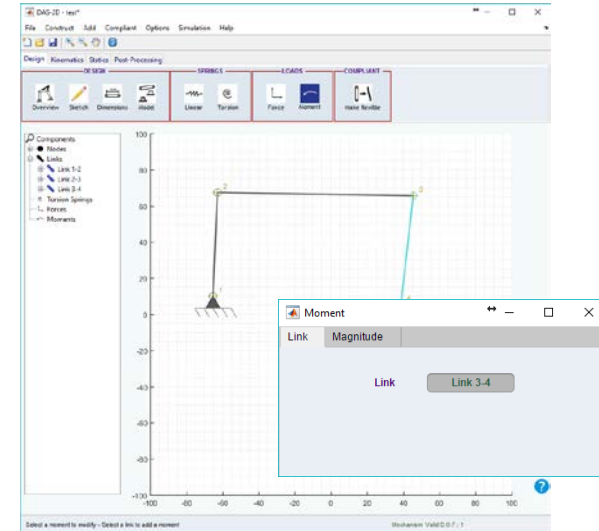


Add a Force - Properties



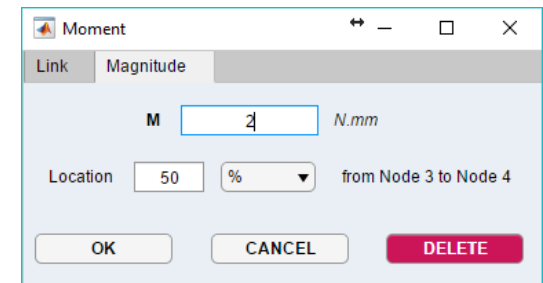
Add a Moment

- 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

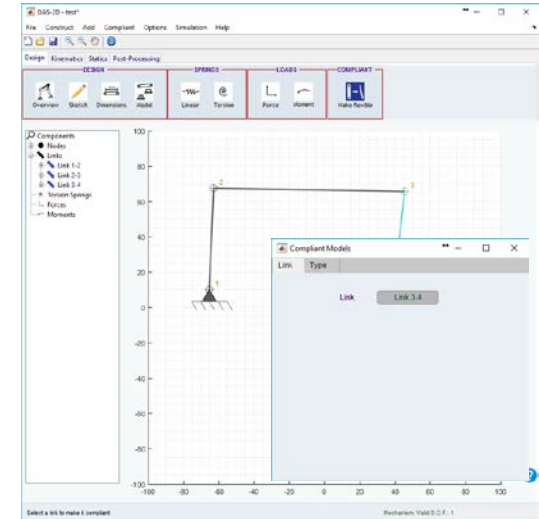


Spring and Load Module Demonstration

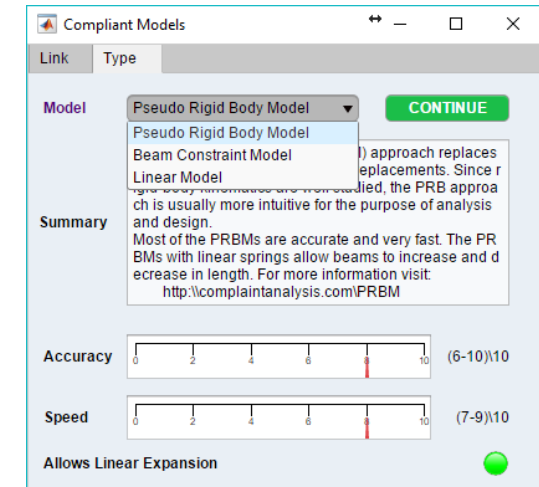


Make a Link Compliant

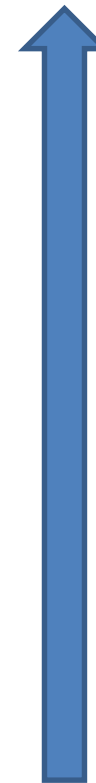
- Select a link that will be compliant
- There are three different models you can select
 - Pseudo-Rigid-Body Models
 - Beam Constraint Model
 - Linear Model
- Beam Constraint Model and Linear Model can consist of multiple segments
- To switch between PRB and other models, the link must be converted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



Make Compliant - Select a Link



- 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

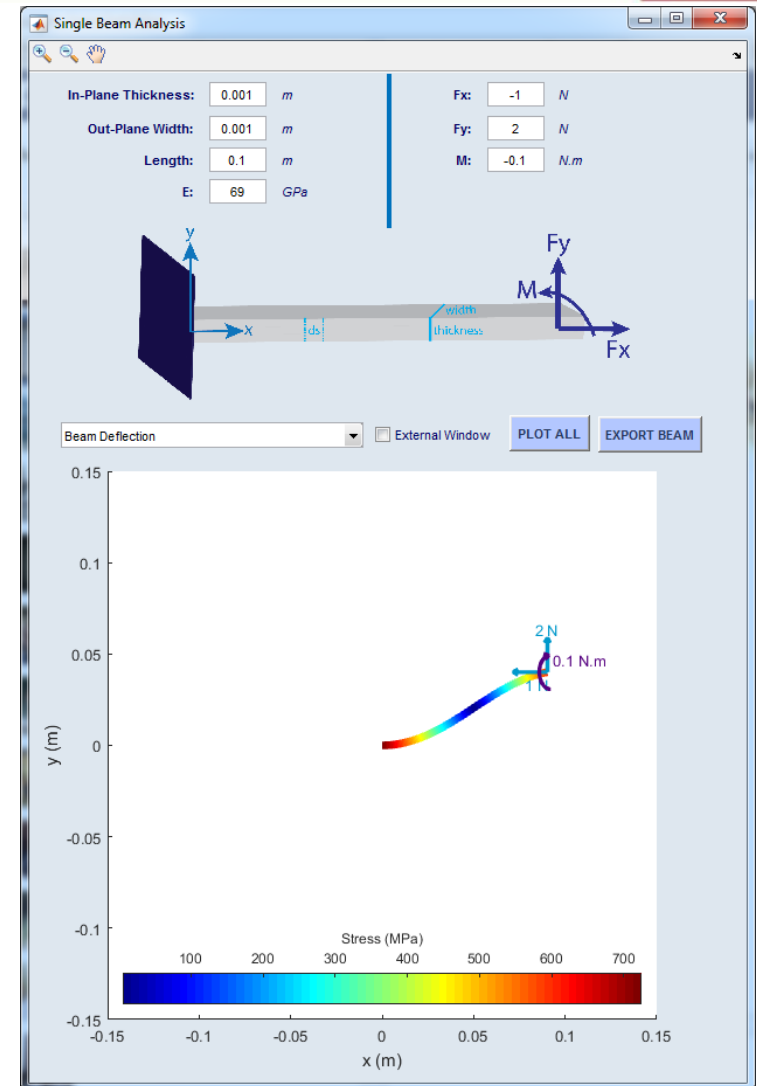
- Single beam subject to arbitrary end load
- Large deflection Euler-Bernoulli beam equation

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

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

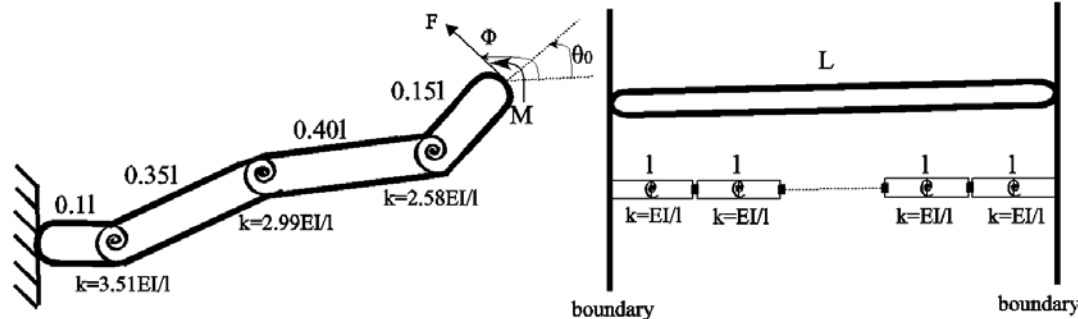
$$\begin{aligned} \theta'' &= F_x \sin\theta - F_y \cos\theta, \\ \theta(0) &= 0, \theta'(L) = M_0, \end{aligned}$$

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



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



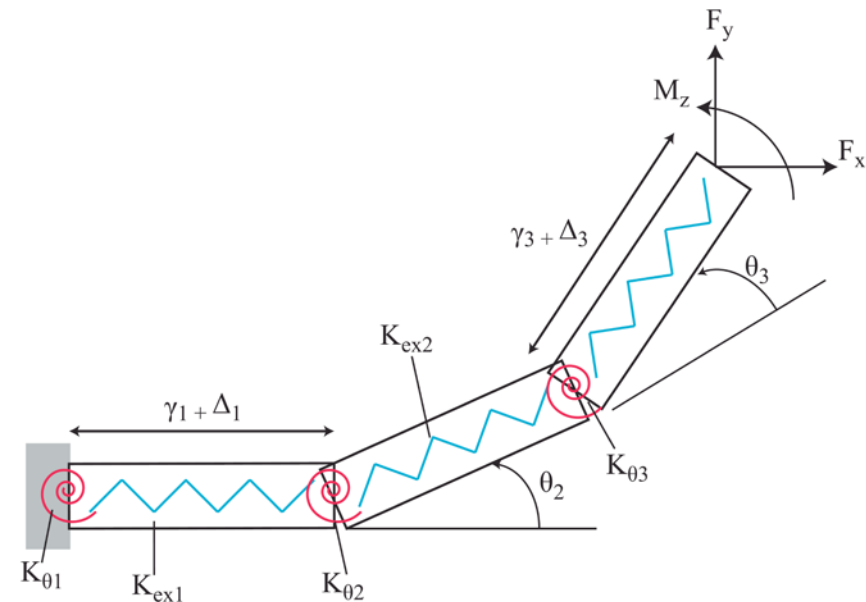
PRB-3R

Finite Segment Model (FSM)

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

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

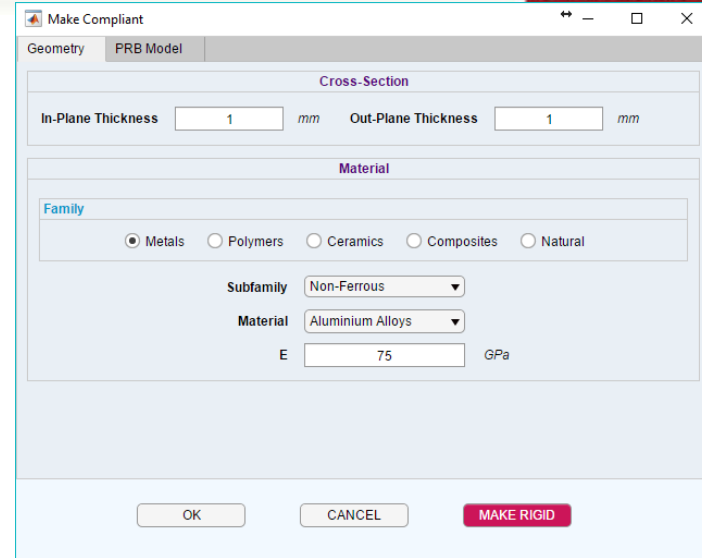
- Each segment has 3 parameters : $\gamma, k_{\theta}, k_{ex}$
- Segment length: $l_i = \gamma_i L_{beam}$
- i^{th} Torsion Spring Magnitude: $K_{\theta i} = k_{\theta i} \frac{EI}{L}$
- Segment Axial Stiffness: $K_{exi} = k_{exi} \frac{EA}{L}$
- If i^{th} torsion spring is not present, $k_{\theta i}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



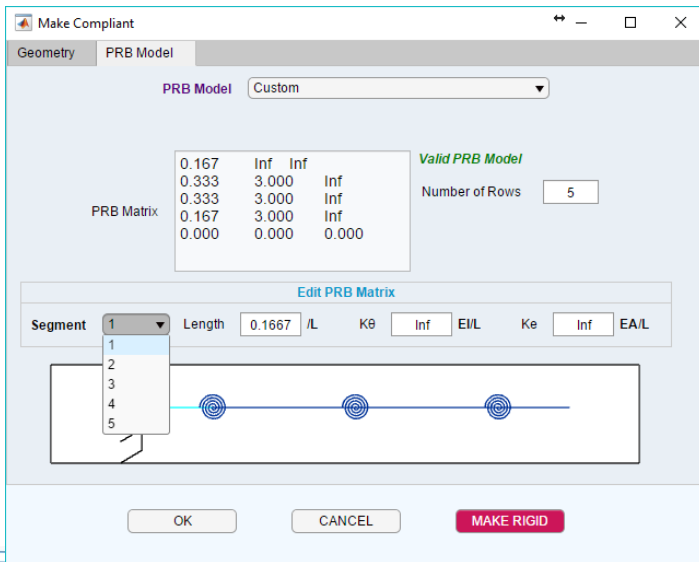
General PRB model with 3 elements

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

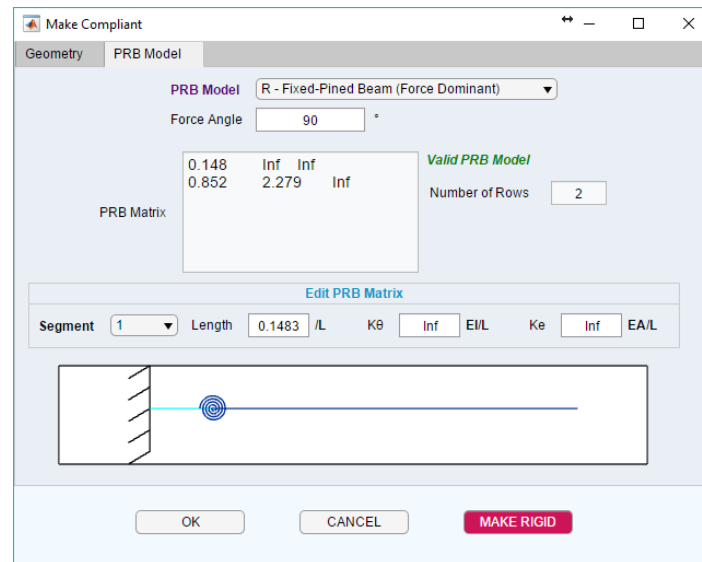
- You need to enter cross-section properties for a PRB beam.
- Since PRB-R is load dependent, an approximate end force angle can be specified for more accurate analysis.
- All PRB Models are customizable. Select a segment and update the three parameters.
- Number of segments can be specified for the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



Cross-Section Properties



PRB - Custom

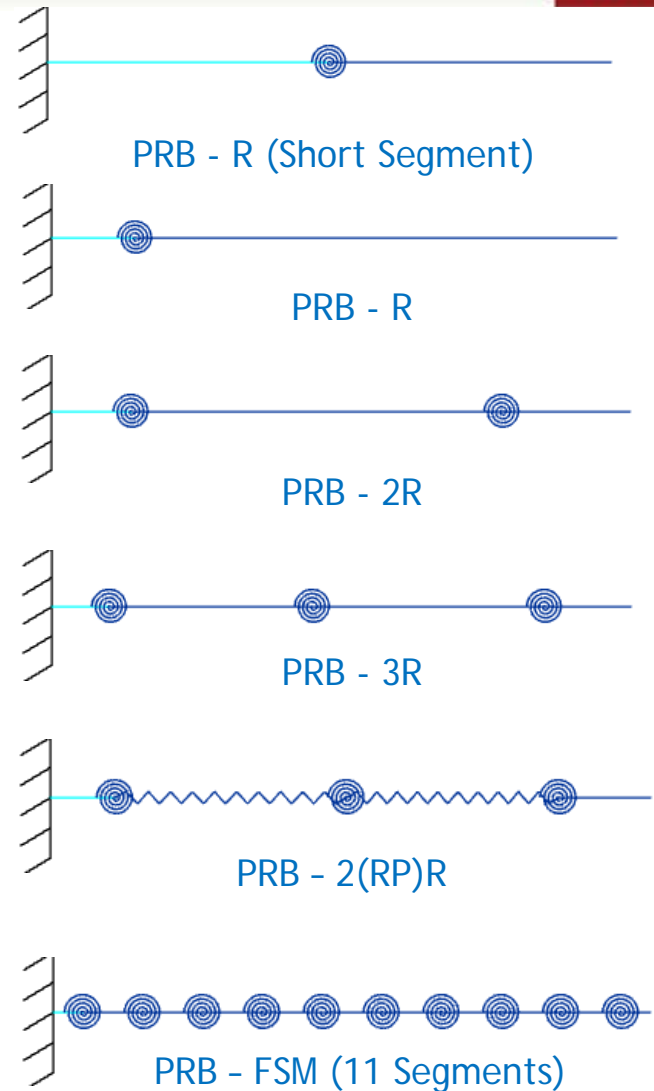


PRB - R



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





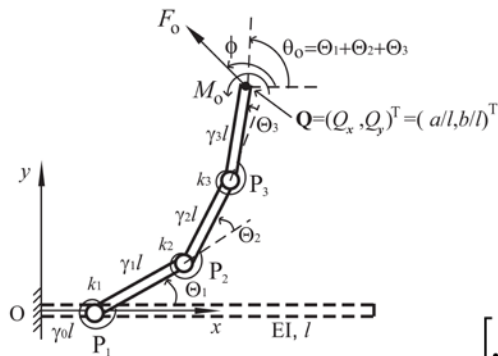
Computing Beam Shape and Stress



- 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:



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

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

Where τ_1, τ_2 and τ_3 are torsion spring torques

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

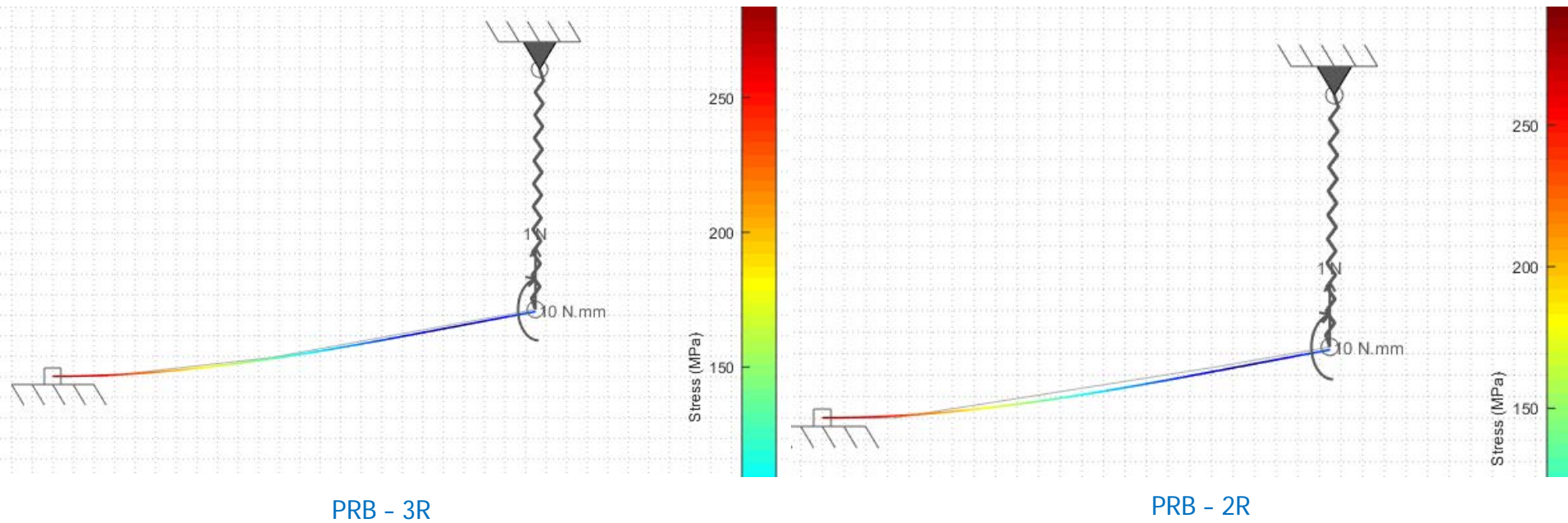
J^T has three columns and n (number of torsion springs + linear springs) rows.

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

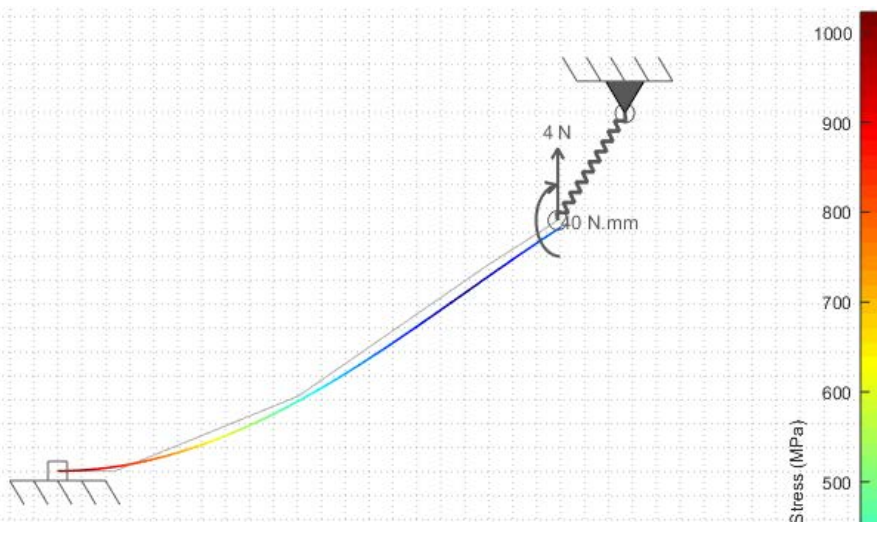
- The J^T can be inverted very easily for PRB models with n (number of torsion springs + number of linear springs) = 3.
- It is not possible to invert J^T for models that $n < 3$ (PRB-R and PRB-2R) or $n > 3$ (PRB-FSM and PRB-2(RP)R).
- Moore-Penrose pseudoinverse can be used to calculate J^T inverse.
- 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 $\left(\sim 10^5 \frac{EA}{l}\right)$ until $n = 3$.

- 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.

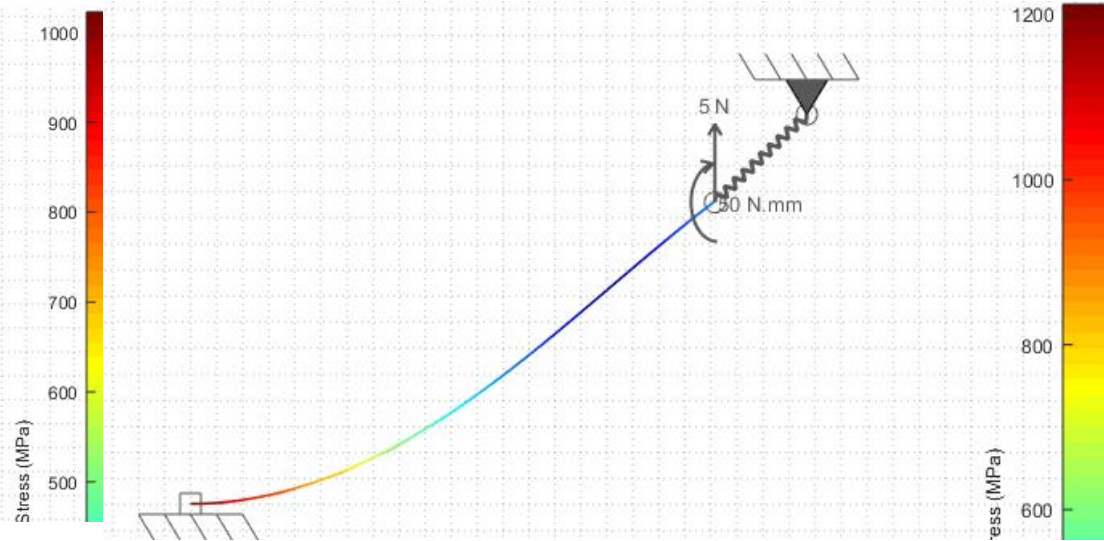
Single Beam Analysis (Linear Spring has no stiffness)



- 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.



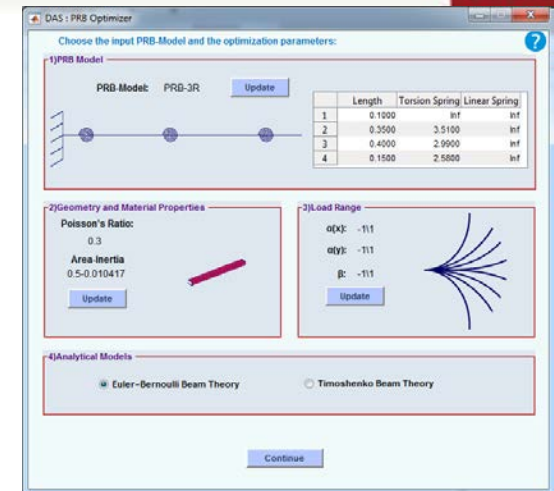
PRB - 3R



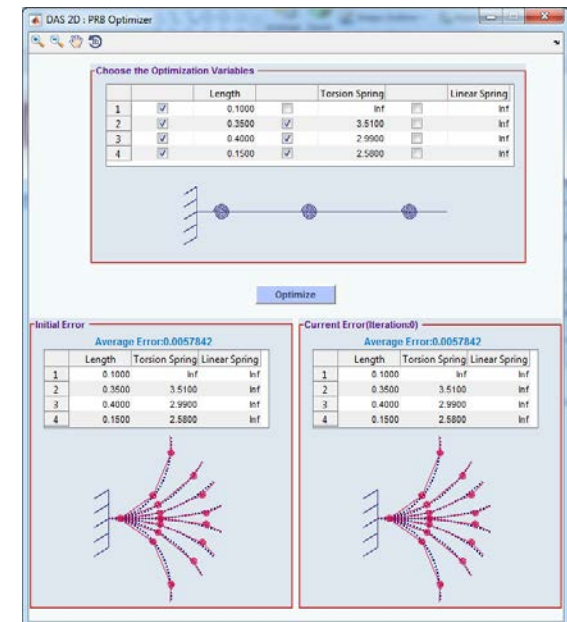
PRB - FSM (11 Segments)

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

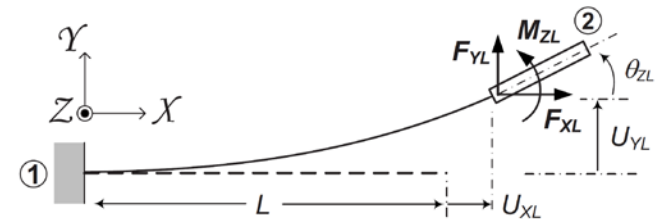


Optimization Screen

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

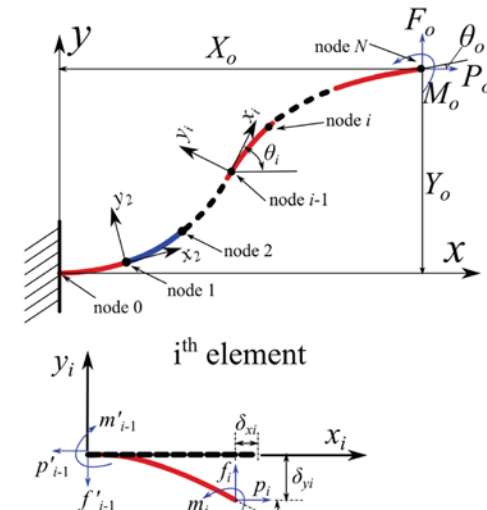


- BCM was developed for accurately analyzing beams in the intermediate deflection range.
- Deflections within 10% slope change of the beam length can be accurately captured by a single BCM beam.
- BCM is parametric and have closed form equations for the beam deformation and strain energy.
- Axial strain 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

Awatar, 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).

- Force - deformation equations for BCM Beams are:

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

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

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.

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

- 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.

Strain Energy stored in a BCM Beam



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

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

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

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

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

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

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

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

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

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

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

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

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

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

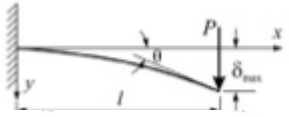
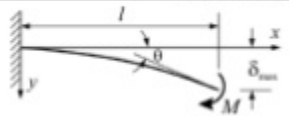
$$k_{22} = \frac{\sin(rx)}{r \cos r}$$

$$c_{11} = \frac{[4rx + 2rx \cos(2r) - 4 \sin(rx - 2r) - 4 \sin(rx)] - \sin(2r - 2rx) - 3 \sinh(2r)}{8r^5 \cos^2 r}$$

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

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

- 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.

| 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  | $\theta = \frac{MI}{EI}$ | $y = \frac{Mx^2}{2EI}$ | $\delta_{max} = \frac{MI^2}{2EI}$ |

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.

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

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

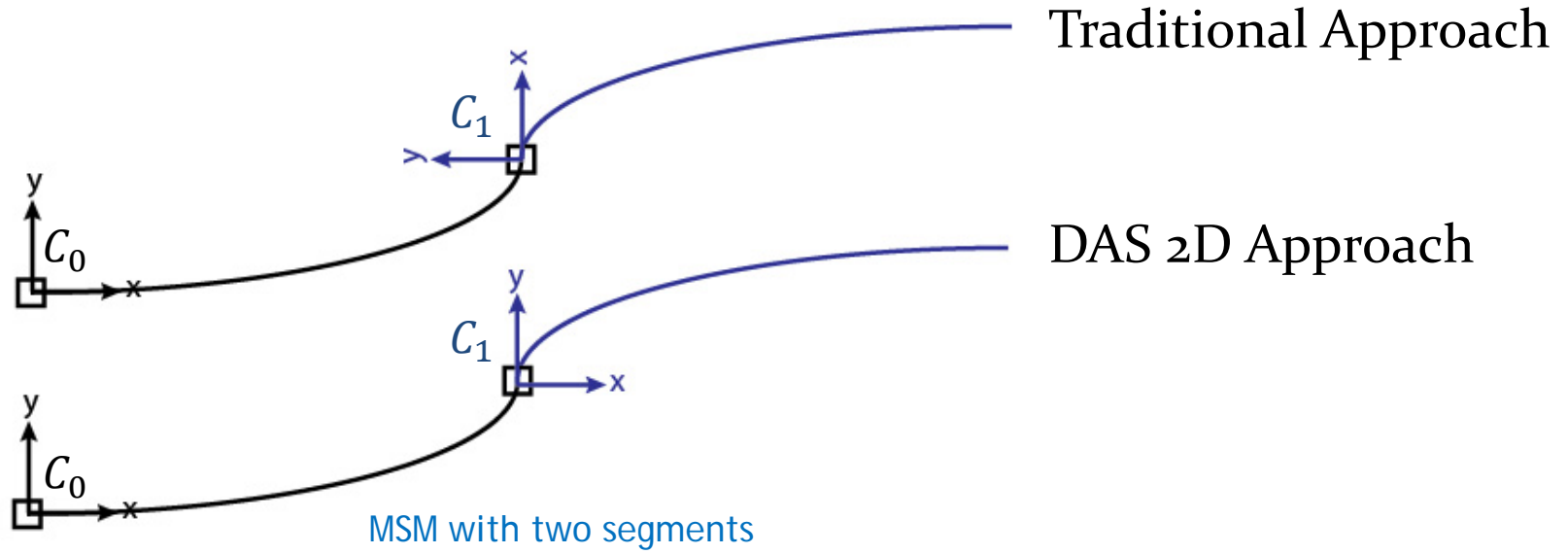
- Bending stress is:

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

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.

- 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.



- 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:

$$\theta_{C_i} = \theta_{C_0} + \sum_0^{i-1} \alpha_i \text{ (} i^{th} \text{ tip angle)}$$

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



- 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).

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

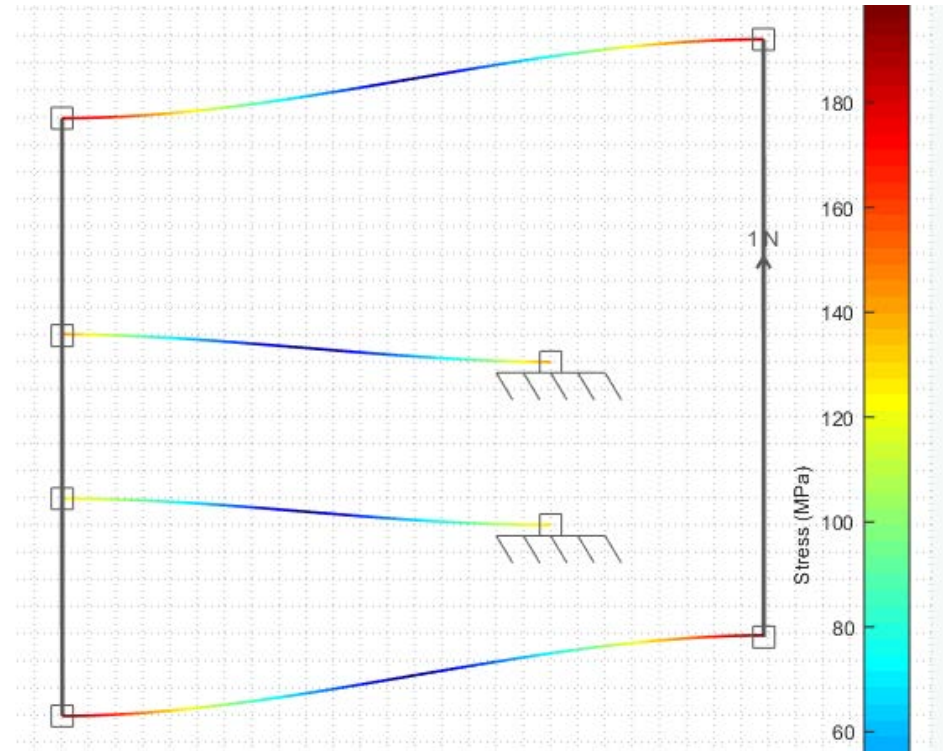
~81 seconds

Energy and Constraint Gradient

~7 seconds

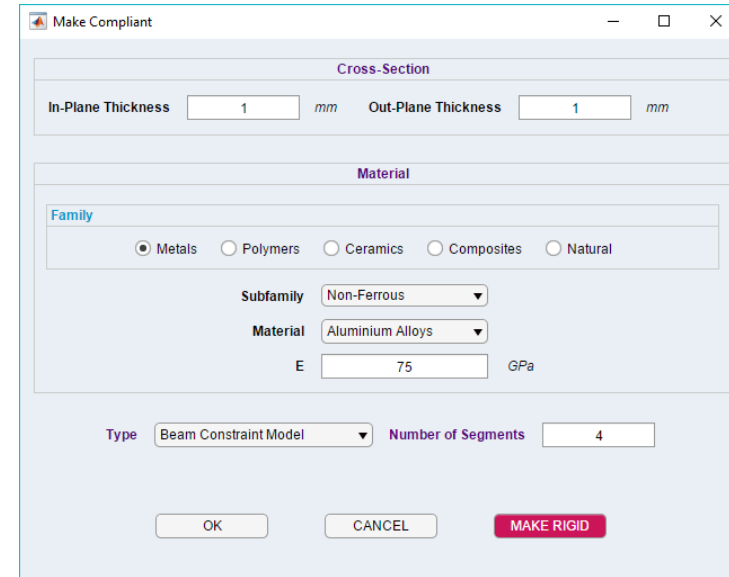
Gradients and Hessian

~1 second



Parallel Guiding Mechanism

- 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



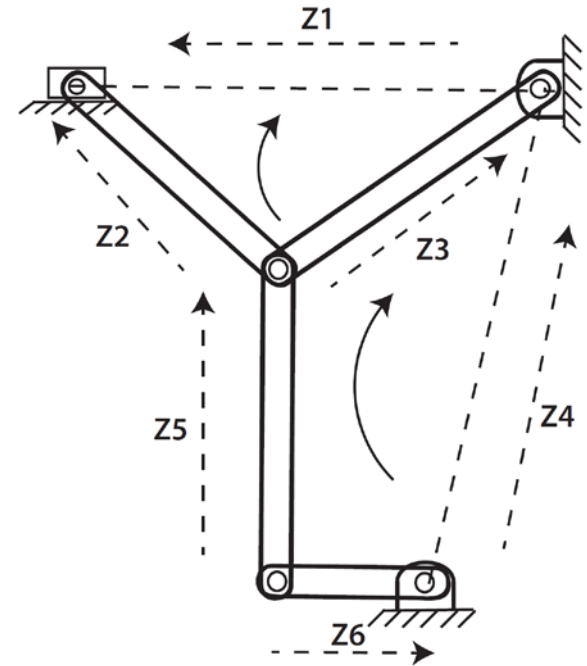
- Kinematic analysis of rigid-body linkages
 - Interactive kinematic analysis: free dragging
 - Range Kinematics: a range of kinematic motion
- Kinetostatic analysis of compliant mechanisms
 - Load analysis
 - Distance analysis
 - Mechanical advantage
- Synthesis routines
 - Flexural stiffness synthesis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

- 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:

$$\vec{Z}_i = \vec{Z}_{i0} + \lambda_i \times \vec{f}(x_i)$$

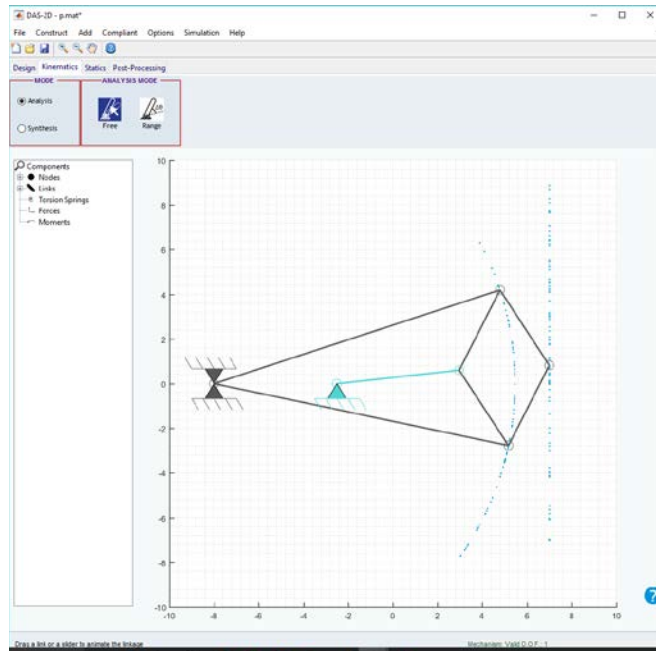
- For n independent loops, the kinematic equations for a mechanism with l links can be formed as:

$$\sum_{i=1}^n C_{ki} \vec{Z}_i = \begin{bmatrix} 0 \\ 0 \end{bmatrix}, \quad 1 \leq k \leq l$$

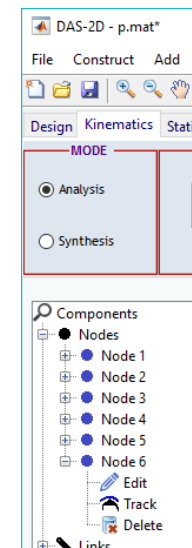


$$\begin{aligned} \vec{Z}_2 - \vec{Z}_1 - \vec{Z}_3 &= 0 \\ -\vec{Z}_6 + \vec{Z}_5 + \vec{Z}_3 - \vec{Z}_4 &= 0 \end{aligned}$$

- 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.

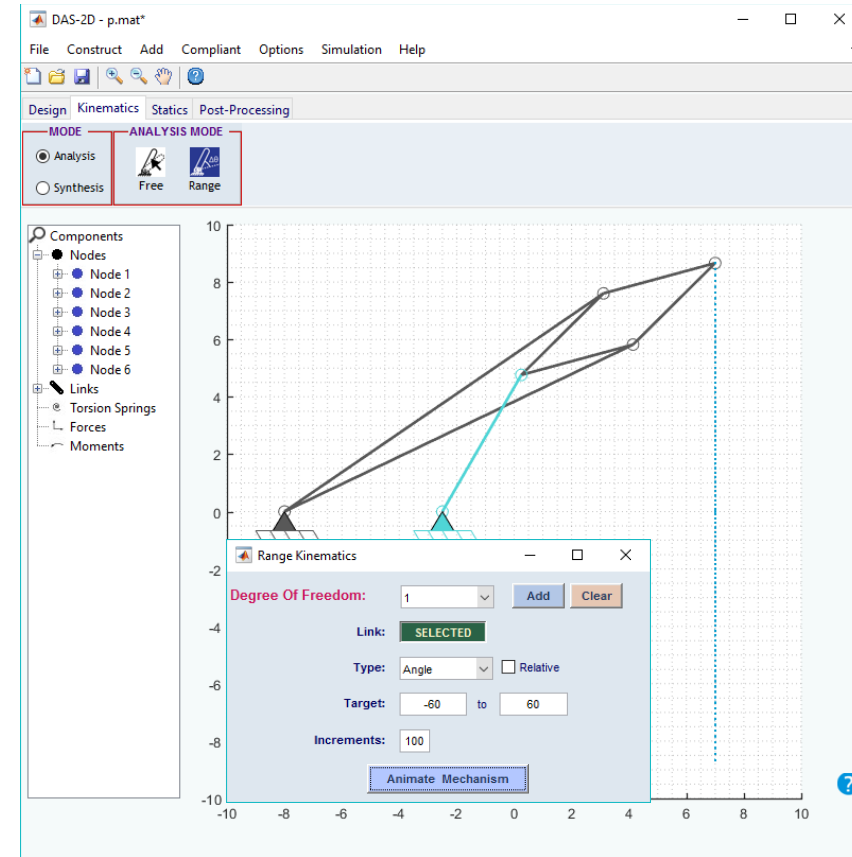


Free Mode



Track a Node

- 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



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:

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

V is the total work done by loads:

$$V = \sum_{i=0}^n \int_{r_{i0}}^{r_i} \vec{F}_i \cdot d\vec{r} + \sum_{j=0}^m \int_{\theta_{j0}}^{\theta_j} M_j d\theta$$

Optimization of the total energy:

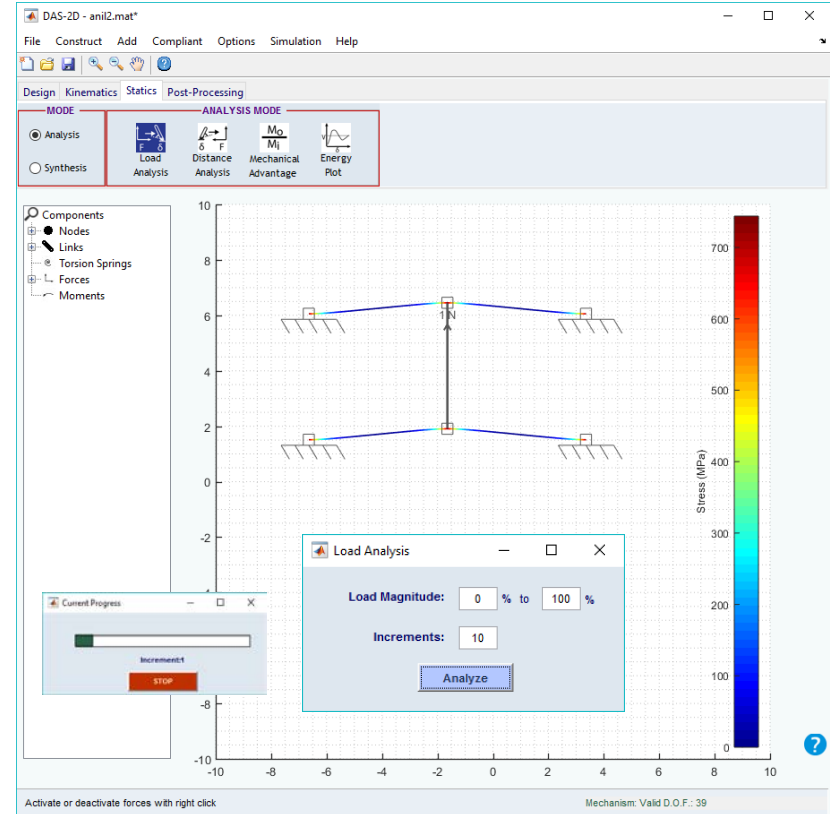
$$\min_{\Psi} f = \left(\sum_{k=1}^n E_k(\Psi) - \sum_{i=0}^n \int_{r_{i0}}^{r_i} \vec{F}_i \cdot d\vec{r} - \sum_{j=0}^m \int_{\theta_{j0}}^{\theta_j} M_j d\theta \right)$$

$$\text{Subject to: } g = \sum_{i=1}^n C_{ki} \vec{Z}_i = 0$$

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

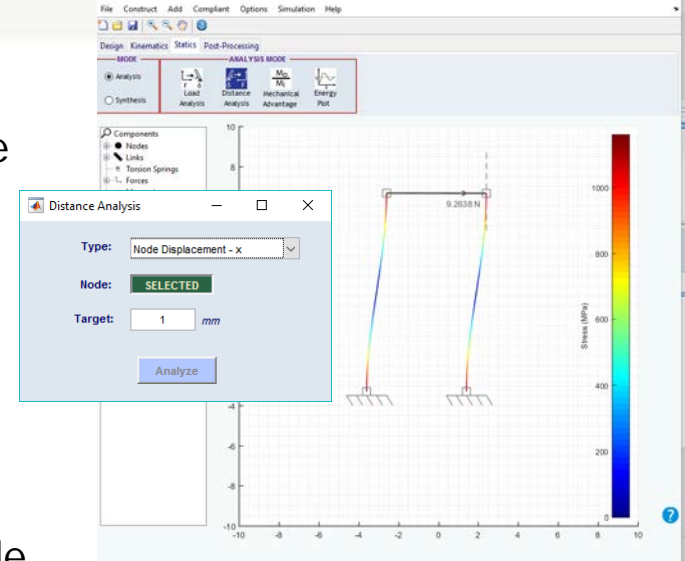


- Load analysis performs the kinetostatic analysis.
- Load can be incrementally increased.
- Lower range for the load must be bigger than or equal to 0%.
- Upper range can be set to any value as long as it is greater than the lower value.
- During the analysis, a progress bar displaying the current increment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

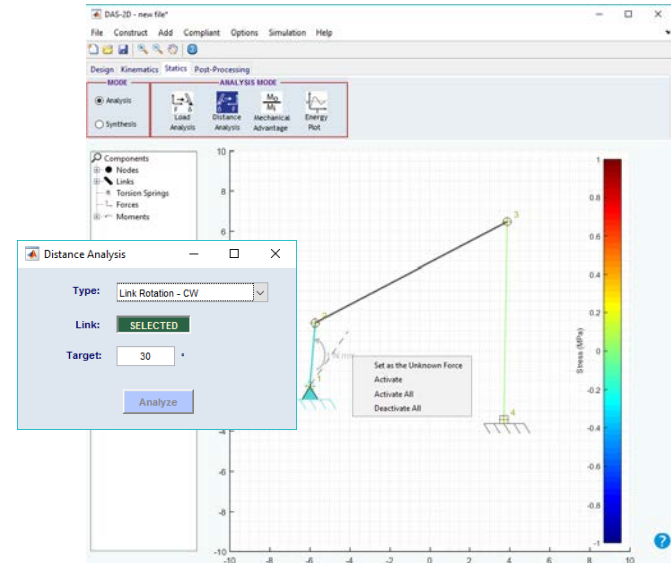


Load Analysis

- Distance analysis 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.



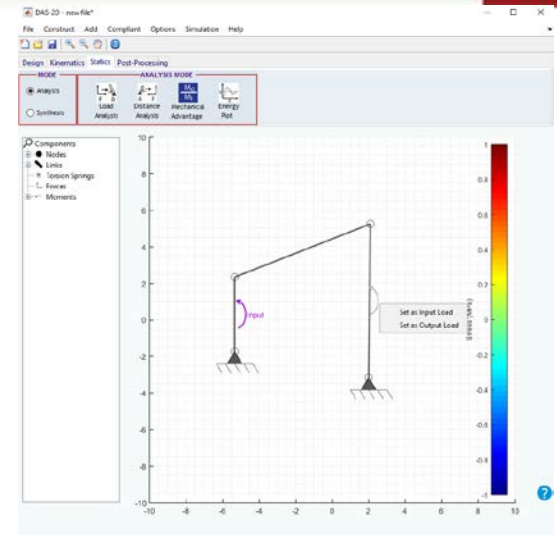
Distance Analysis - Node Displacement



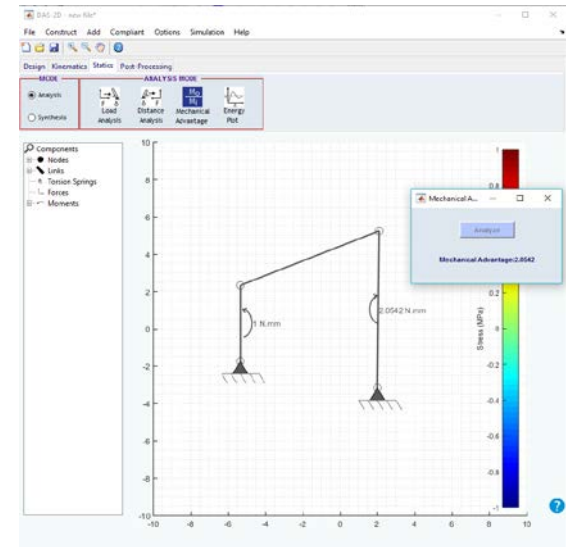
Distance Analysis - Link Rotation



- Mechanical advantage 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.



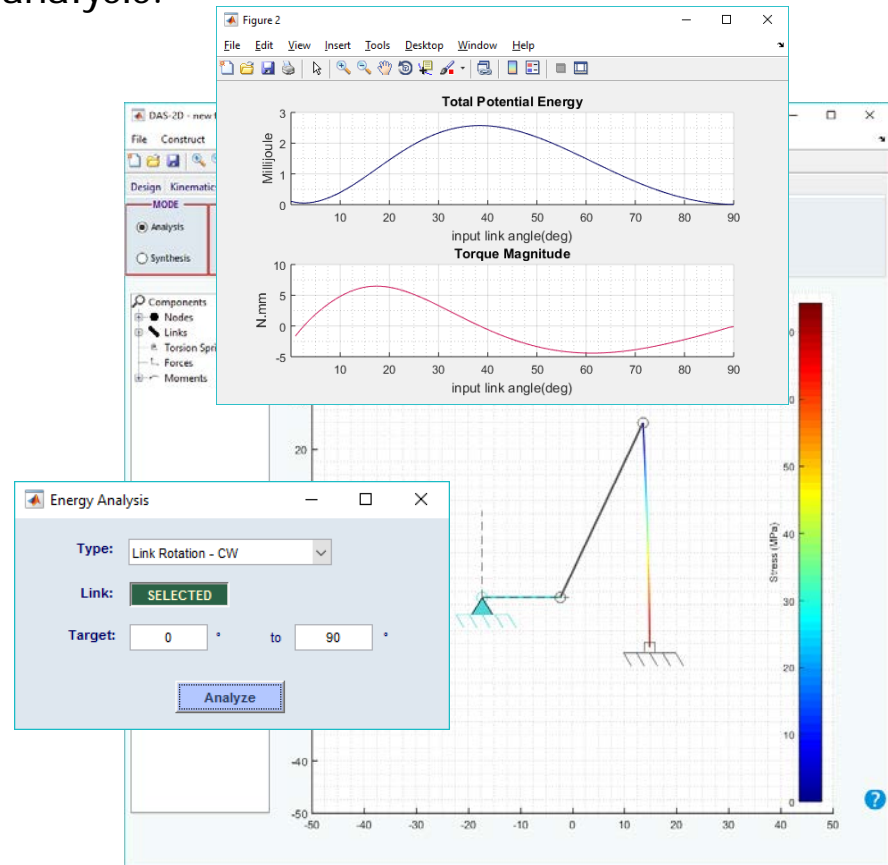
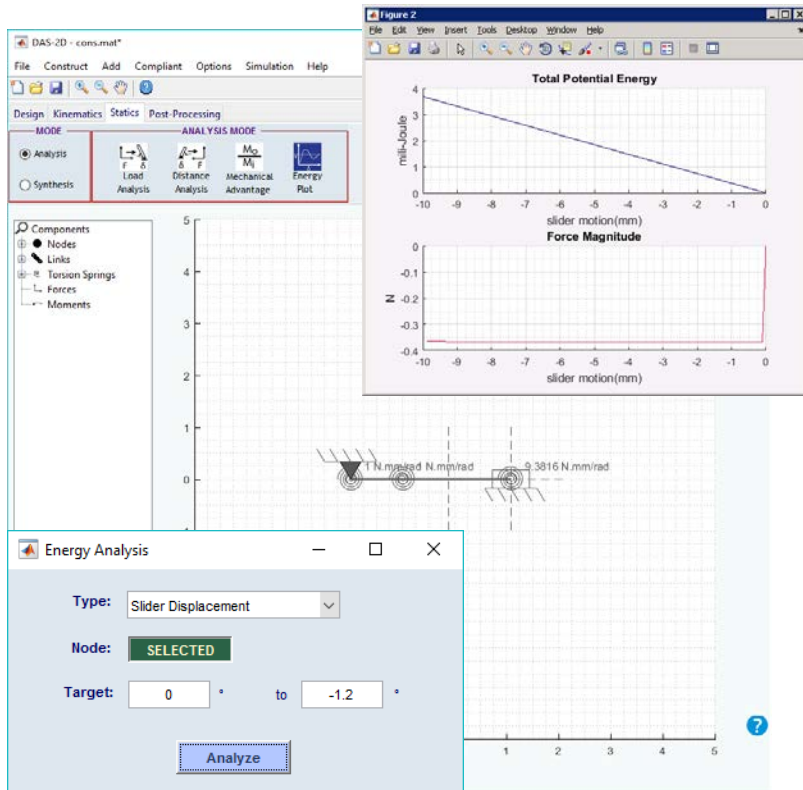
Mechanical Advantage - Select Loads



Mechanical Advantage - After Analysis



- Energy Plot 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.



Kinetostatic Analysis Demonstration

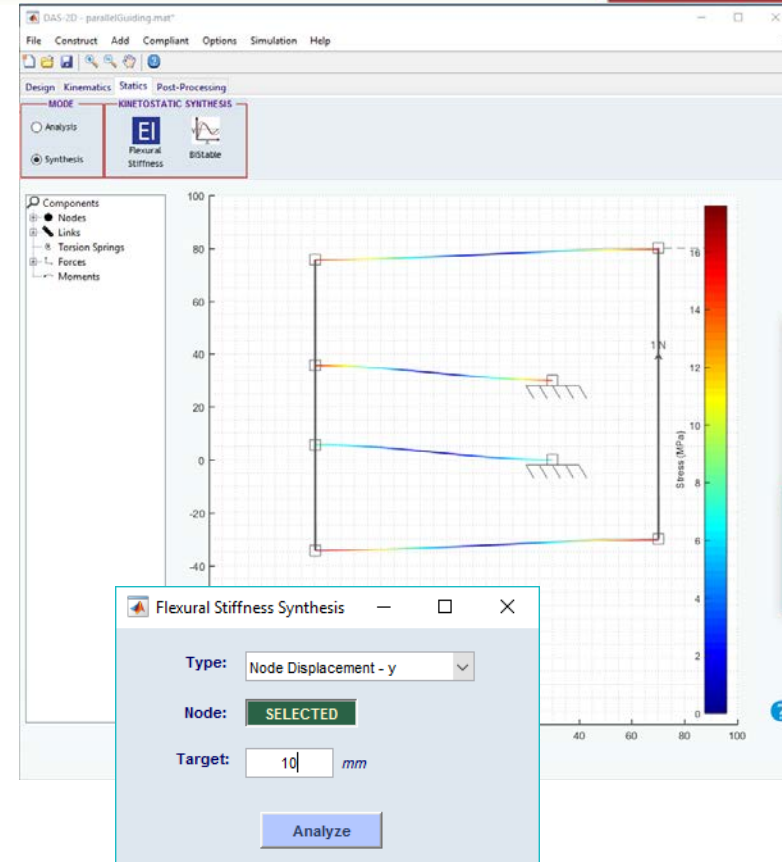
Ex_LoadAnalysis.mat

Ex_DistanceAnalysis.mat

Ex_MA.mat

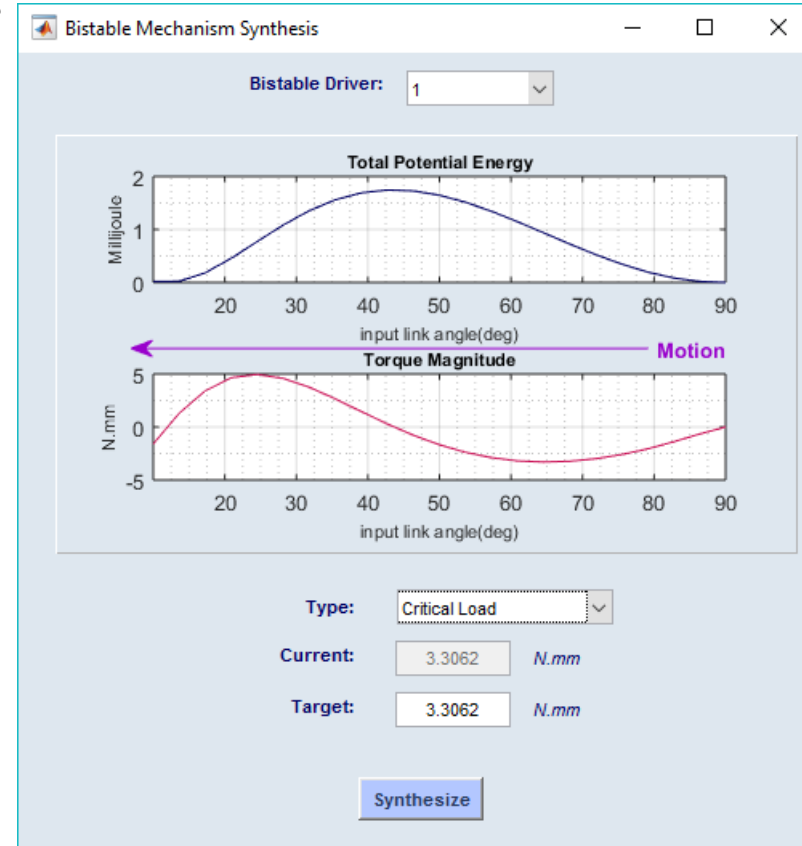
Ex_Energy.mat

- 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

- 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

The screenshot shows the DAS-2D:Post-Processing software interface. The main window is titled "DAS-2D:Post-Processing".

- Session Type:** Located at the top, it includes radio buttons for "Kinematics", "Load Analysis" (selected), "Distance Analysis", "Mechanical Advantage", and "Energy Analysis". Below these are "Session:" (1) and "Run:" (1) dropdown menus, and an "Export" button with a dropdown set to "Excel File".
- Workspace:** The central area contains a 2D plot of a mechanical linkage with four nodes (1, 2, 3, 4) and two ground connections. A context menu is open over node 3, listing options:
 - Set Node-3 X Coordinates as X Axis
 - Set Node-3 Y Coordinates as X Axis
 - Add Node-3 X Coordinates to Y Axis
 - Add Node-3 Y Coordinates to Y Axis
- Create a Plot:** A panel on the right with "X Axis:" and "Y Axis:" input fields, a "No X Axis" button, and "New Plot" and "Show Data" (checked) buttons.
- Animation:** A panel showing "Total Frames: 11" and "Current Frame: 1". It includes "Start Frame: 1" and "End Frame: 11" with corresponding sliders, and an "Animate" button with "Show Nodes" (checked).
- Tracking:** A panel with "Tracked Nodes:" and a "Delete" button. It has two sections:
 - Existing Node:** "Node:" dropdown set to 1, with a "Track" button.
 - New Node:** "50 % Between Node" dropdown set to 1, "Node" dropdown set to 4, and a "Track" button.

At the bottom of the window, a status bar reads: "Right Click on the Nodes or on the Links to create a plot."

Session Type

Workspace

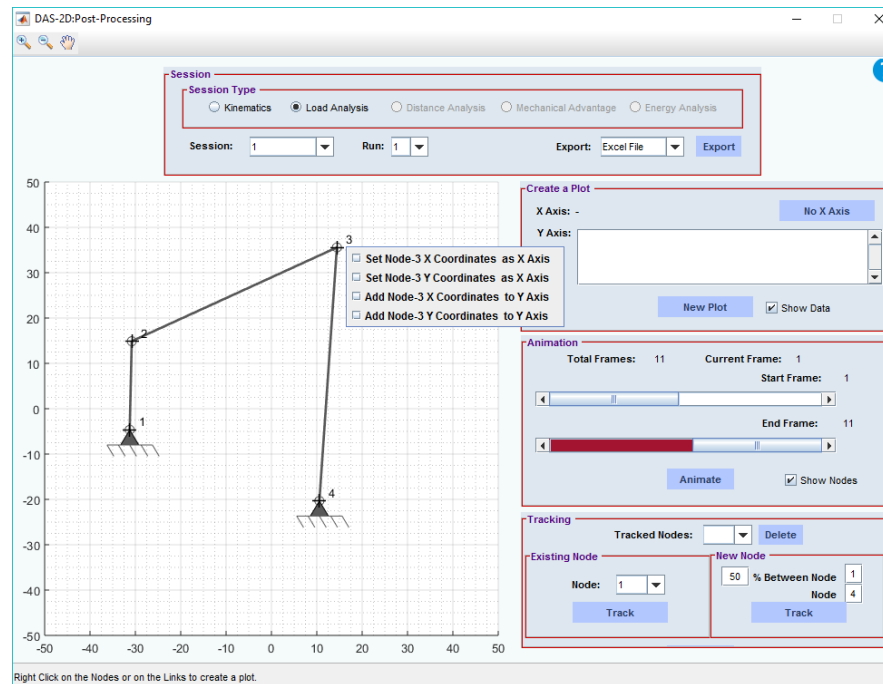
Create a Plot

Animate

Track a Node



- 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

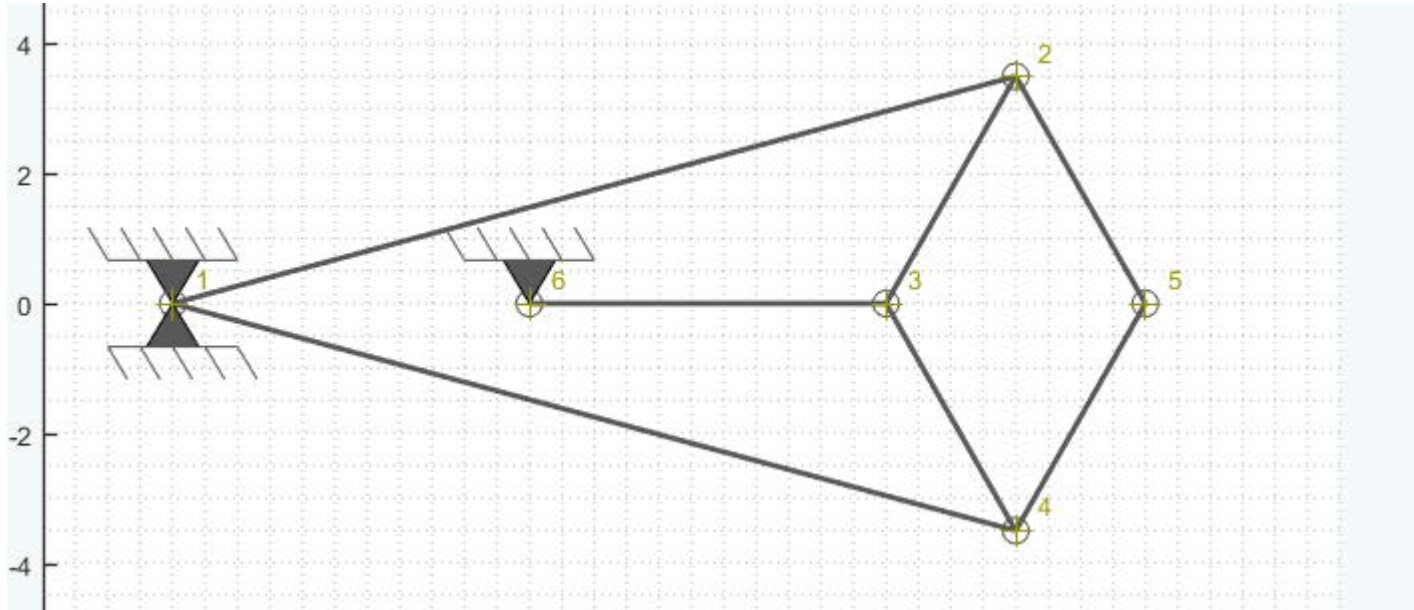




You can send comments and requests for DAS 2D to turkkan.1@osu.edu

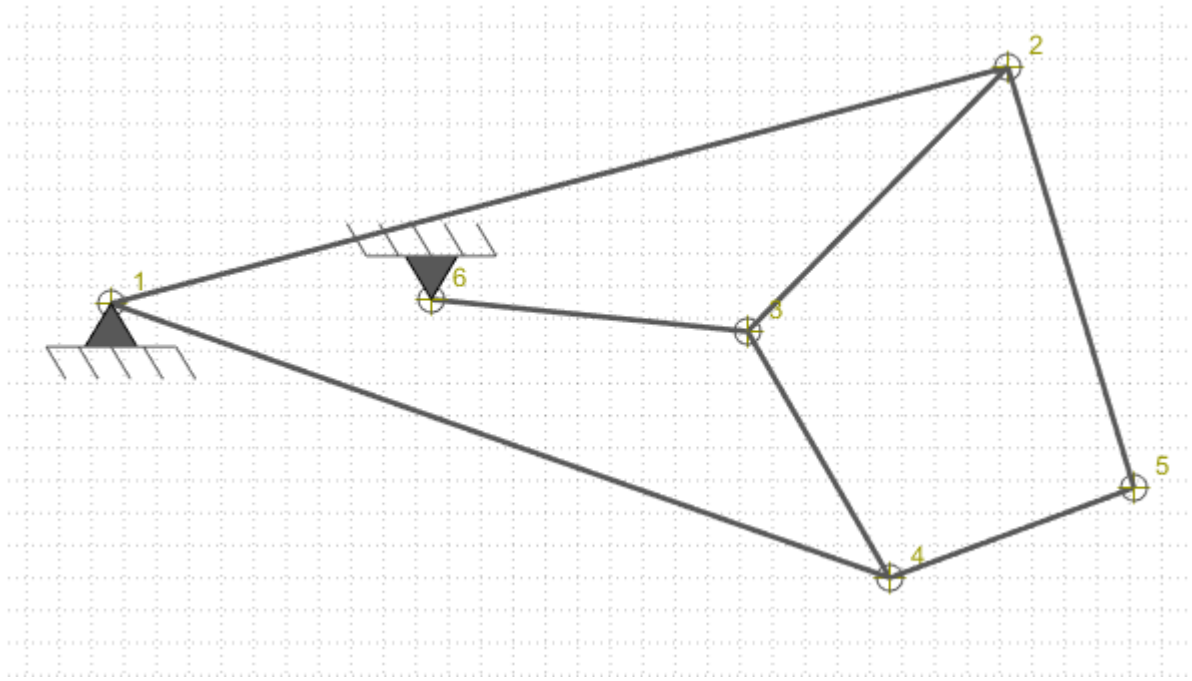
The first true straight line mechanism

Start by setting workspace size 10 by 10 from Options-> Workspace

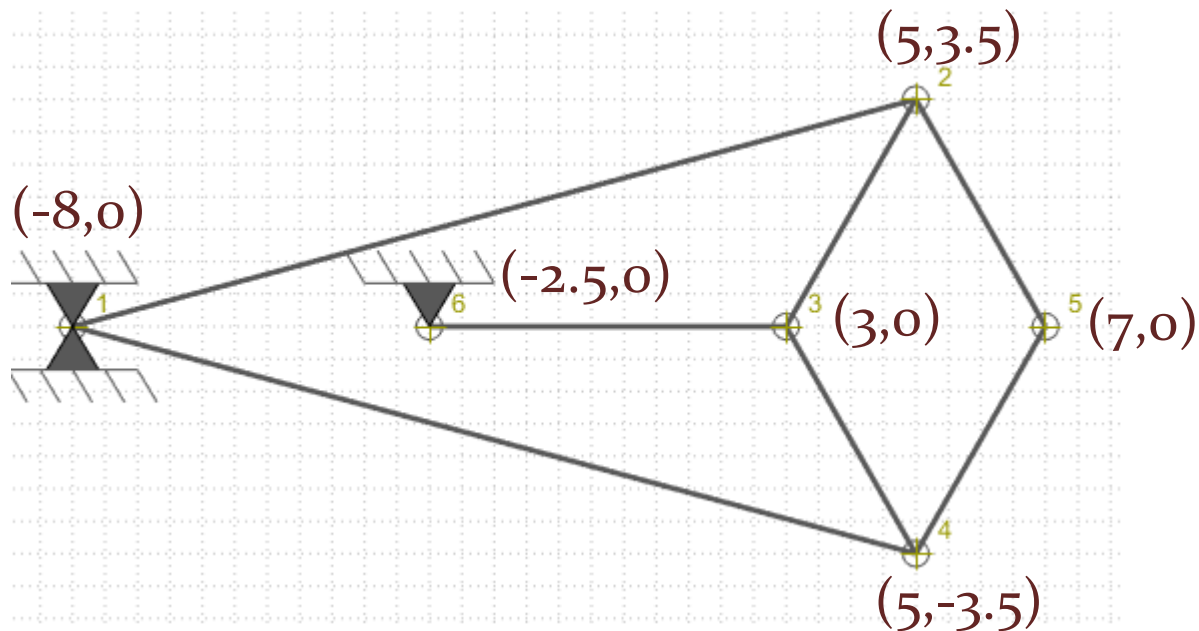


Step 1. Go to sketch mode and draw the mechanism. You will need to press Esc time to time.

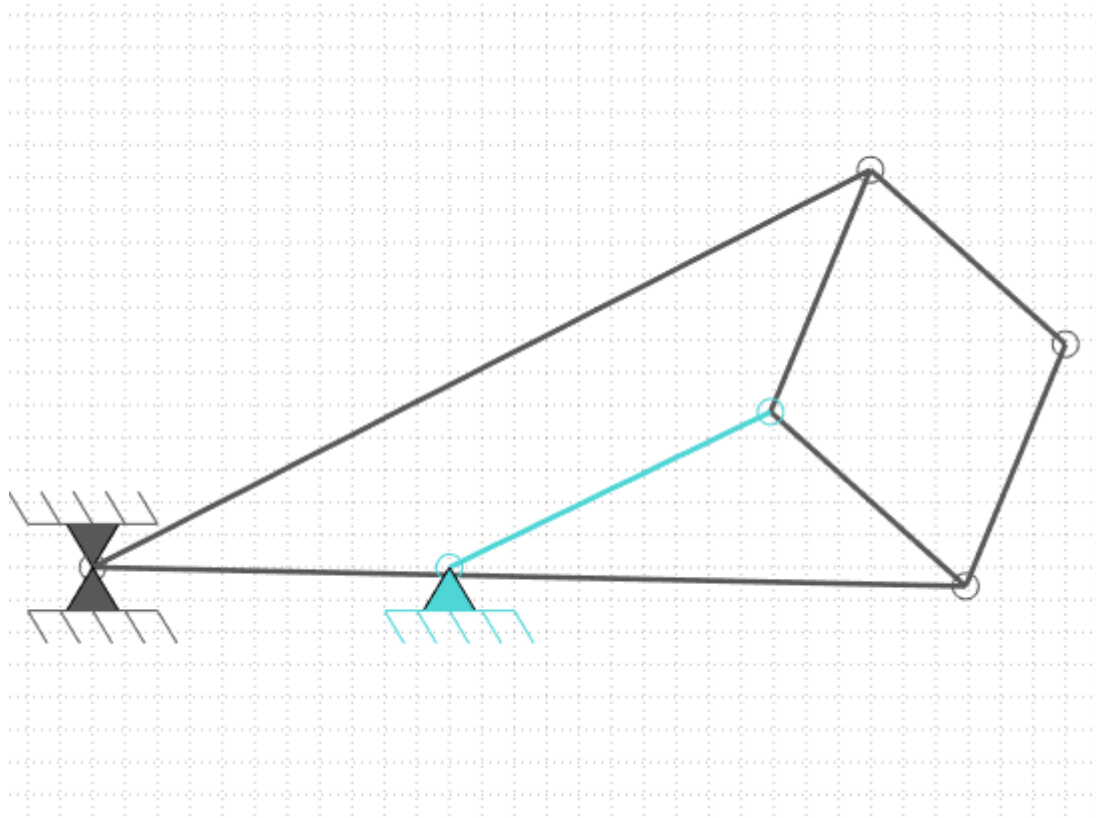
Step 2. Go to Model Module and click the links that have different joint types compared to the mechanism below.



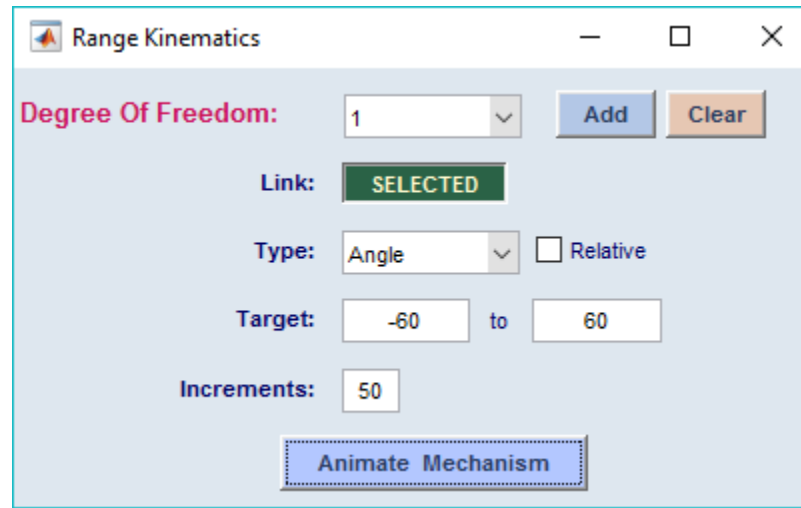
Step 3. Either go to Overview mode and right click nodes and select Edit or go to dimensions and click nodes one by one. Enter the coordinates below.



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.



Range Kinematics

Degree Of Freedom: 1 Add Clear

Link: SELECTED

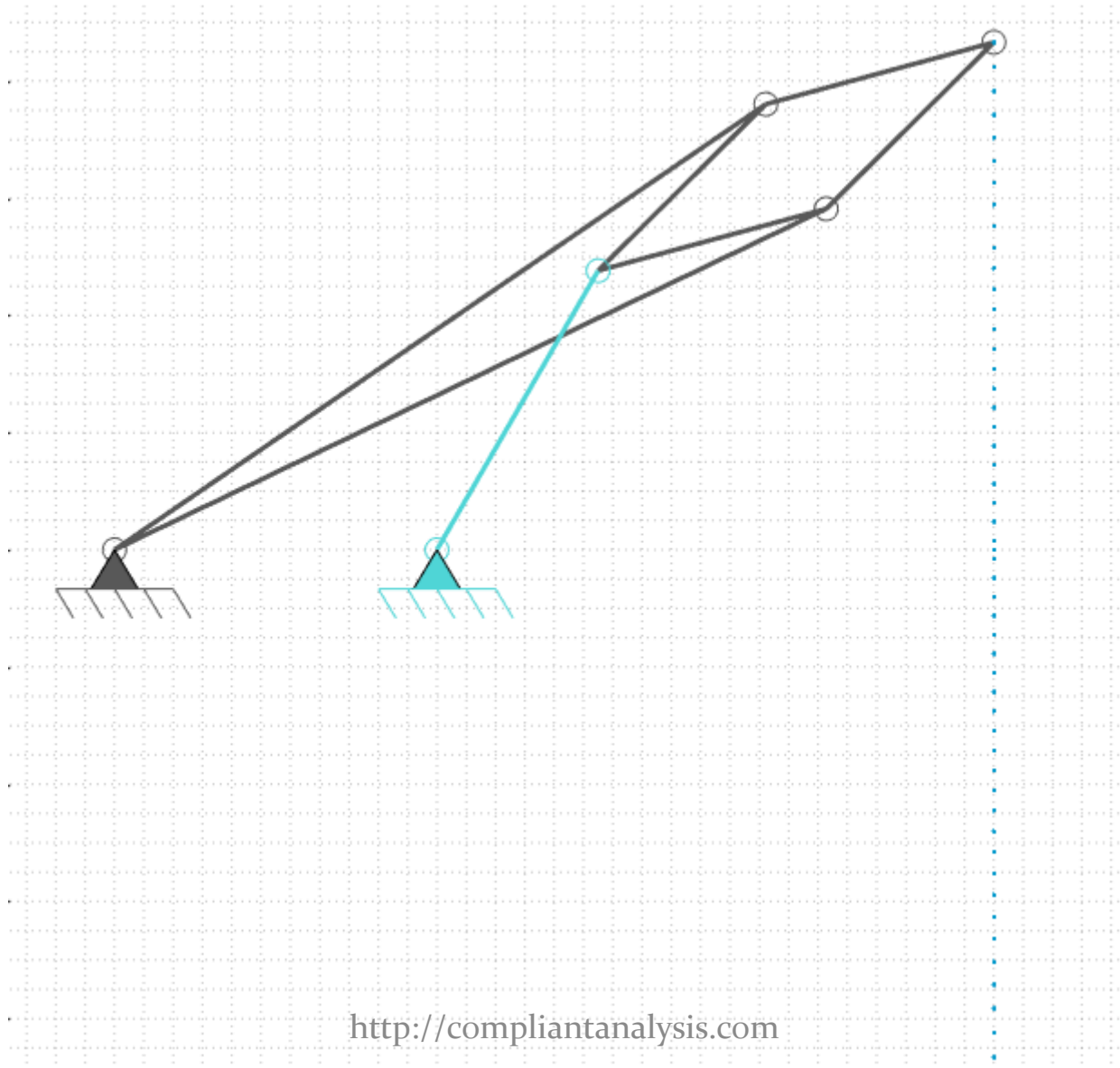
Type: Angle Relative

Target: -60 to 60

Increments: 50

Animate Mechanism

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.



<http://compliantanalysis.com>

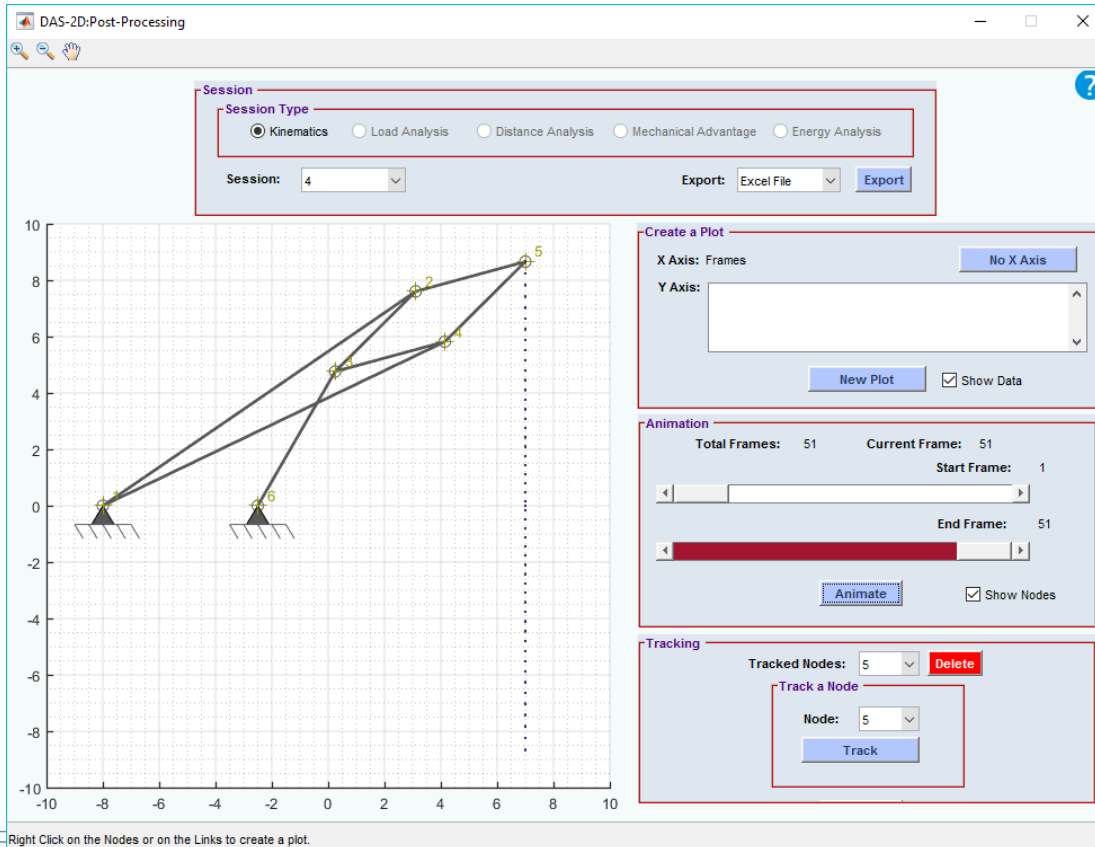
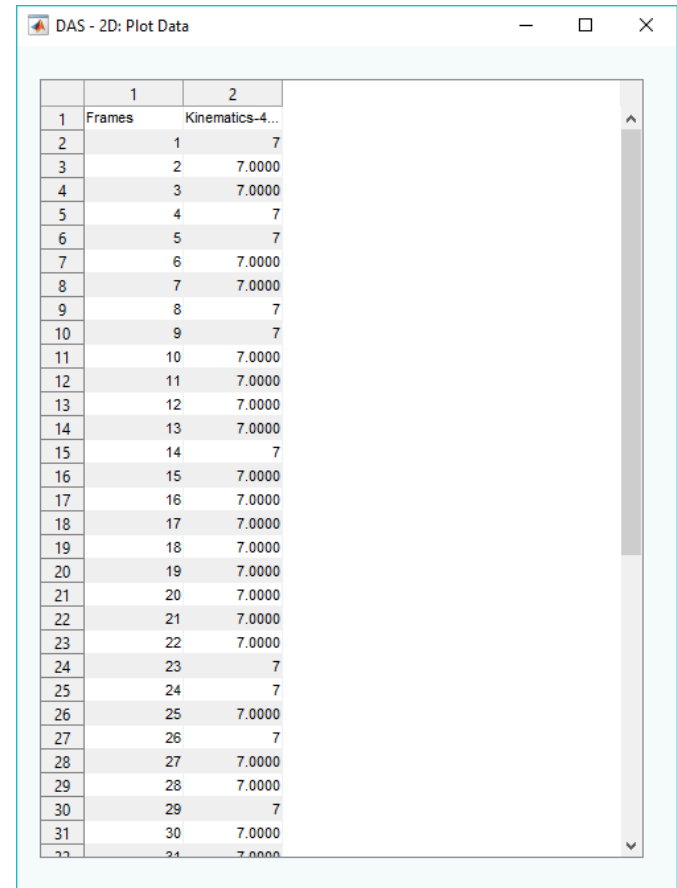


Step 8. Go to Post-Processing Module

Step 9. From down right corner select the correct node to track

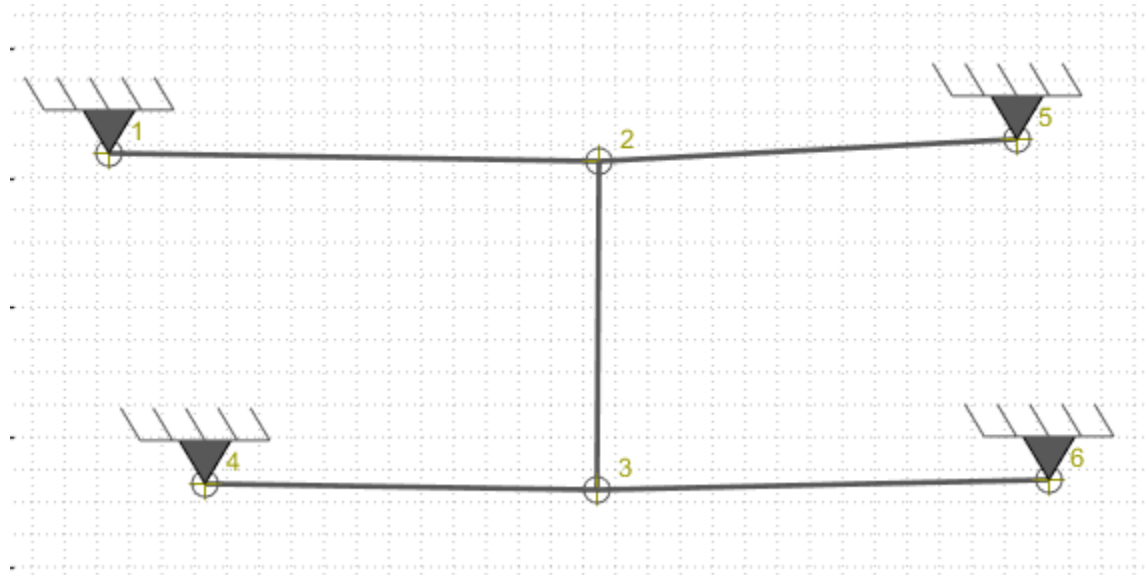
Step 10. Hit Animate

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

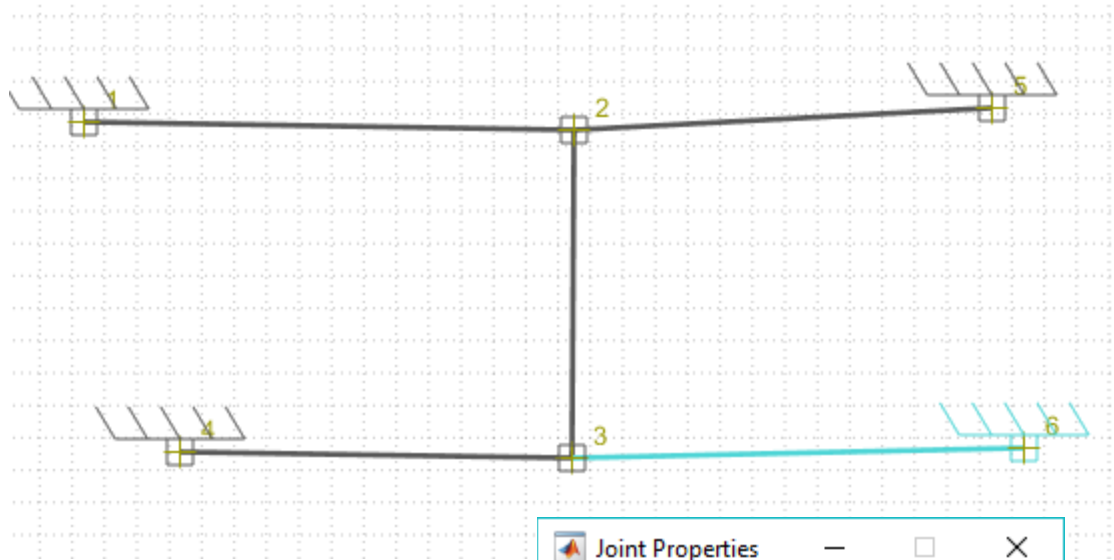



| | 1 | 2 |
|----|--------|-----------------|
| 1 | Frames | Kinematics-4... |
| 2 | 1 | 7 |
| 3 | 2 | 7.0000 |
| 4 | 3 | 7.0000 |
| 5 | 4 | 7 |
| 6 | 5 | 7 |
| 7 | 6 | 7.0000 |
| 8 | 7 | 7.0000 |
| 9 | 8 | 7 |
| 10 | 9 | 7 |
| 11 | 10 | 7.0000 |
| 12 | 11 | 7.0000 |
| 13 | 12 | 7.0000 |
| 14 | 13 | 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 |
| 32 | 31 | 7.0000 |

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



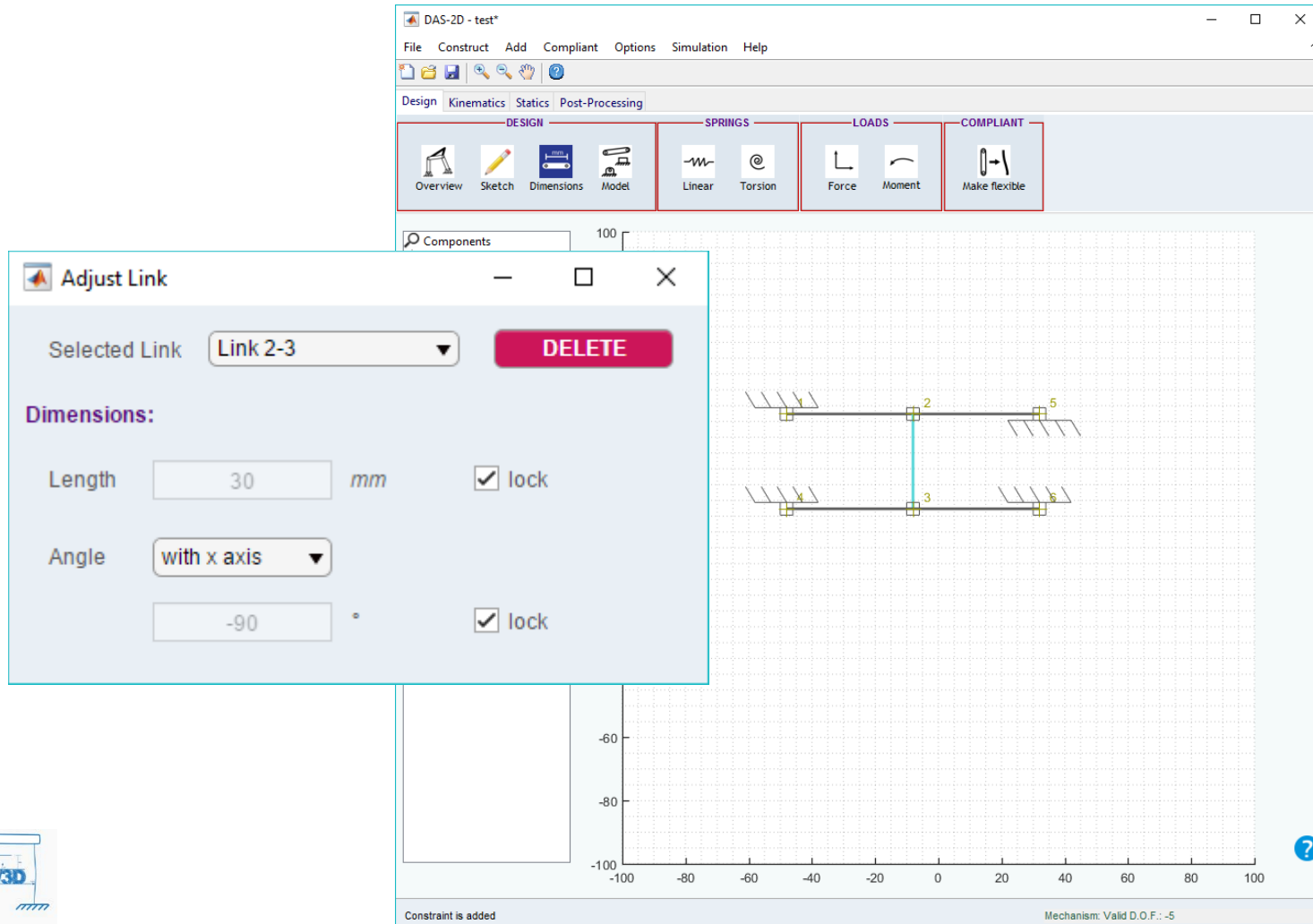
Step 2. Go to Model Module and click links one by one and set joints as welded joint.



Joint Properties — □ ×

| | | |
|-----------------|---|---|
| Link: | Link 3-6 ▼ | Valid Link |
| Joint 3: | Welded Joint ▼ | <input type="checkbox"/> ground |
| Joint 6: | Welded Joint ▼ | <input checked="" type="checkbox"/> ground |

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.

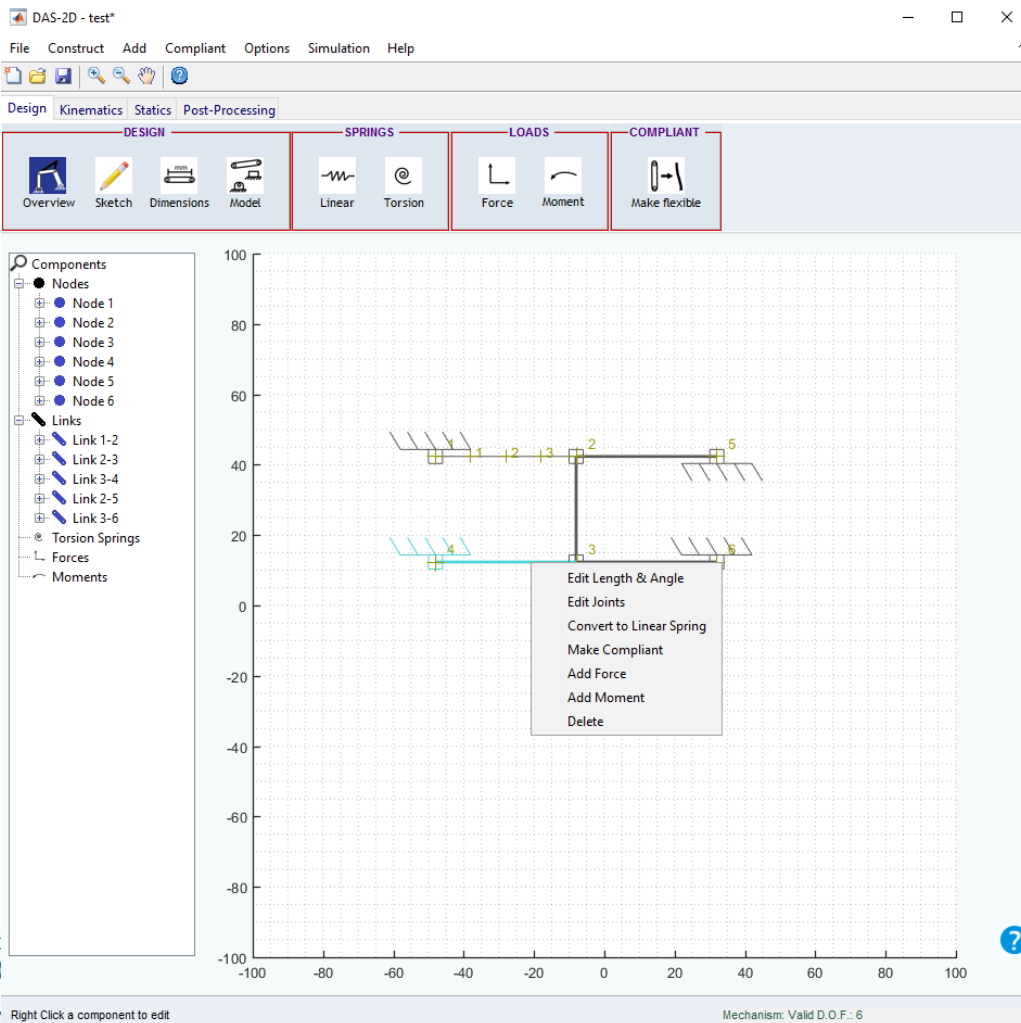


The screenshot displays the DAS-2D software interface. The main window shows a mechanism diagram with joints labeled 1 through 6. A coordinate system is visible at the bottom with x and y axes ranging from -100 to 100. The 'Adjust Link' dialog box is open, showing the following settings:

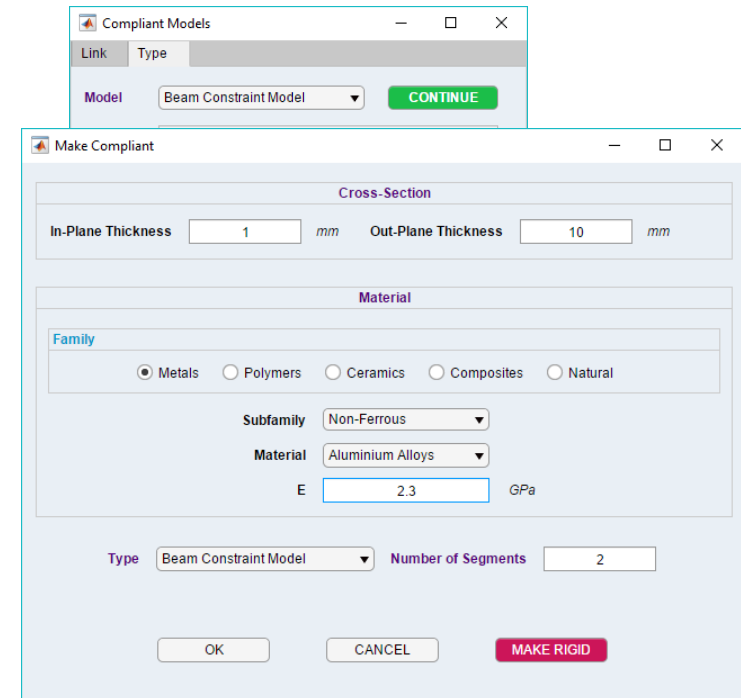
- Selected Link: Link 2-3
- Length: 30 mm (checked lock)
- Angle: with x axis (checked lock)
- Angle value: -90 °

The software interface includes a menu bar (File, Construct, Add, Compliant, Options, Simulation, Help) and a toolbar with icons for Design, Kinematics, Statics, and Post-Processing. The Design toolbar includes icons for Overview, Sketch, Dimensions, Model, Linear, Torsion, Force, Moment, and Make flexible.

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.

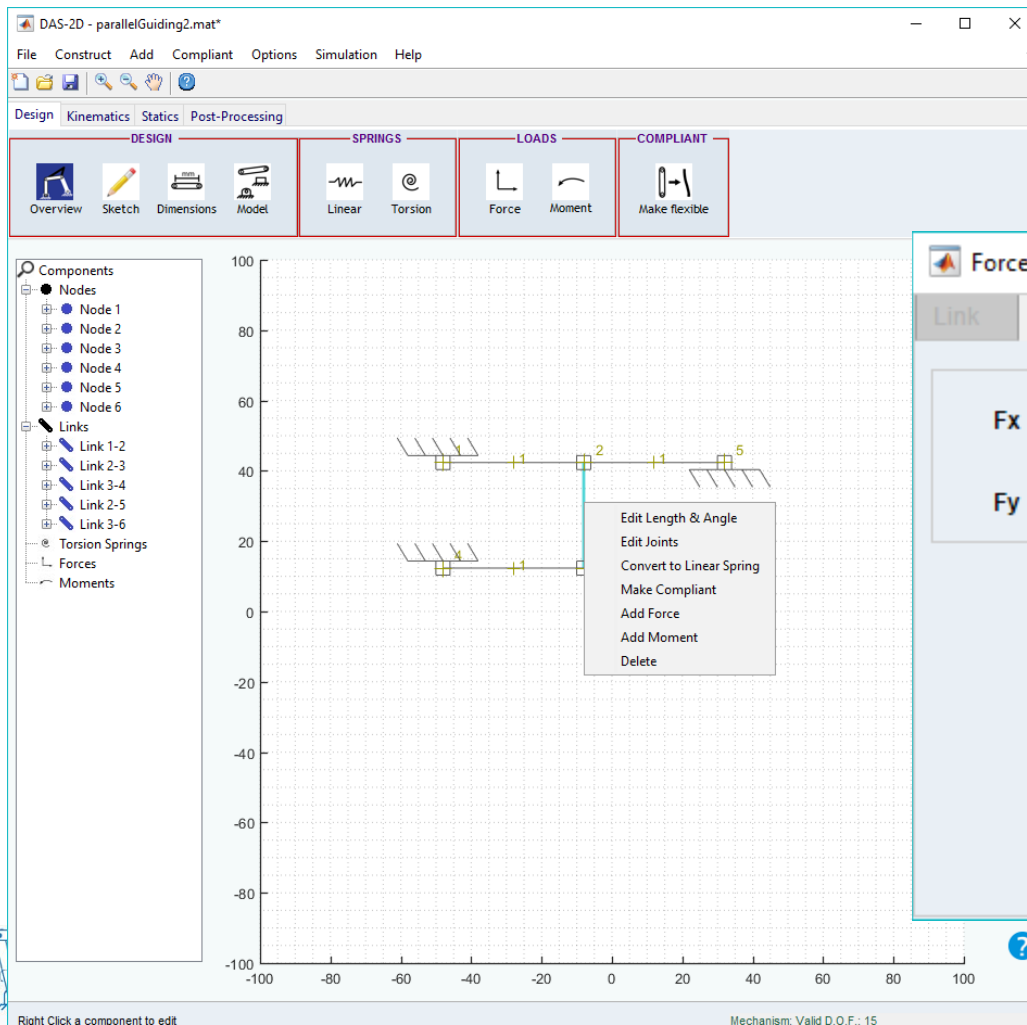


The screenshot shows the software interface for a mechanism simulation. The main window displays a 2D mechanism diagram on a grid. The diagram consists of six nodes (Node 1 to Node 6) and six links (Link 1-2, Link 2-3, Link 3-4, Link 2-5, Link 3-6). The mechanism is a parallelogram mechanism. A context menu is open over the mechanism, listing options: Edit Length & Angle, Edit Joints, Convert to Linear Spring, Make Compliant, Add Force, Add Moment, and Delete. The 'Make Compliant' option is highlighted. The left sidebar shows the 'Components' tree with 'Nodes' and 'Links' listed. The top menu bar includes 'File', 'Construct', 'Add', 'Compliant', 'Options', 'Simulation', and 'Help'. The bottom status bar indicates 'Mechanism: Valid D.O.F.: 6'.

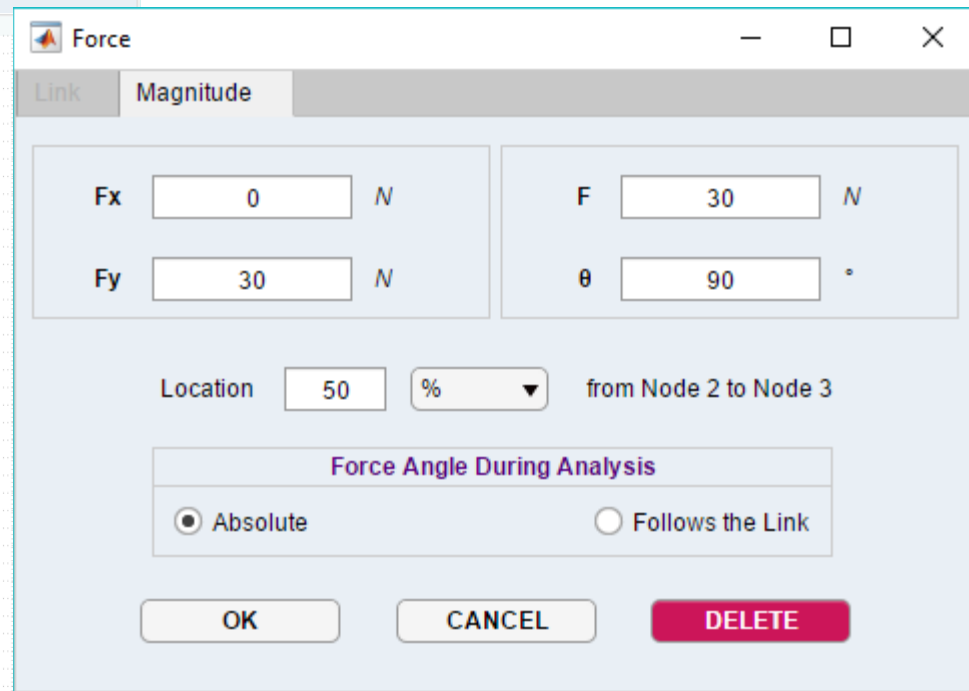


The screenshot shows the 'Make Compliant' dialog box. The 'Cross-Section' section has 'In-Plane Thickness' set to 1 mm and 'Out-Plane Thickness' set to 10 mm. The 'Material' section has 'Family' set to 'Metals', 'Subfamily' set to 'Non-Ferrous', and 'Material' set to 'Aluminium Alloys'. The 'E' (Young's Modulus) is set to 2.3 GPa. The 'Type' is set to 'Beam Constraint Model' and the 'Number of Segments' is set to 2. The 'MAKE RIGID' button is highlighted in red.

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



The screenshot shows the DAS-2D software interface. The main window displays a 2D coordinate system with a parallelogram mechanism. The mechanism consists of six nodes and five links. A context menu is open over the middle link (Link 2-3), listing options: Edit Length & Angle, Edit Joints, Convert to Linear Spring, Make Compliant, Add Force, Add Moment, and Delete. The 'Add Force' option is highlighted. The 'Components' panel on the left shows the hierarchy of nodes, links, torsion springs, forces, and moments. The status bar at the bottom indicates 'Mechanism: Valid D.O.F.: 15'.

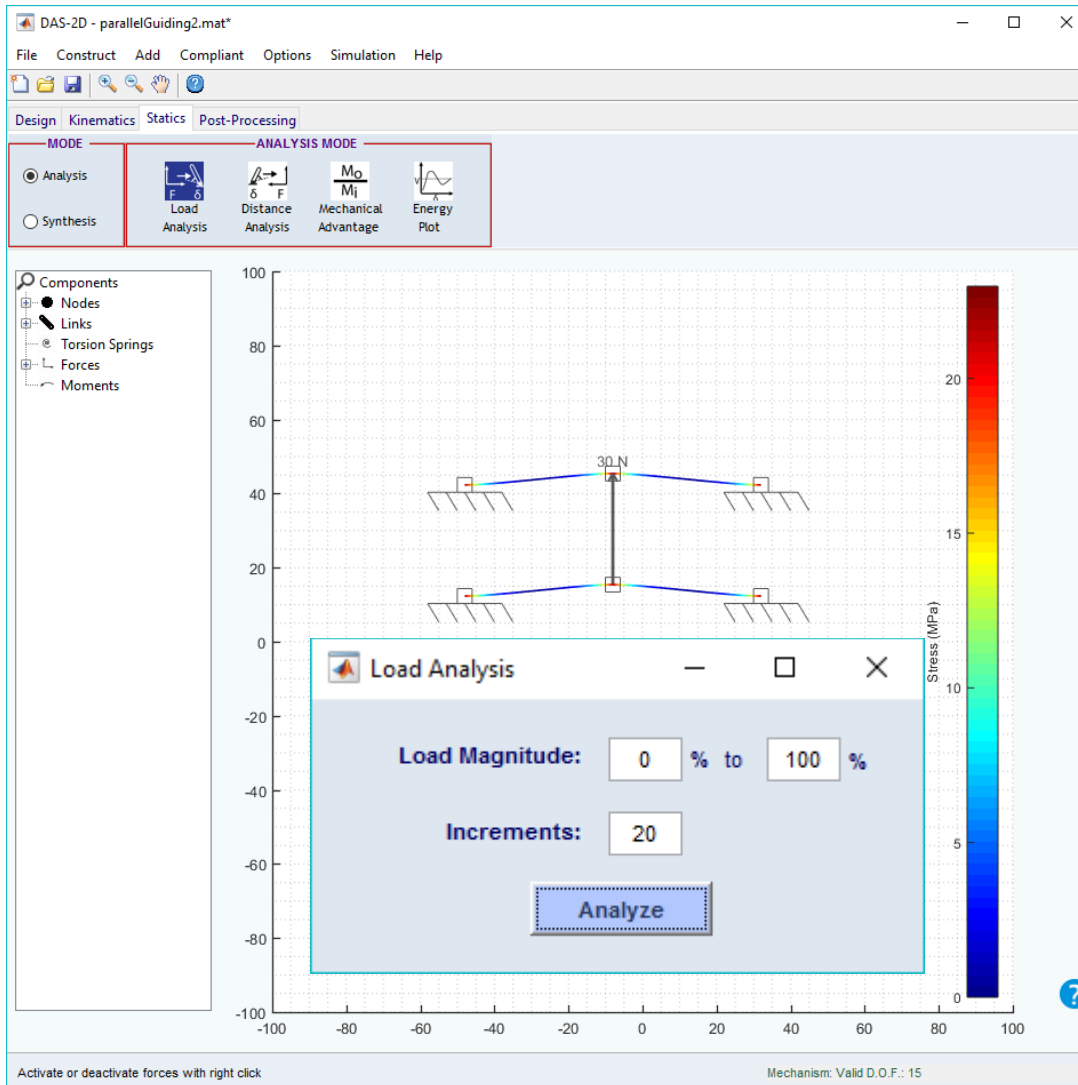


The 'Force' dialog box is shown, allowing configuration of a force applied to a link. The 'Link' tab is selected, and the 'Magnitude' section is active. The force components are set as follows:

| Component | Value | Unit |
|------------------|-------|----------|
| F_x | 0 | N |
| F_y | 30 | N |
| F (Magnitude) | 30 | N |
| θ (Angle) | 90 | $^\circ$ |

The 'Location' is set to 50% from Node 2 to Node 3. The 'Force Angle During Analysis' is set to 'Absolute'. The dialog includes 'OK', 'CANCEL', and 'DELETE' buttons.

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

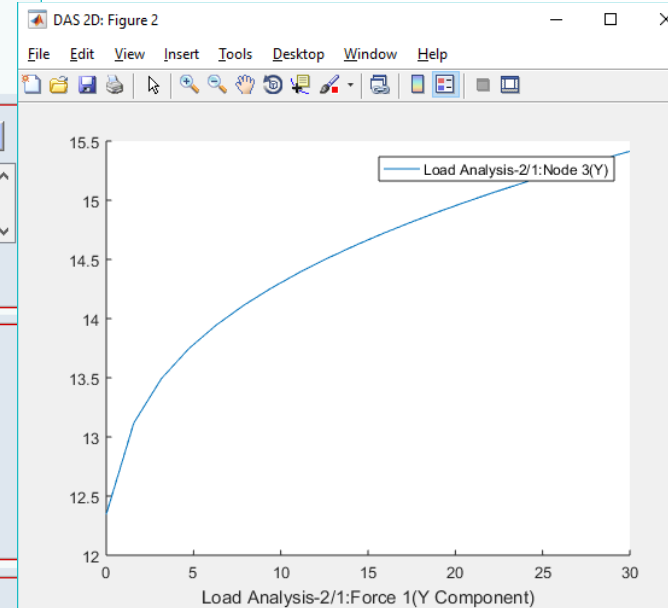
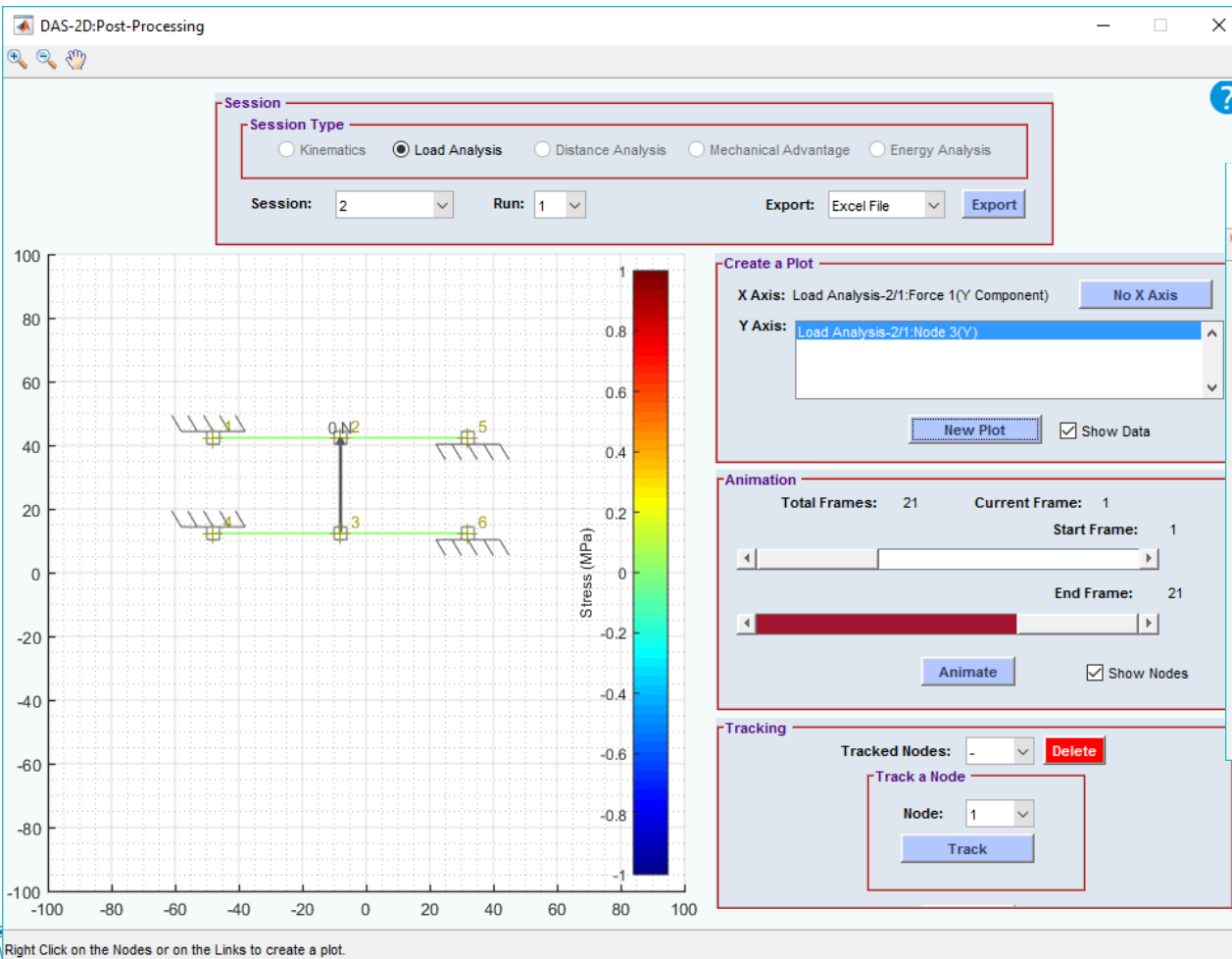


The screenshot shows the DAS-2D software interface. The main window displays a stress plot of a parallelogram mechanism. The plot shows a central vertical link with a 30 N force applied at the top. The mechanism is supported by four fixed supports. A color scale on the right indicates stress in MPa, ranging from 0 (blue) to 20 (red). A dialog box titled "Load Analysis" is open in the foreground, showing the following settings:

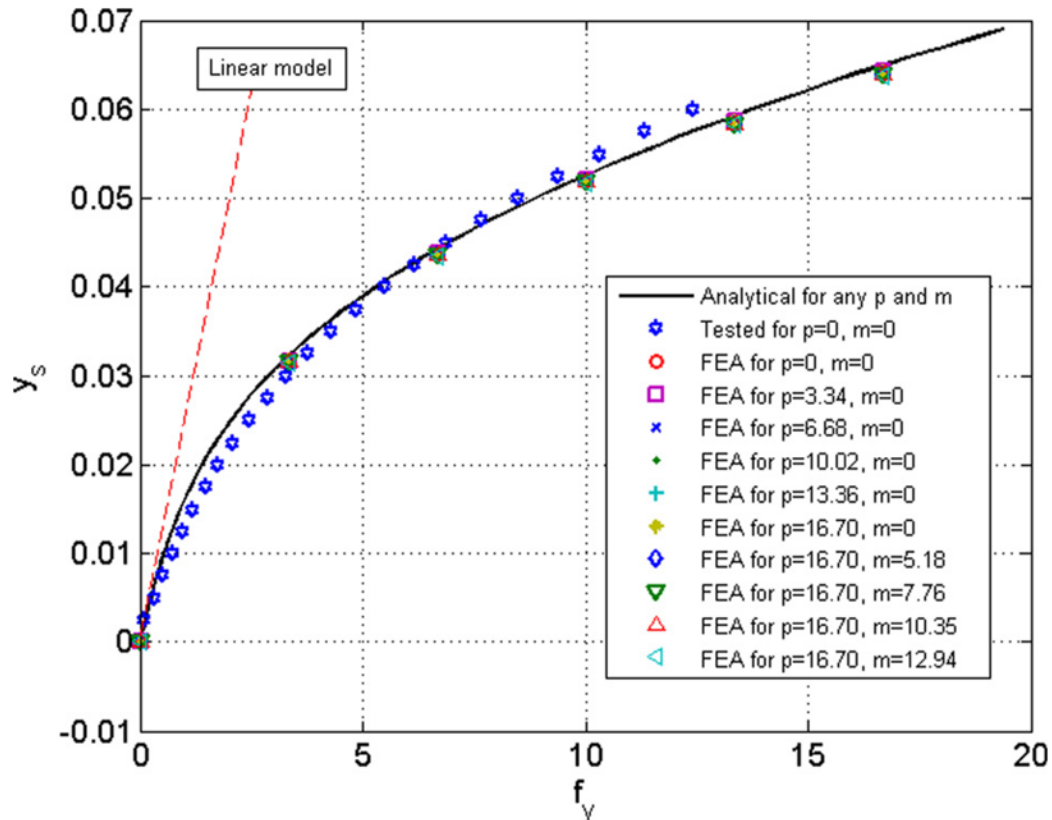
- Load Magnitude: 0 % to 100 %
- Increments: 20
- Analyze button

The software interface includes a menu bar (File, Construct, Add, Compliant, Options, Simulation, Help) and a toolbar. The "Statics" tab is selected, and the "Load Analysis" mode is active. The "Load Analysis" dialog box is currently open, allowing the user to set the load magnitude and increments.

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.

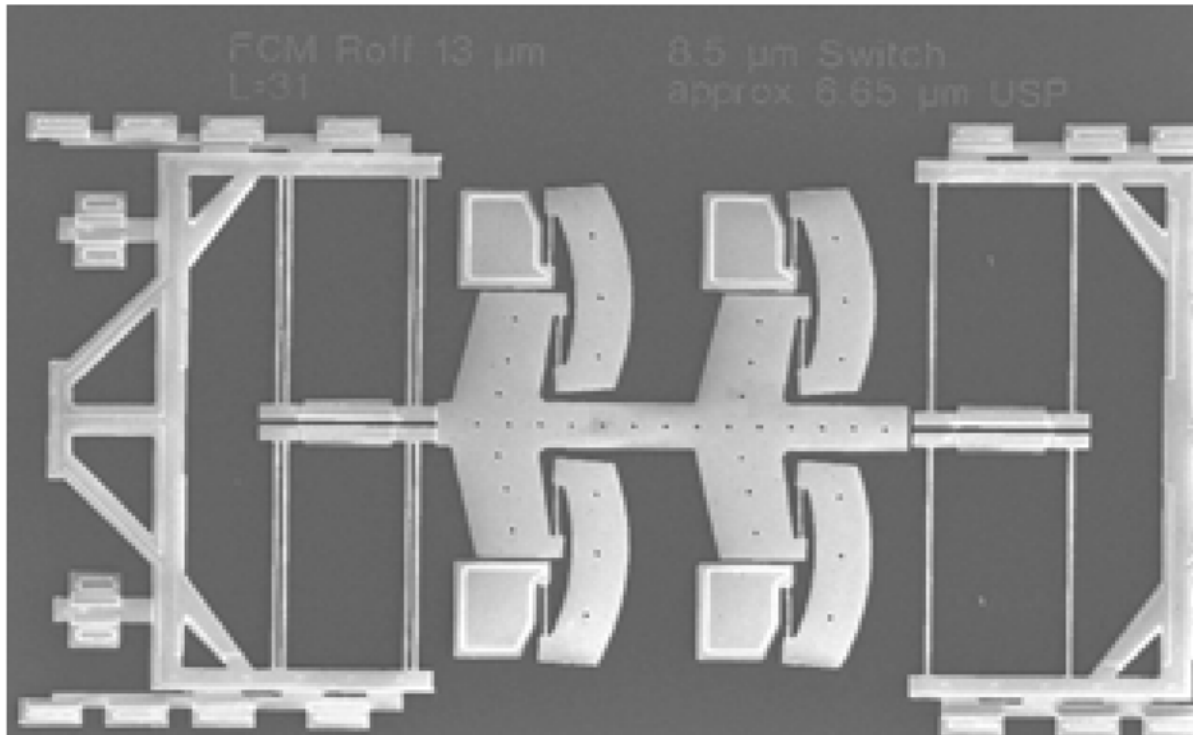


Observe that multiple BCM can capture increasing stiffness with Y .



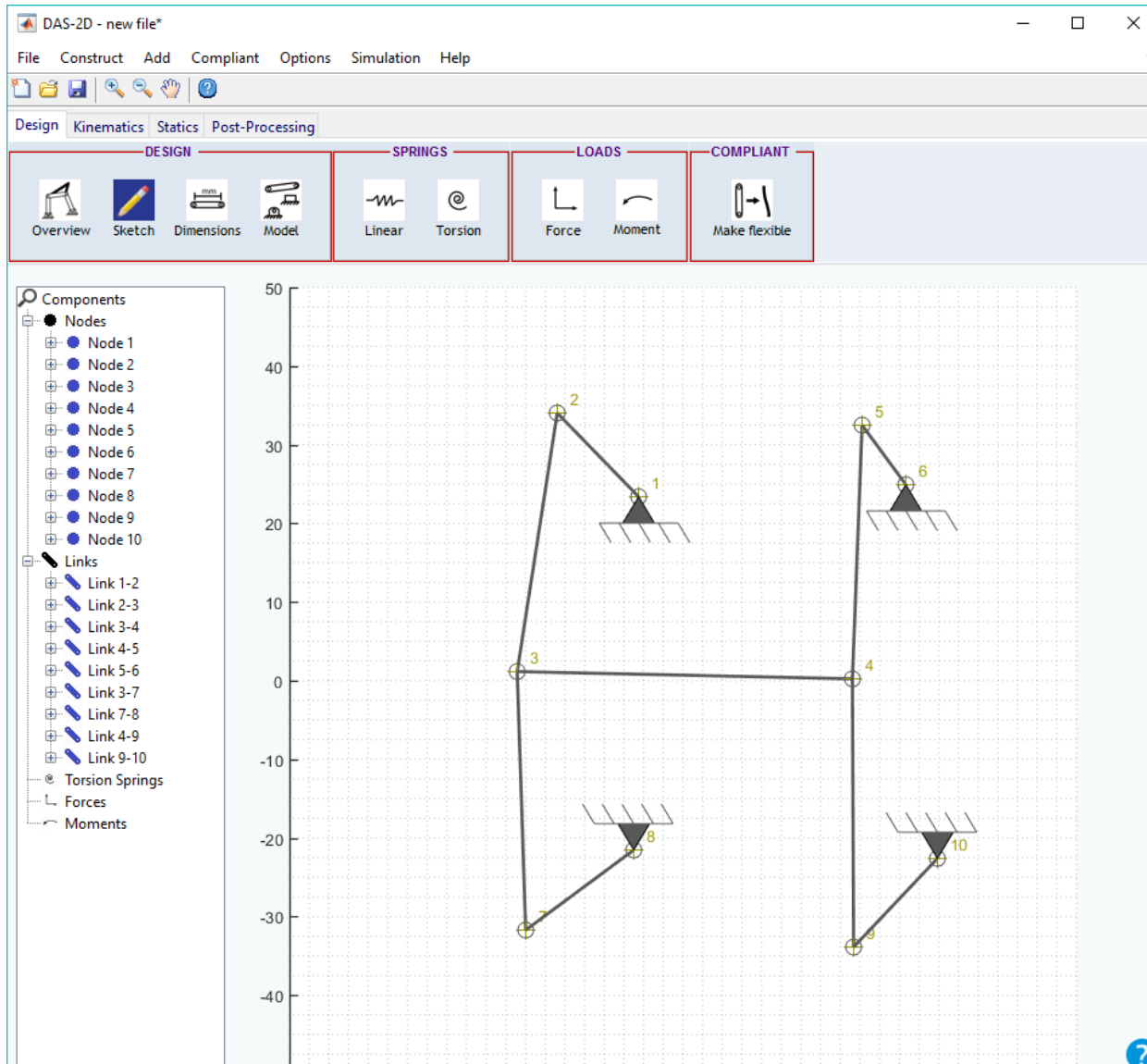
Hao, Guangbo, and Haiyang Li. "Nonlinear Analytical Modeling and Characteristic Analysis of a Class of Compound Multibeam Parallelogram Mechanisms." *Journal of Mechanisms and Robotics* 7, no. 4 (November 1, 2015): 041016-041016. doi:10.1115/1.4029556.

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

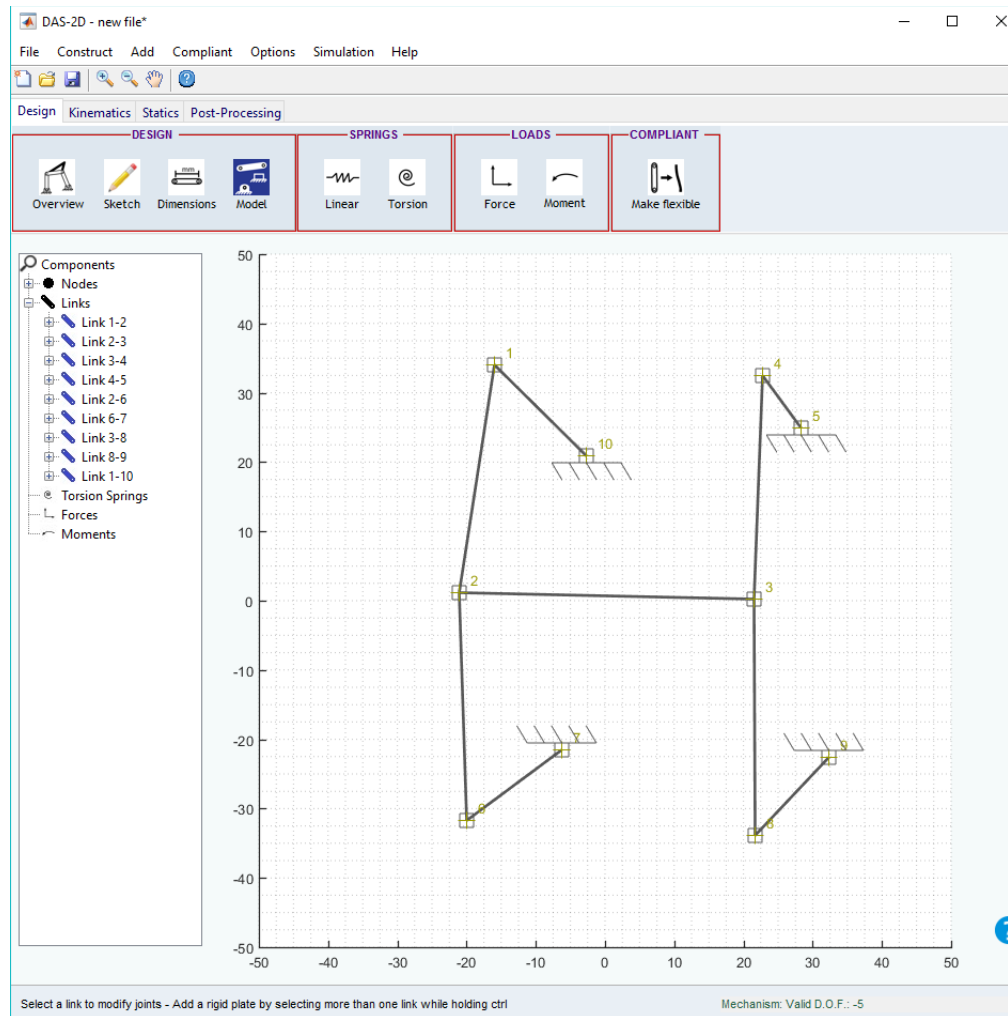


Masters, Nathan D., and Larry L. Howell. "A Three Degree-of-Freedom Model for Self-Retracting Fully Compliant Bistable Micromechanisms." *Journal of Mechanical Design* 127, no. 4 (June 27, 2005): 739-44. doi:10.1115/1.1828463.

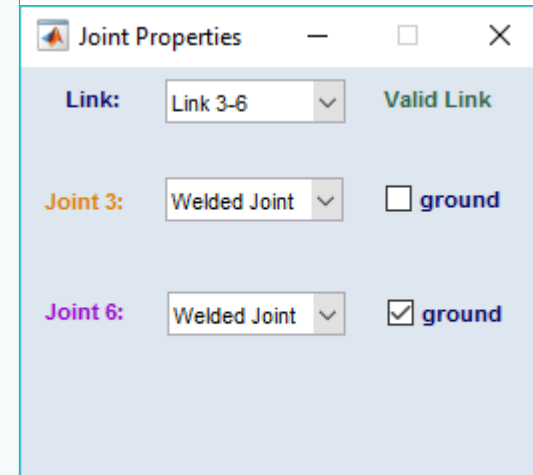
Step 1. Sketch the mechanism as shown below



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



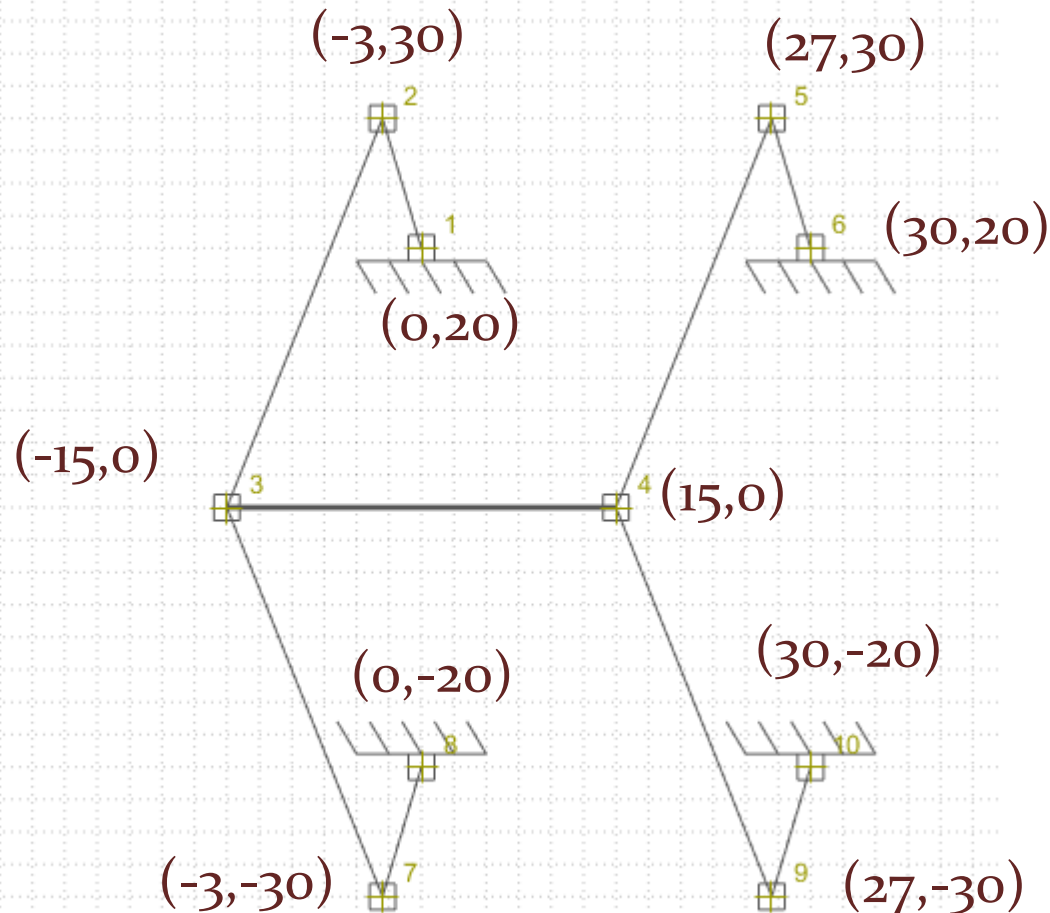
The screenshot shows the DAS-2D software interface. The main window displays a mechanism diagram on a coordinate grid. The diagram consists of several links connected at 10 numbered joints (1-10). Joints 1, 2, 3, 4, 5, 6, 7, 8, 9, and 10 are marked with small squares. The mechanism is a bistable switch mechanism. The left sidebar shows the 'Components' tree with 'Links' expanded, listing Link 1-2 through Link 1-10. The top menu bar includes 'File', 'Construct', 'Add', 'Compliant', 'Options', 'Simulation', and 'Help'. The top toolbar has icons for 'Overview', 'Sketch', 'Dimensions', 'Model', 'Linear', 'Torsion', 'Force', 'Moment', and 'Make flexible'. The bottom status bar reads 'Select a link to modify joints - Add a rigid plate by selecting more than one link while holding ctrl' and 'Mechanism: Valid D.O.F.: -5'.



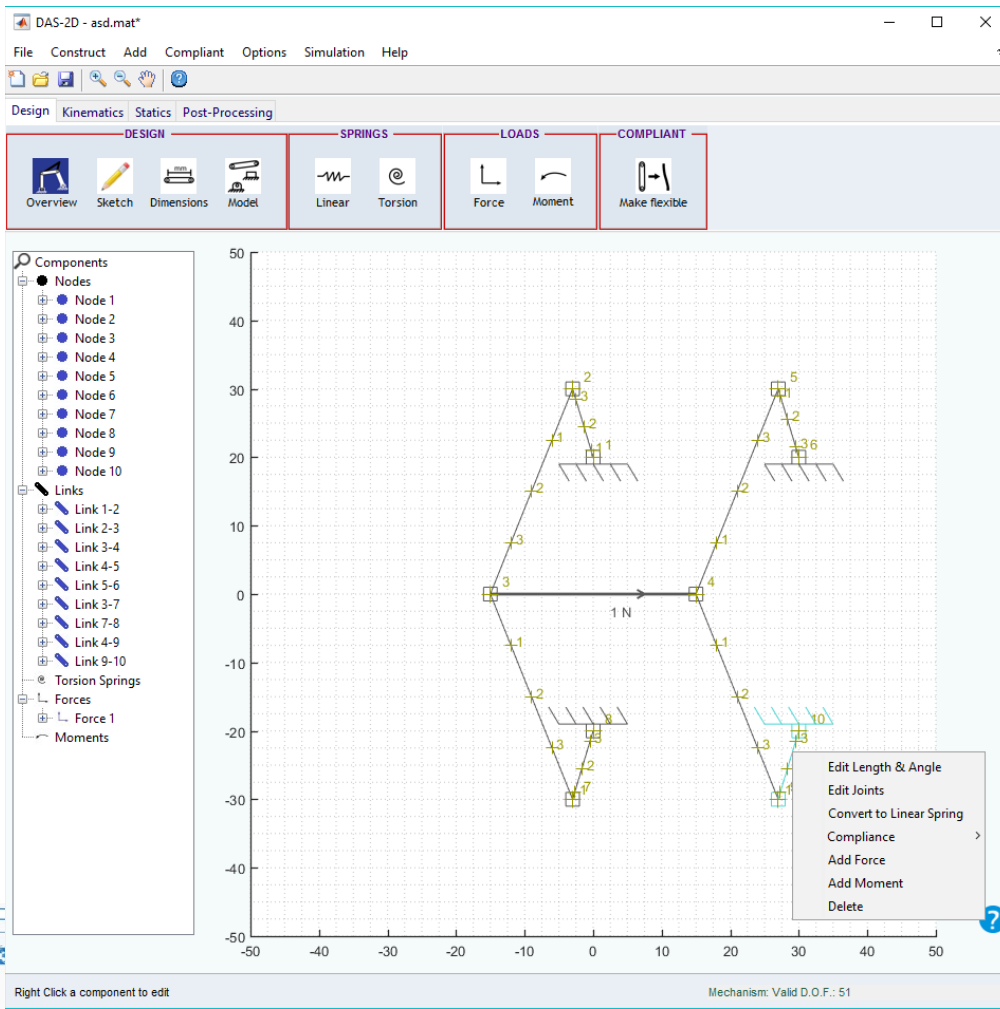
The 'Joint Properties' dialog box is shown, detailing the configuration for three joints. The 'Link' dropdown is set to 'Link 3-6', which is marked as a 'Valid Link'. 'Joint 3' is set to 'Welded Joint' and is not grounded. 'Joint 6' is also set to 'Welded Joint' and is grounded, as indicated by the checked 'ground' checkbox.

| Joint | Link | Joint Type | Grounded |
|----------|--------------|--------------|--|
| Link 3-6 | Link 3-6 | Valid Link | - |
| Joint 3 | Welded Joint | Welded Joint | <input type="checkbox"/> ground |
| Joint 6 | Welded Joint | Welded Joint | <input checked="" type="checkbox"/> ground |

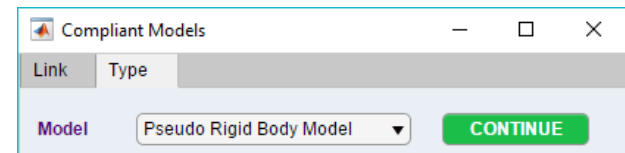
Step 3. Either go to Overview mode and right click nodes and select Edit or go to dimensions and click nodes one by one. Enter the coordinates below.



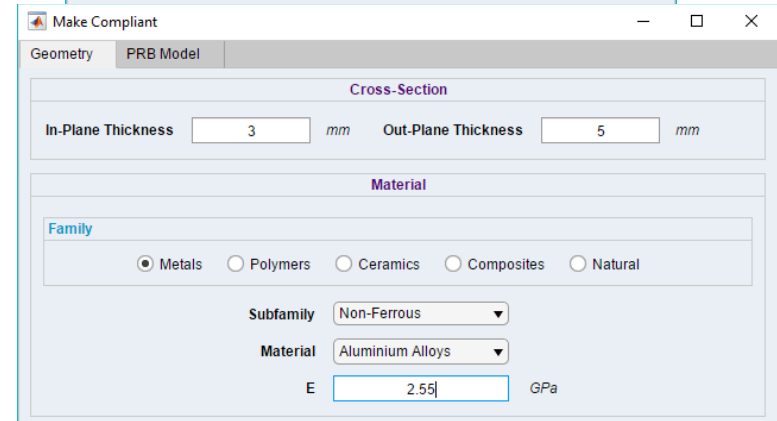
Step 4. Go to Overview Module. Right Click short links and select make compliant. Choose PRB and hit continue. PRB Model can be chosen as PRB-3R. In-Plane thickness is 3 mm and out-of-plane thickness is 5 mm. Set E to 2.55 Gpa. Hit Ok.



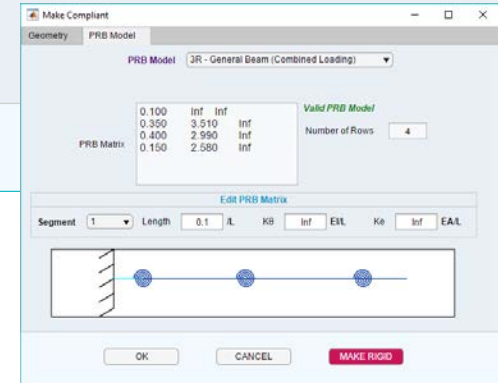
The screenshot shows the DAS-2D software interface. The main window displays a mechanism diagram with nodes (1-10) and links (1-10). A force of 1 N is applied to link 4. A context menu is open over link 10, listing options: Edit Length & Angle, Edit Joints, Convert to Linear Spring, Compliance, Add Force, Add Moment, and Delete. The 'Compliance' option is highlighted. The bottom status bar indicates 'Mechanism: Valid D.O.F.: 51'.



The 'Compliant Models' dialog box is shown. The 'Link' tab is active. The 'Model' dropdown is set to 'Pseudo Rigid Body Model'. A green 'CONTINUE' button is visible.



The 'Make Compliant' dialog box is shown. The 'PRB Model' tab is active. The 'Cross-Section' section shows 'In-Plane Thickness' set to 3 mm and 'Out-Plane Thickness' set to 5 mm. The 'Material' section shows 'Family' set to 'Metals', 'Subfamily' set to 'Non-Ferrous', and 'Material' set to 'Aluminium Alloys'. The Young's Modulus 'E' is set to 2.55 GPa.

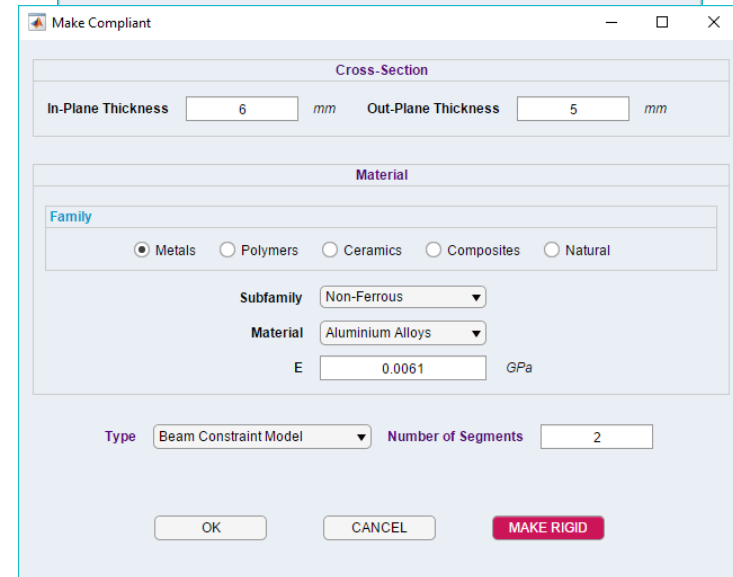
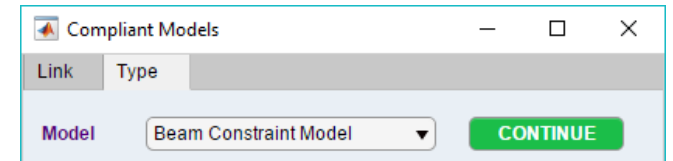
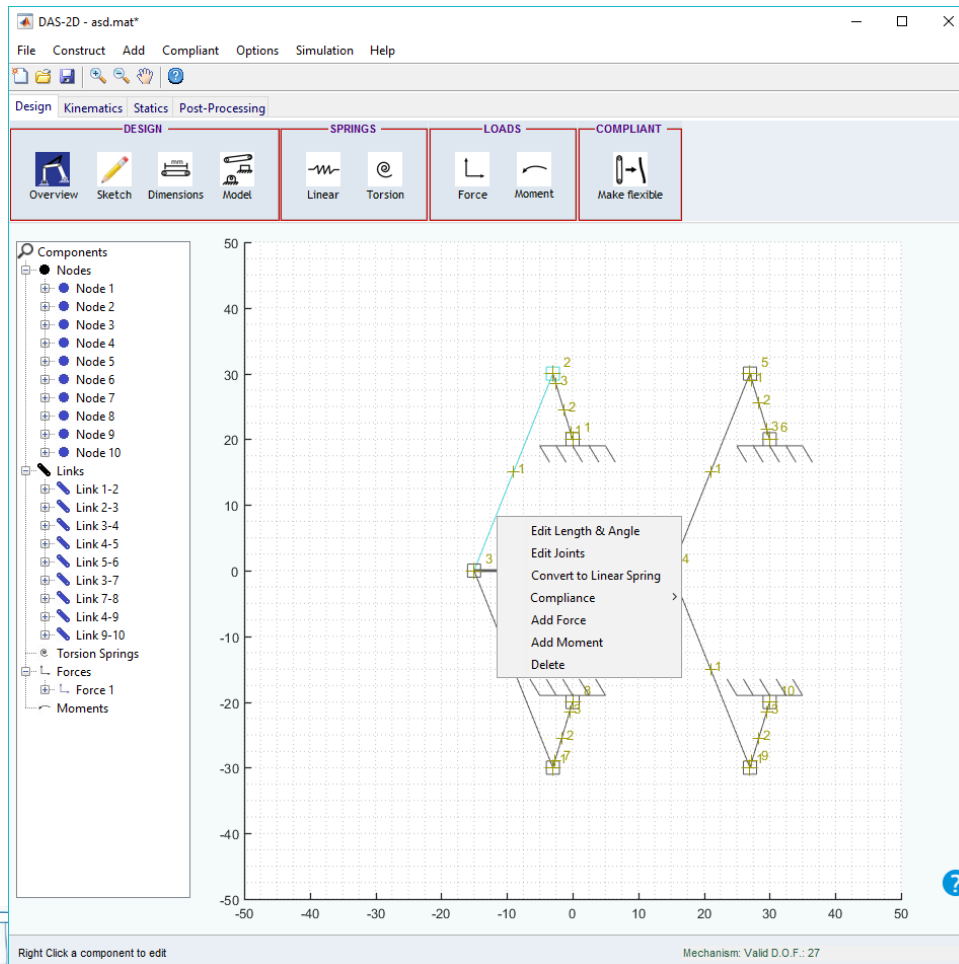


The 'Make Compliant' dialog box is shown. The 'PRB Model' tab is active. The 'PRB Model' dropdown is set to '3R - General Beam (Combined Loading)'. A 'Valid PRB Model' message is displayed. The 'PRB Matrix' table is shown with the following values:

| | | | |
|-------|-------|-----|-----------------|
| 0.100 | Inf | Inf | Valid PRB Model |
| 0.350 | 3.510 | Inf | Number of Rows |
| 0.400 | 2.990 | Inf | 4 |
| 0.150 | 2.580 | Inf | |

The 'Edit PRB Matrix' section shows a diagram of a beam with three nodes and a 'MAKE RIGID' button.

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.

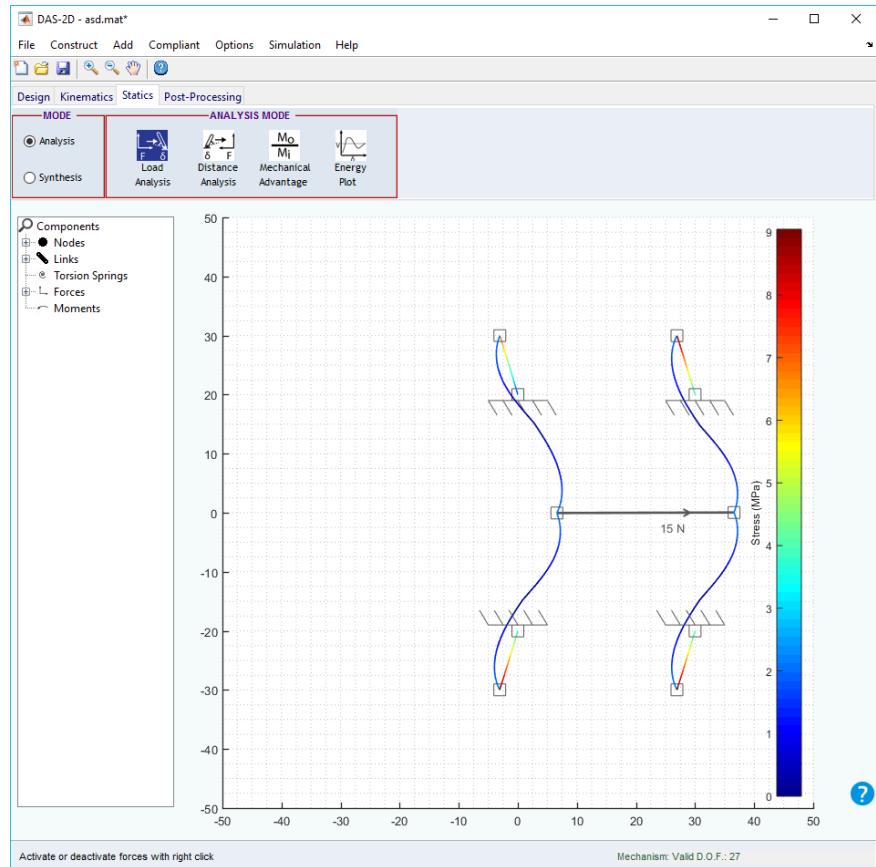
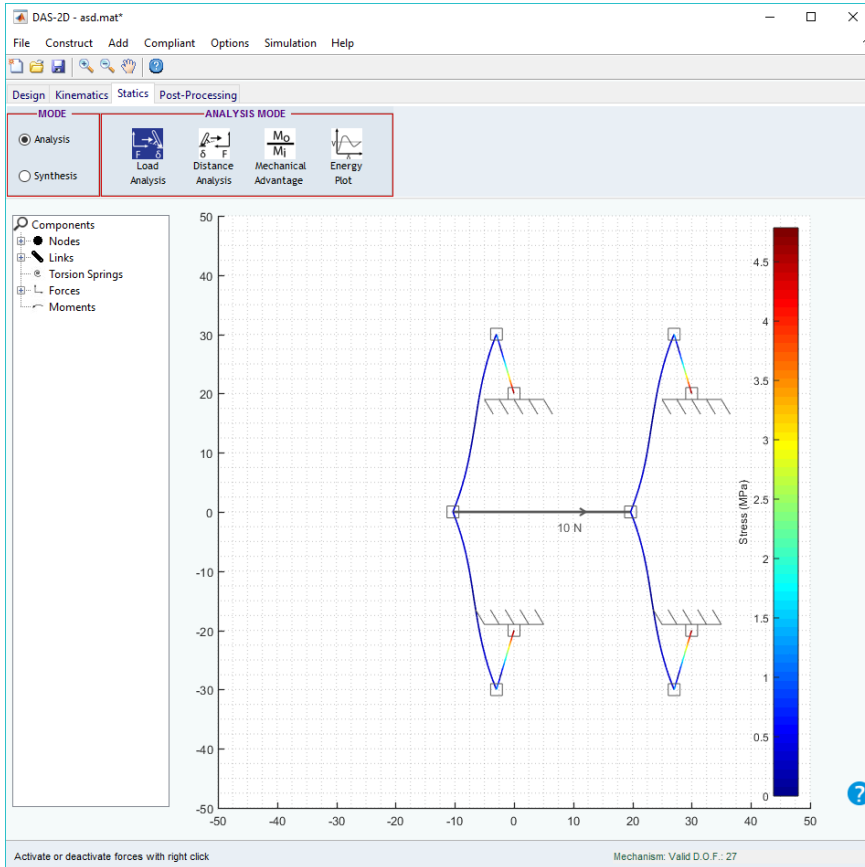


Step 6. Right Click to middle link and select add force. $F_x=15$ N and $F_y=0$ N. Location=50%. Hit Ok.

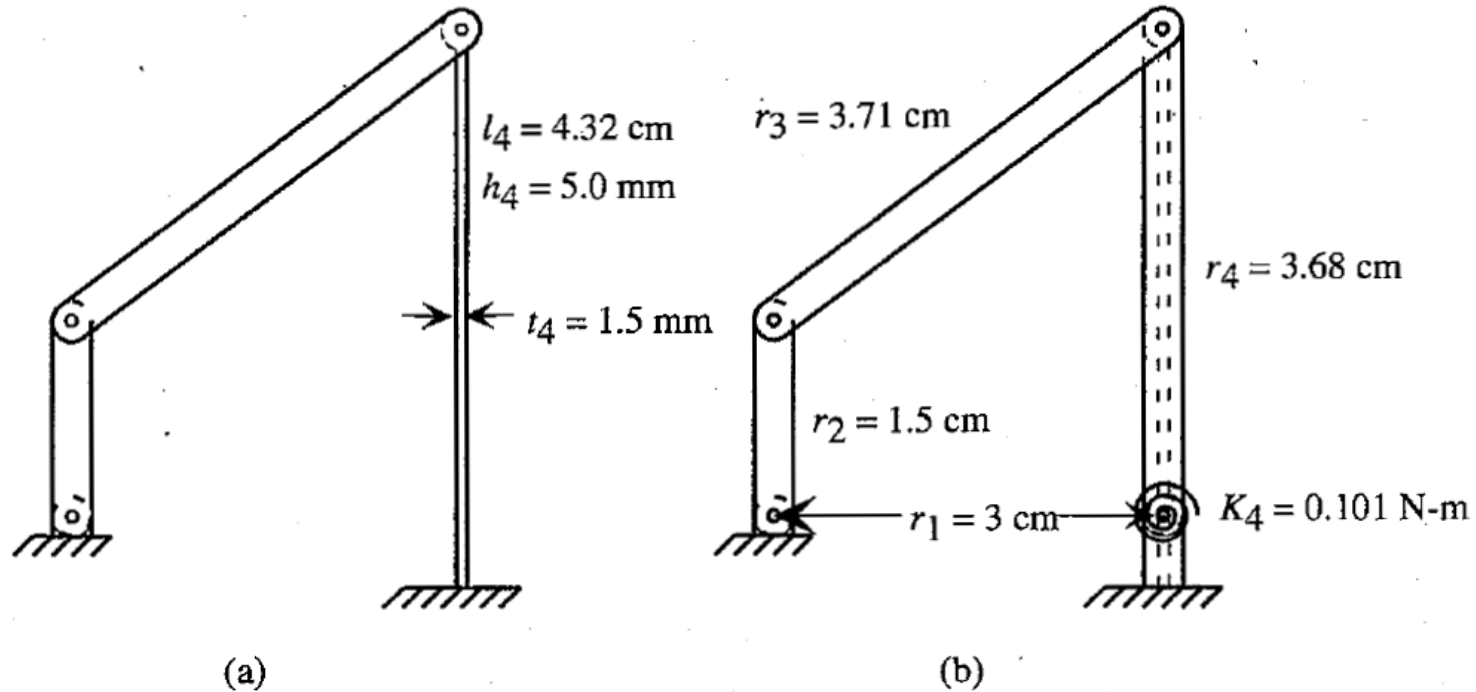
The screenshot shows the DAS-2D software interface. The main window displays a mechanism diagram on a coordinate grid. The diagram consists of 10 nodes and 10 links. A context menu is open over the middle link (Link 3-4), listing options: Edit Length & Angle, Edit Joints, Convert to Linear Spring, Make Compliant, Add Force, Add Moment, and Delete. The 'Add Force' option is highlighted. The status bar at the bottom indicates 'Mechanism: Valid D.O.F.: 27'.

The 'Force' dialog box is shown, allowing configuration of a force applied to a link. The 'Magnitude' tab is active. The force components are set to $F_x = 15$ N and $F_y = 0$ N. The force magnitude is set to 1 N and the angle is 0 degrees. The location is set to 50% from Node 3 to Node 4. Under 'Force Angle During Analysis', the 'Absolute' radio button is selected. The 'OK' button is highlighted.

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



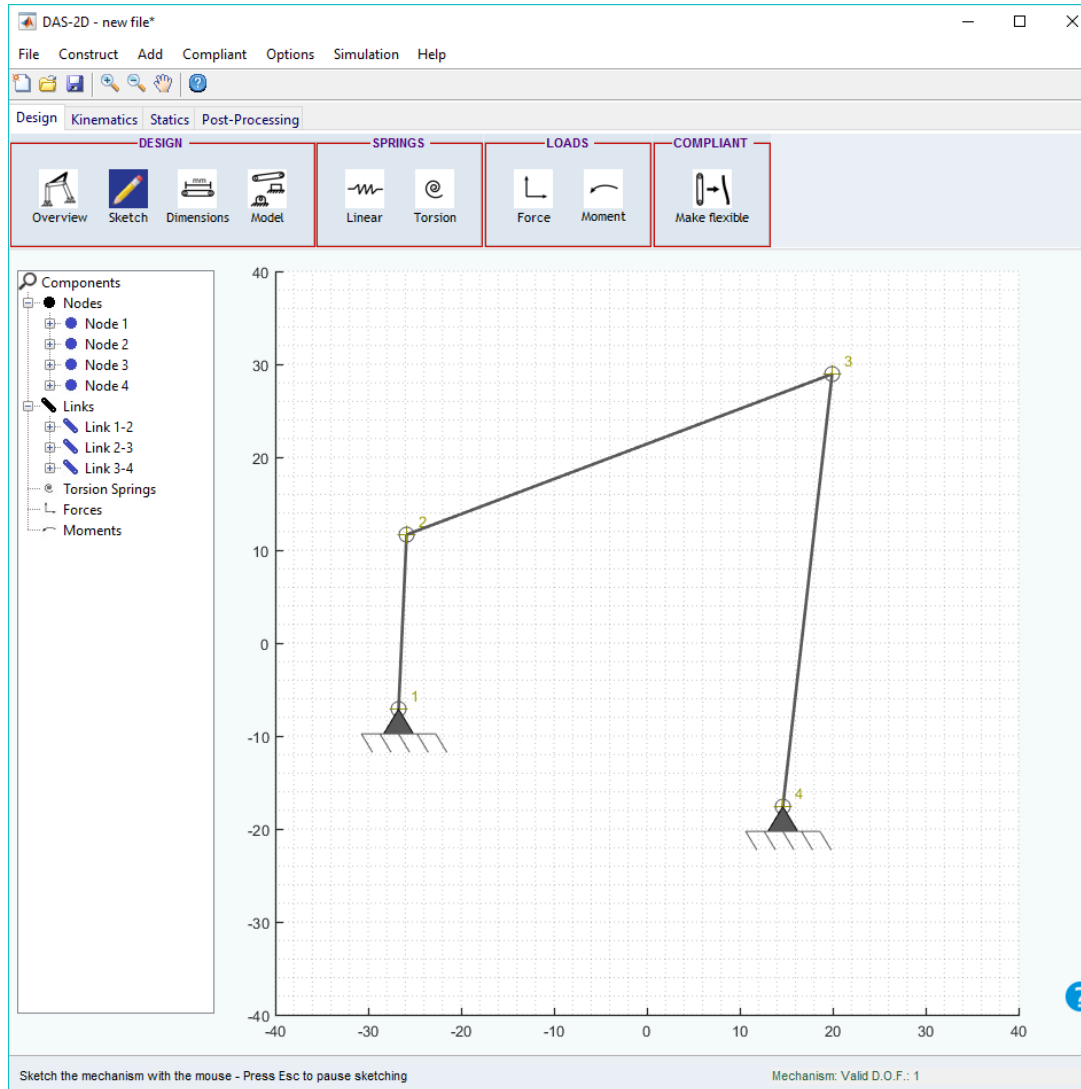
Start by setting workspace size 40 by 40 from Options-> Workspace



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

$E = 1.4 \text{ GPa}$,
In-plane thickness $t_4 = 1.5 \text{ mm}$
Out-of-plane thickness 5 mm

Step 1. Sketch the mechanism as shown below



Components

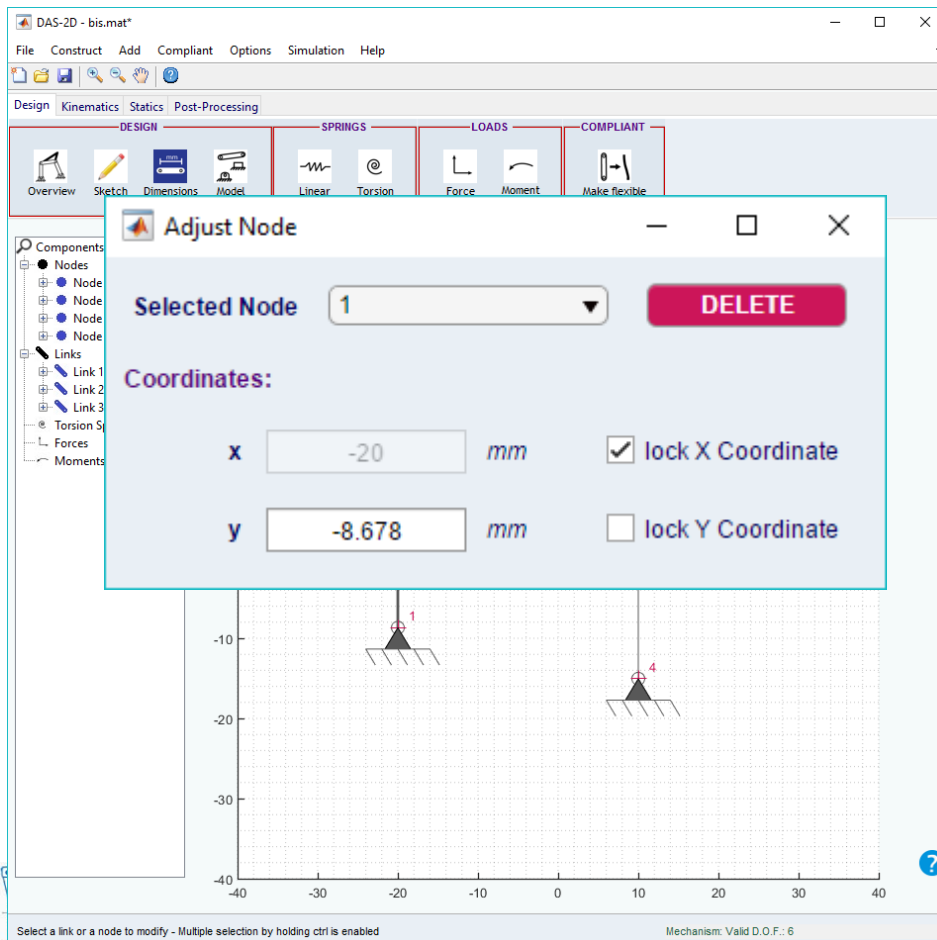
- Nodes
 - Node 1
 - Node 2
 - Node 3
 - Node 4
- Links
 - Link 1-2
 - Link 2-3
 - Link 3-4
- Torsion Springs
- Forces
- Moments

Sketch the mechanism with the mouse - Press Esc to pause sketching

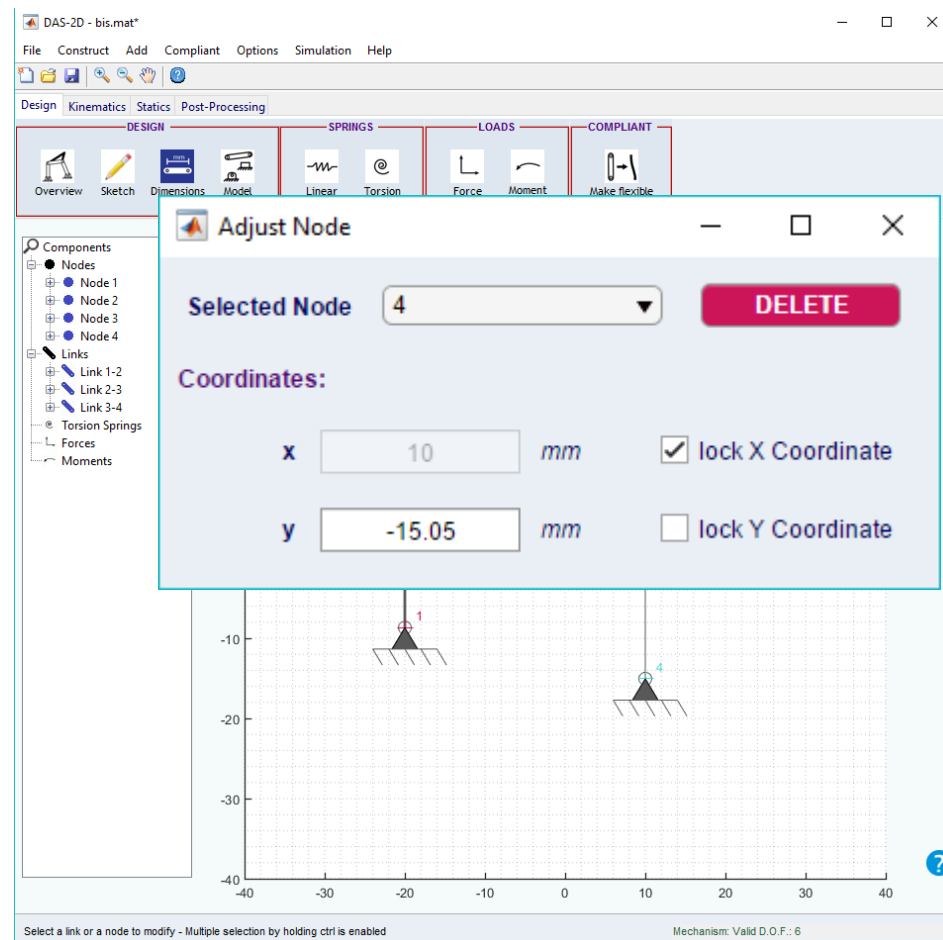
Mechanism: Valid D.O.F.: 1



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.

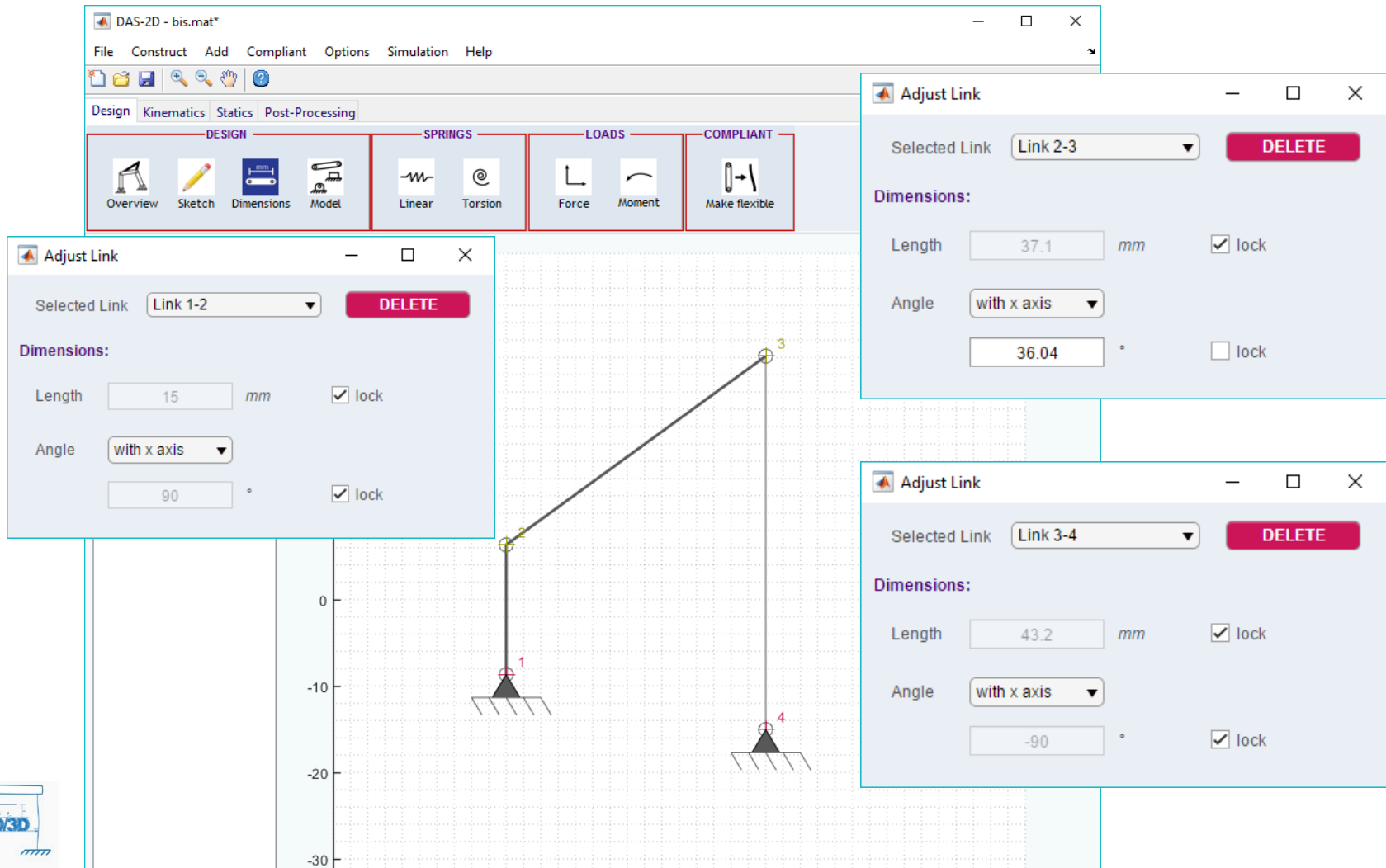


The screenshot shows the 'Adjust Node' dialog box for Node 1. The 'Selected Node' dropdown is set to 1. The 'Coordinates' section shows x = -20 mm and y = -8.678 mm. The 'lock X Coordinate' checkbox is checked, and the 'lock Y Coordinate' checkbox is unchecked. A 2D plot below the dialog shows the mechanism with Node 1 highlighted in red at the top pivot point.



The screenshot shows the 'Adjust Node' dialog box for Node 4. The 'Selected Node' dropdown is set to 4. The 'Coordinates' section shows x = 10 mm and y = -15.05 mm. The 'lock X Coordinate' checkbox is checked, and the 'lock Y Coordinate' checkbox is unchecked. A 2D plot below the dialog shows the mechanism with Node 4 highlighted in red at the bottom pivot point.

Step 3. Click to Link 1-2: $L=15\text{ mm}$ $\theta = 90^\circ(-90^\circ)$. Link 2-3: $L=37.1\text{ mm}$
 Link 3-4: $L=43.2\text{ mm}$ $\theta = 90^\circ(-90^\circ)$



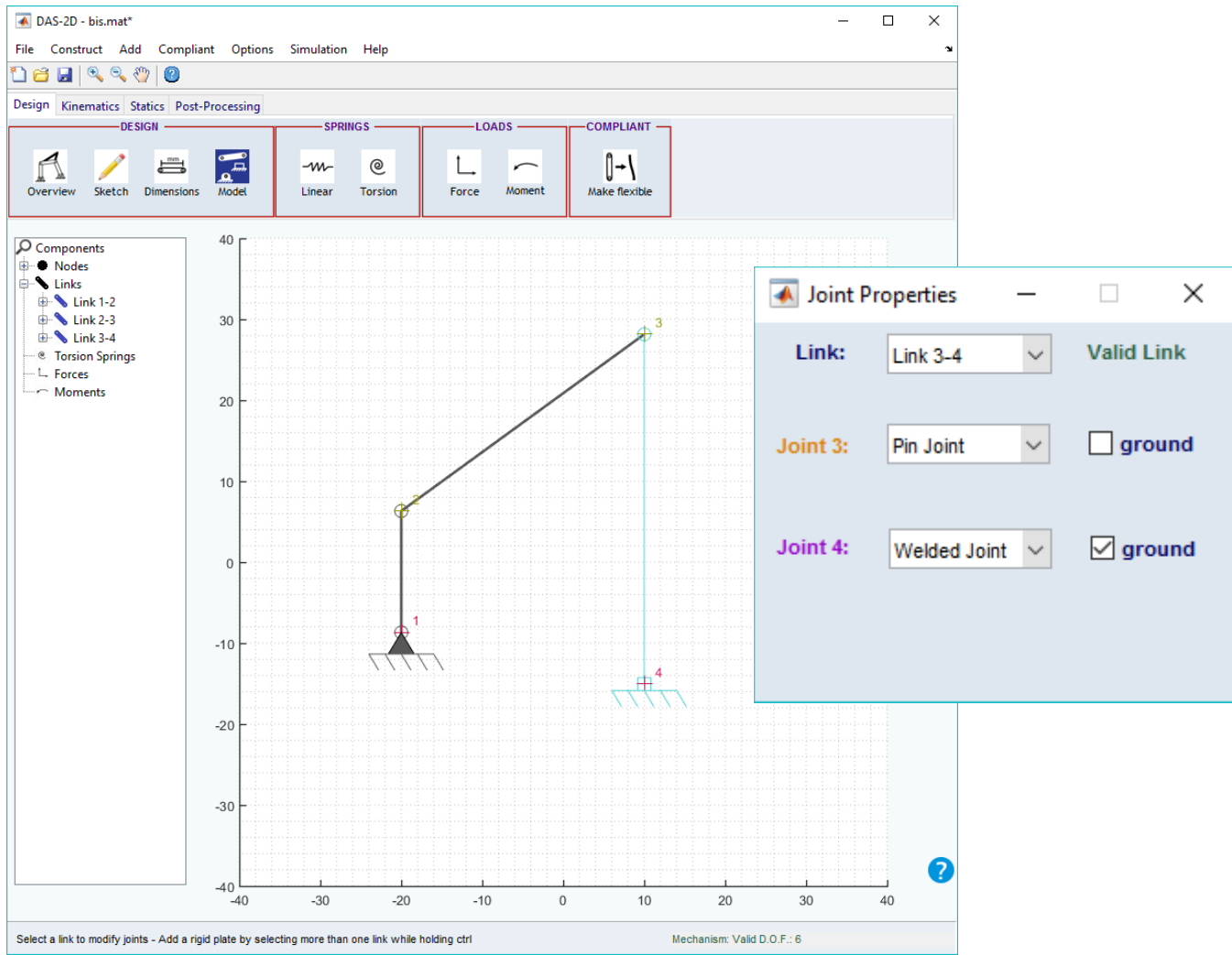
The screenshot displays the DAS-2D software interface with a four-bar mechanism model. The mechanism consists of four links connected at four joints (1, 2, 3, 4). Link 1-2 is a vertical link of length 15 mm, Link 2-3 is a diagonal link of length 37.1 mm, and Link 3-4 is a vertical link of length 43.2 mm. The joints are located at (1, -10), (2, -10), (3, 0), and (4, -20) on a coordinate system where the x-axis is horizontal and the y-axis is vertical.

Three 'Adjust Link' dialog boxes are shown, each corresponding to one of the links:

- Adjust Link 1-2:** Selected Link: Link 1-2. Dimensions: Length = 15 mm (locked), Angle = 90° (locked).
- Adjust Link 2-3:** Selected Link: Link 2-3. Dimensions: Length = 37.1 mm (locked), Angle = 36.04° (unlocked).
- Adjust Link 3-4:** Selected Link: Link 3-4. Dimensions: Length = 43.2 mm (locked), Angle = -90° (locked).

The software interface includes a menu bar (File, Construct, Add, Compliant, Options, Simulation, Help) and a toolbar with icons for Overview, Sketch, Dimensions, Model, Springs (Linear, Torsion), Loads (Force, Moment), and Compliant (Make flexible).

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



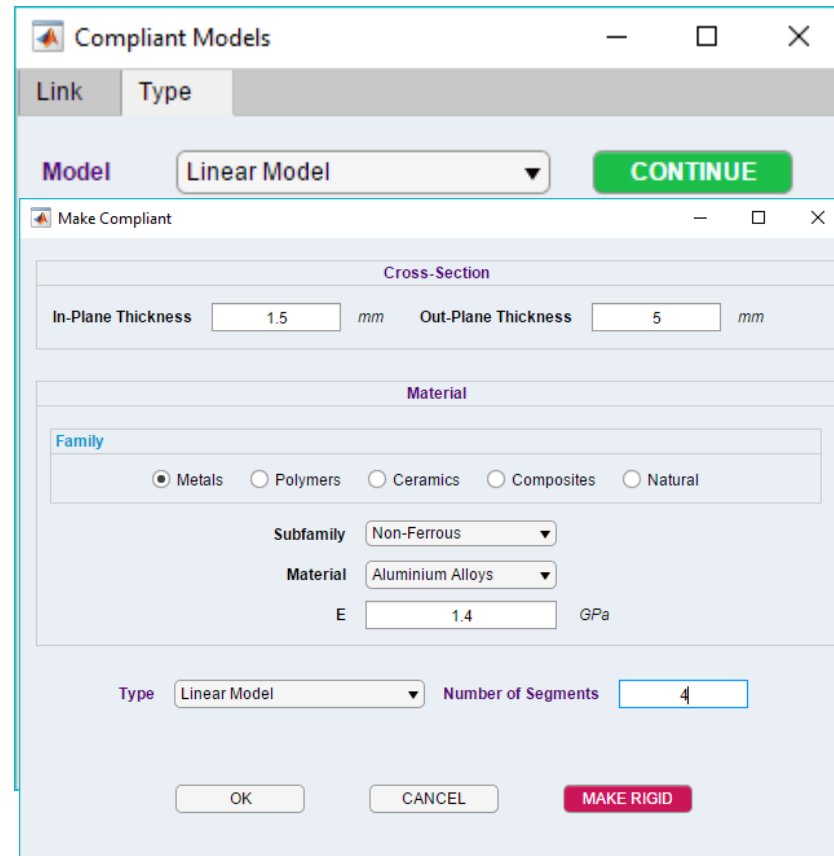
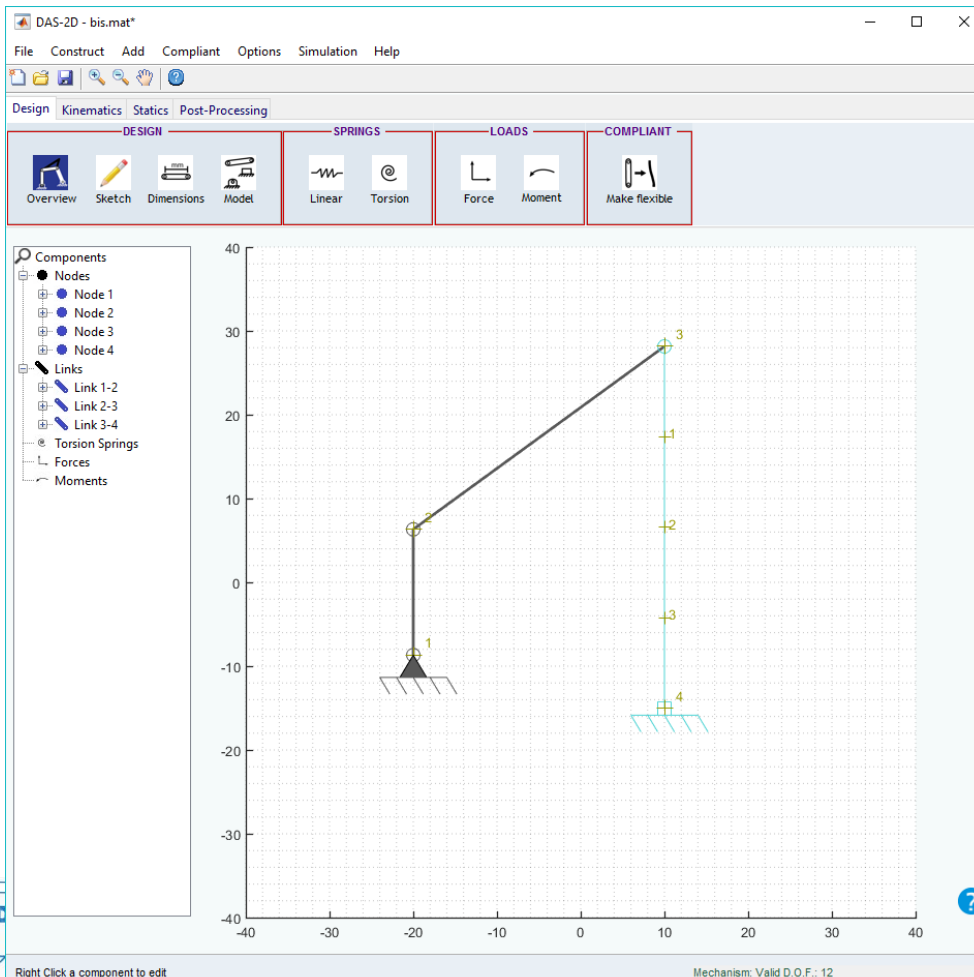
The screenshot displays the DAS-2D software interface. The main window shows a 2D plot of a four-bar mechanism with four links and four nodes. Node 1 is a revolute joint to ground, Node 2 is a revolute joint between Link 1-2 and Link 2-3, Node 3 is a revolute joint between Link 2-3 and Link 3-4, and Node 4 is a revolute joint to ground. The plot axes range from -40 to 40. A 'Joint Properties' dialog box is open, showing the following settings:

| Property | Value | Status |
|----------|--------------|--|
| Link: | Link 3-4 | Valid Link |
| Joint 3: | Pin Joint | <input type="checkbox"/> ground |
| Joint 4: | Welded Joint | <input checked="" type="checkbox"/> ground |

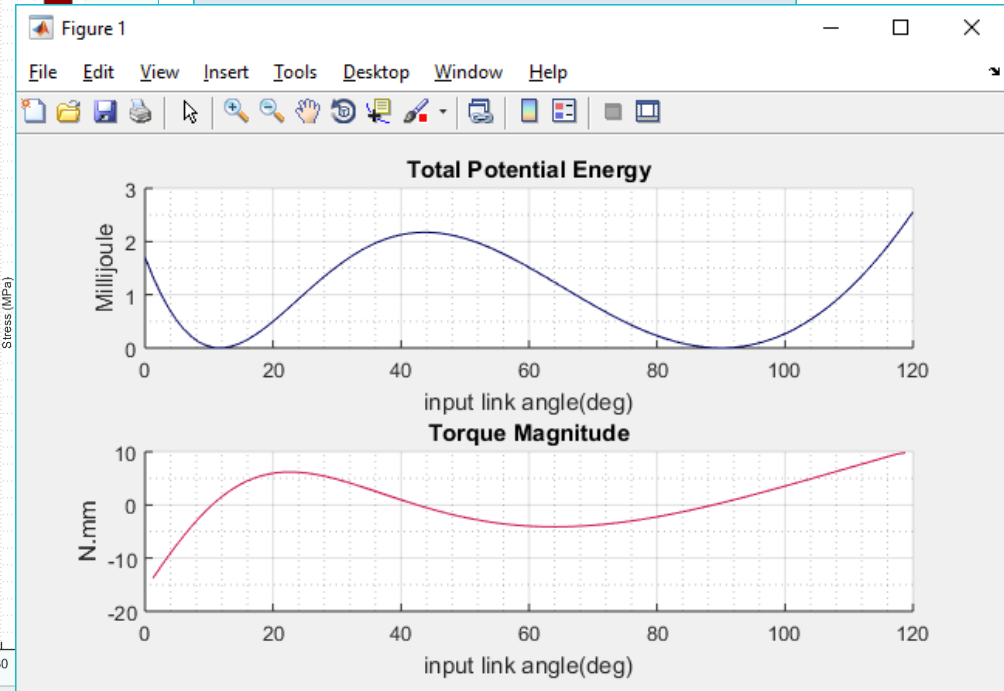
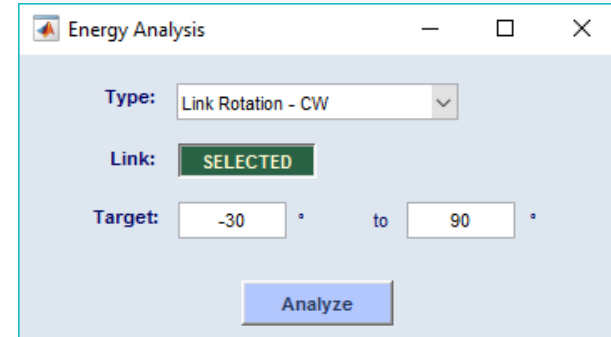
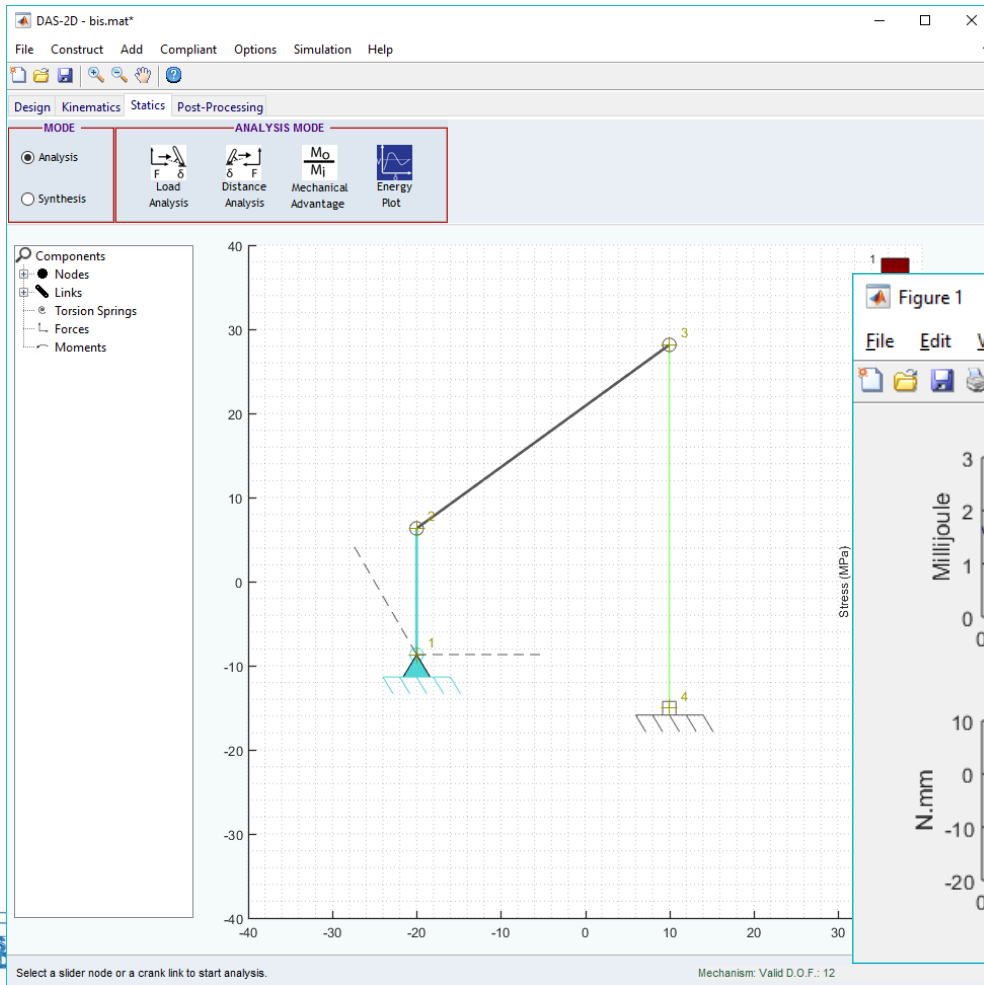
At the bottom of the software window, a status bar reads: "Select a link to modify joints - Add a rigid plate by selecting more than one link while holding ctrl" and "Mechanism: Valid D.O.F.: 6".



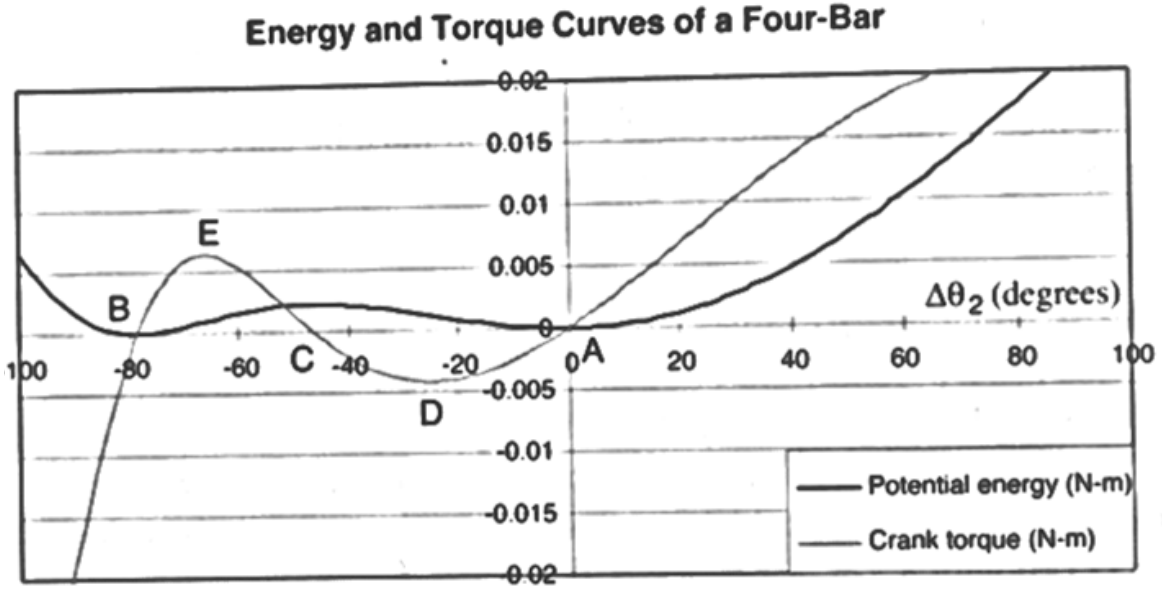
Step 5. Go to Overview Module. Right Click Link 3-4 and Make Compliant. Choose Linear Model. In Thickness = 1.5 mm Out of Thickness = 5 mm. E= 1.4 Gpa . Choose 4 Segments. Hit Ok.



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

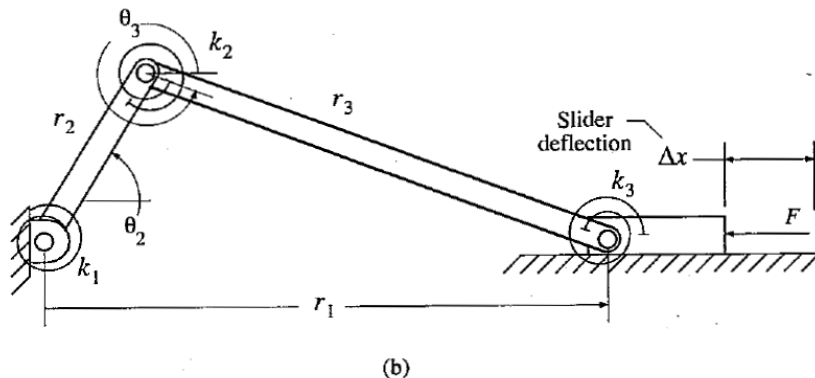
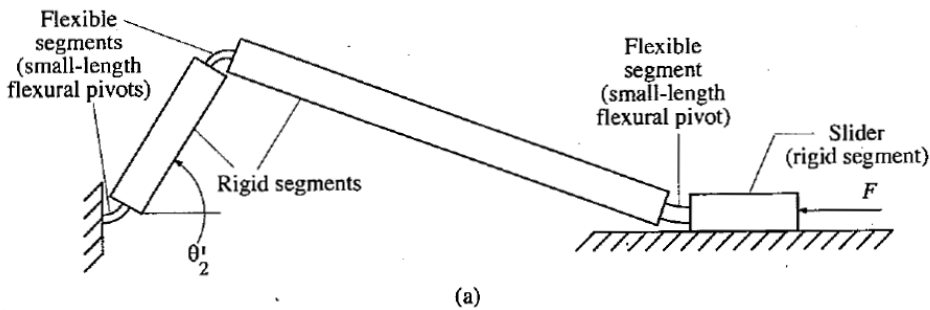


Compare your curve with curve below.

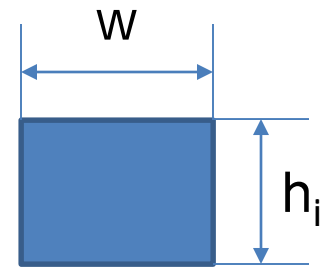


Compliant Mechanisms - Larry L. Howell
Page 364 - Figure 11.9

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



- $r_2=40.2\text{mm}$
- $r_3=107.1\text{mm}$
- $E=1665\text{ Mpa}$
- $h_2=h_1=0.76\text{mm}$
- $h_3=1.78\text{mm}$
- $I=5.08\text{mm}^4$



Rectangular cross-section

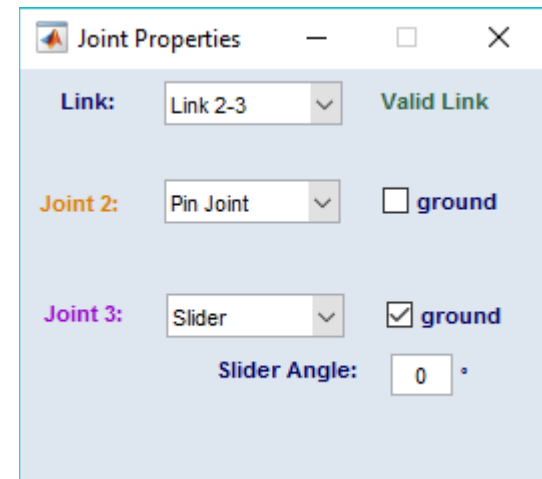
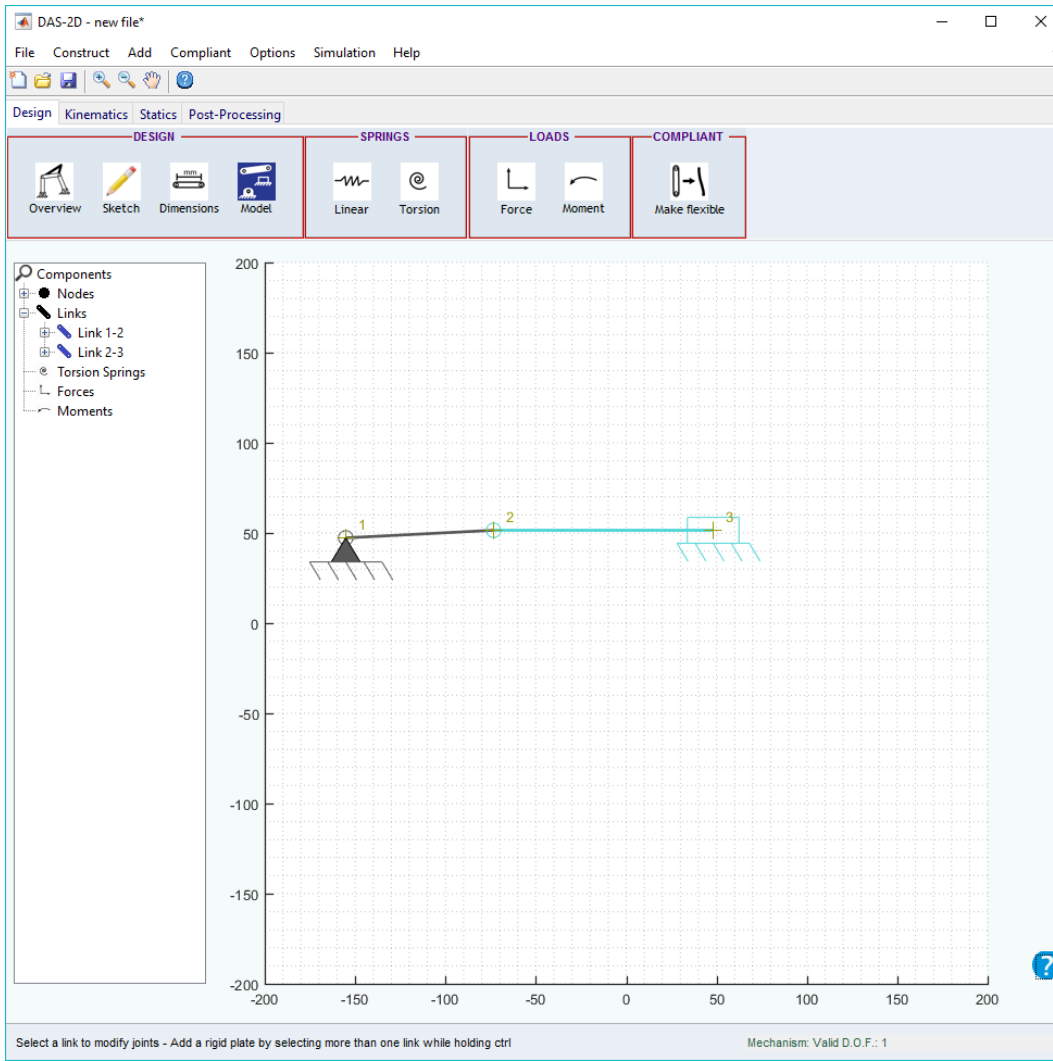
$$k_1 = \frac{EI}{l} = 151 \frac{\text{N}\cdot\text{mm}}{\text{rad}} \quad \left(1.35 \frac{\text{in}\cdot\text{lb}}{\text{rad}} \right)$$

Compliant Mechanisms - Larry L. Howell
Figure 10.1 Page 338

Step 1. Sketch the mechanism as shown below



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



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

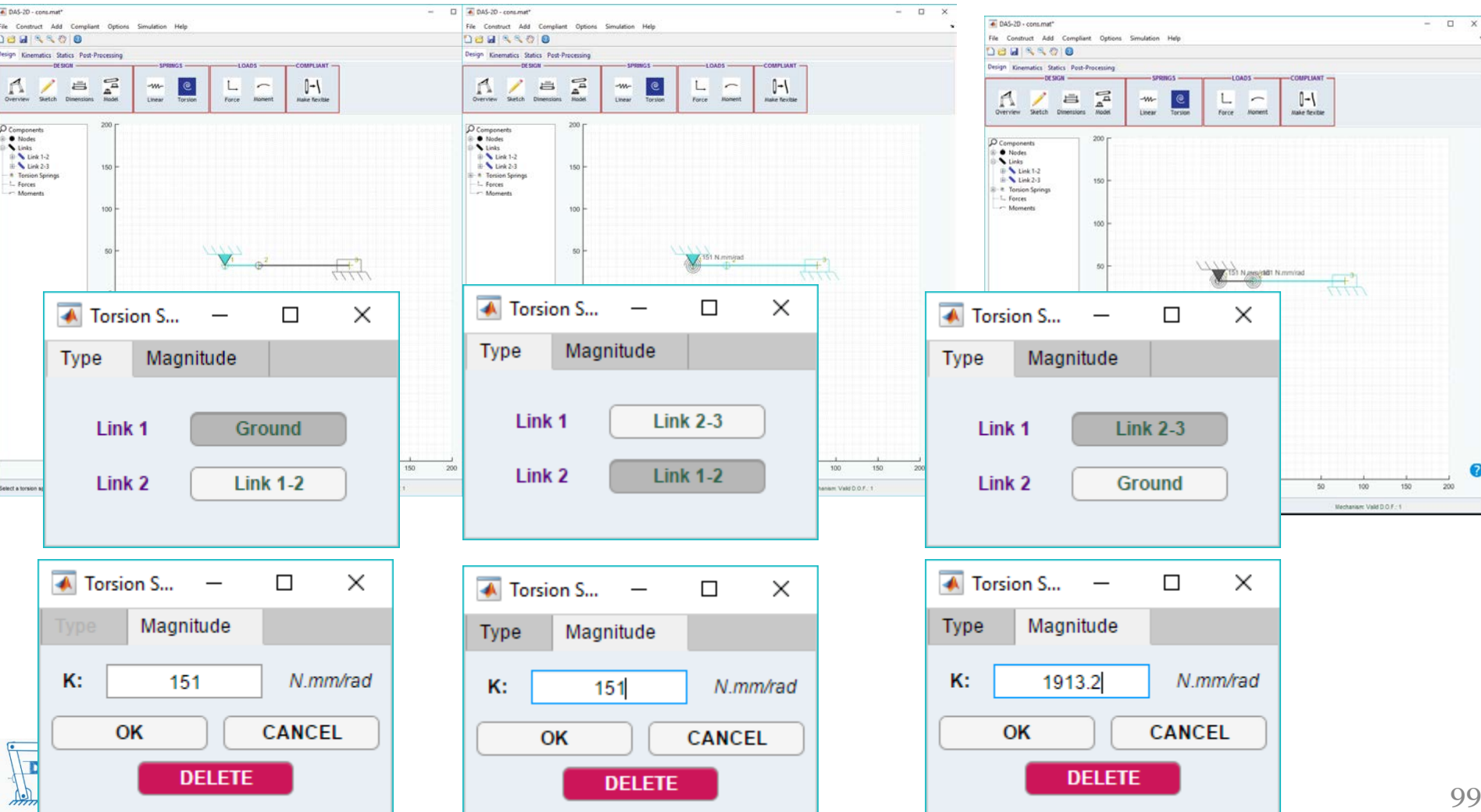
The screenshot shows the DAS-2D Beta software interface. The 'Adjust Link' dialog box is open for 'Link 1-2'. The dimensions are as follows:

| Parameter | Value | Unit | Lock |
|-----------|-------------|------|-------------------------------------|
| Length | 40.2 | mm | <input checked="" type="checkbox"/> |
| Angle | with x axis | | |
| | 6.076e-14 | ° | <input checked="" type="checkbox"/> |

The screenshot shows the DAS-2D Beta software interface. The 'Adjust Link' dialog box is open for 'Link 2-3'. The dimensions are as follows:

| Parameter | Value | Unit | Lock |
|-----------|-------------|------|-------------------------------------|
| Length | 107.1 | mm | <input checked="" type="checkbox"/> |
| Angle | with x axis | | |
| | -3.801e-14 | ° | <input checked="" type="checkbox"/> |

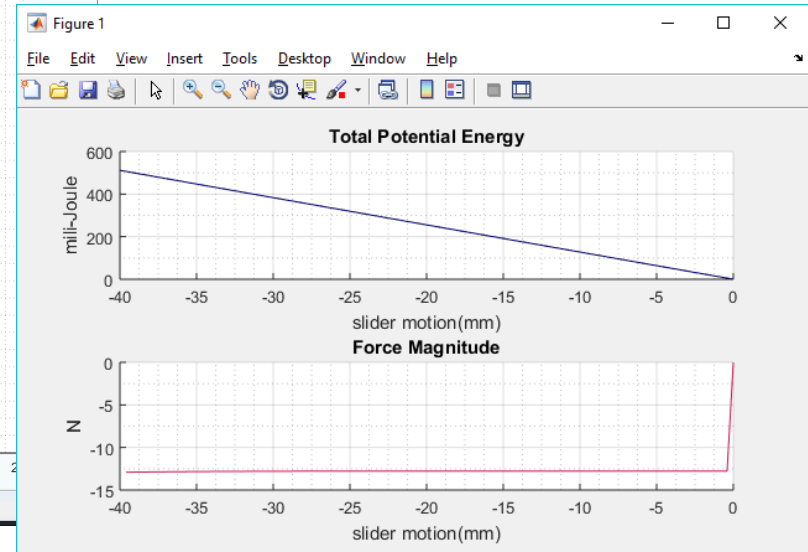
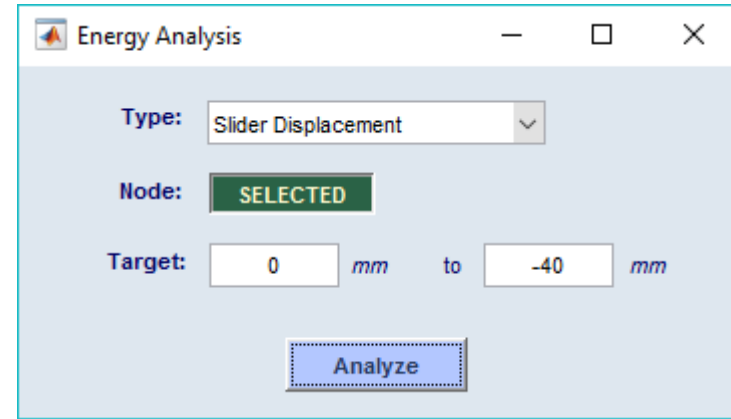
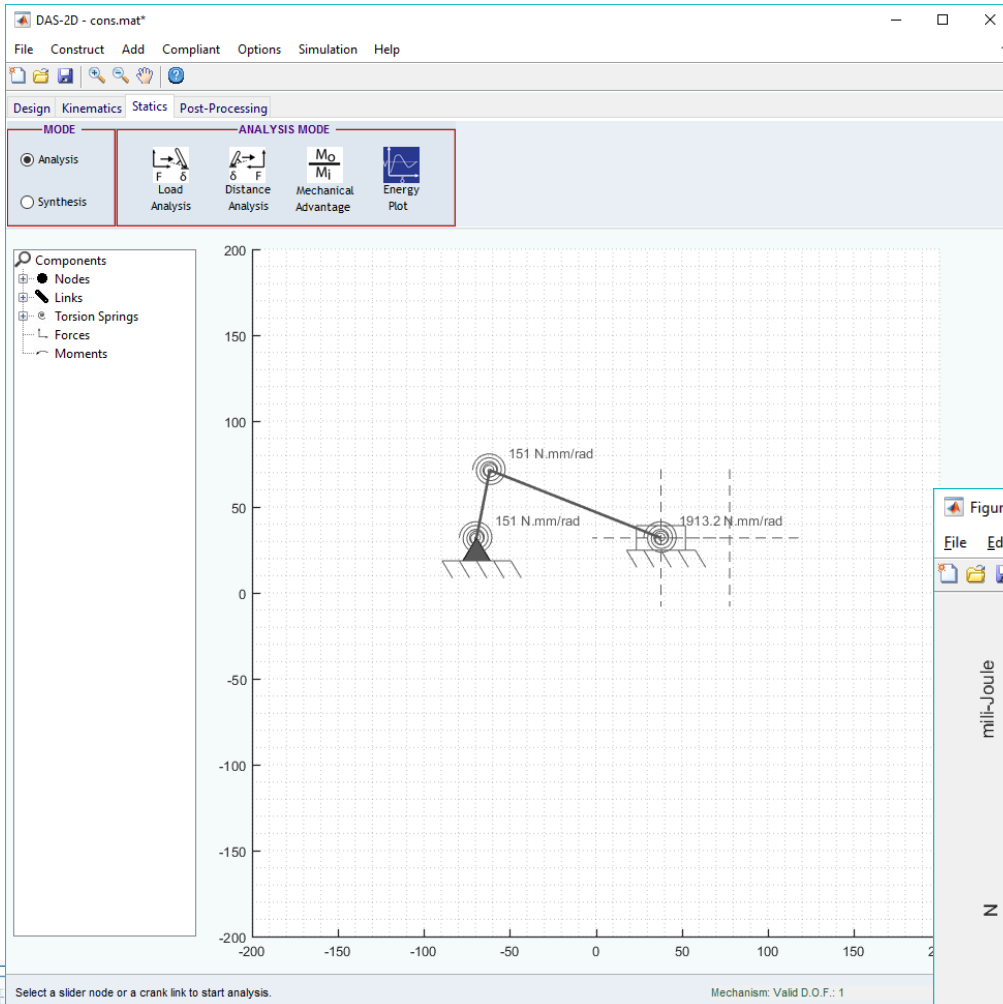
- Step 4. Go to Torsion Spring Module. Select Link 1-2 and Ground (Click empty space) Magnitude = 151
- Go to Torsion Spring Module. Select Link 1-2 and Link 2-3. Magnitude = 151.
- Go to Torsion Spring Module. Select Link 1-2 and Link 2-3. Magnitude = 1913.2.



The following table summarizes the configuration of the torsion springs shown in the screenshots:

| Screenshot | Link 1 | Link 2 | Magnitude (K) |
|------------|----------|----------|---------------|
| Top Left | Ground | Link 1-2 | 151 |
| Top Middle | Link 2-3 | Link 1-2 | 151 |
| Top Right | Link 2-3 | Ground | 1913.2 |

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



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 $h_3 = h_1 K_2^{1/3} = 1.78$ mm (0.070 in.).

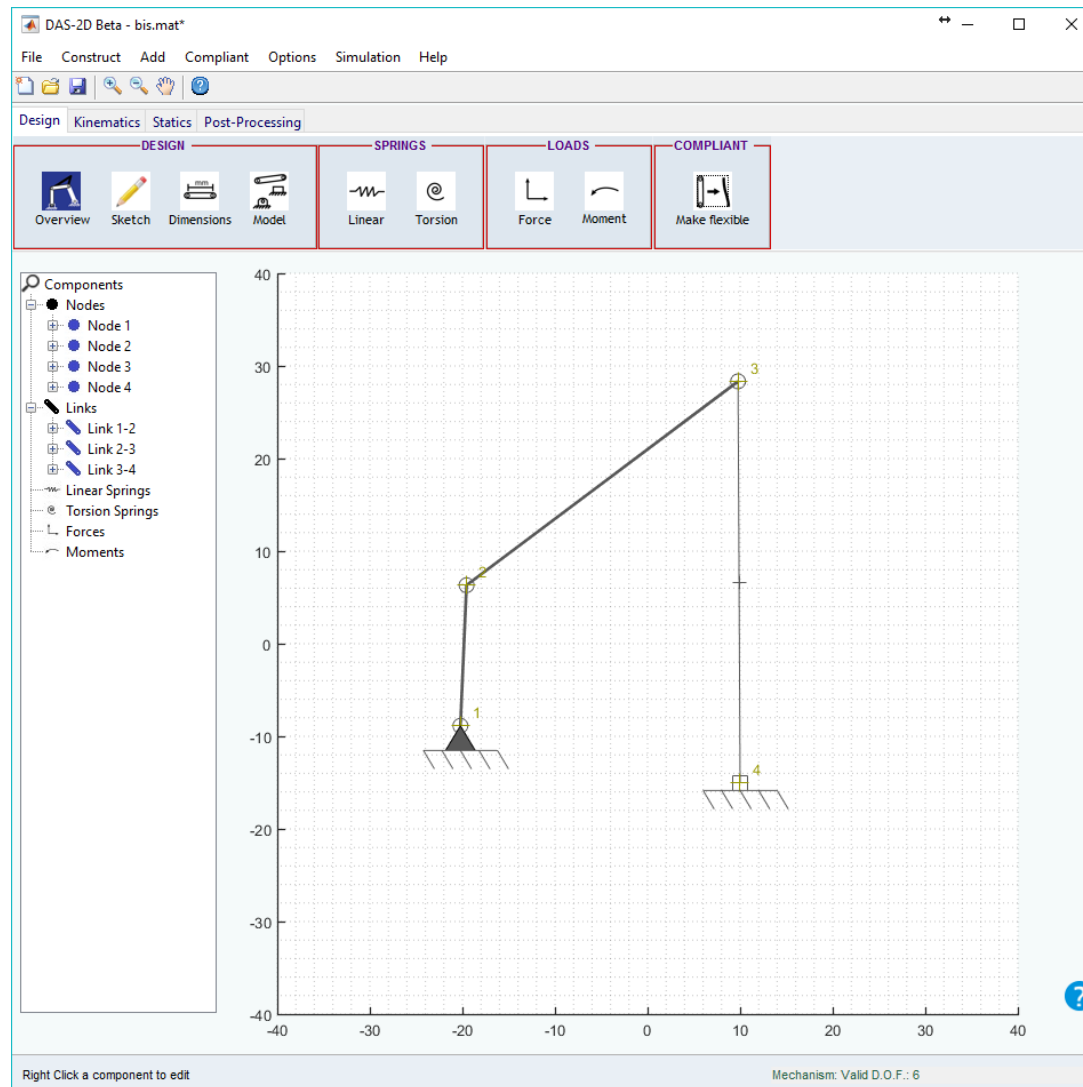
The nondimensionalized force term is $\Phi = 3.4016$ (Table 10.1). If the mechanism has a width $w = 12.7$ mm (0.50 in.), modulus of elasticity $E = 1655$ MPa (240,000 lb/in.²), and the flexural pivots each have a length of $l = 5.08$ mm (0.20 in.), then

$$k_1 = \frac{EI}{l} = 151 \frac{\text{N}\cdot\text{mm}}{\text{rad}} \quad \left(1.35 \frac{\text{in}\cdot\text{lb}}{\text{rad}} \right) \quad (10.19)$$

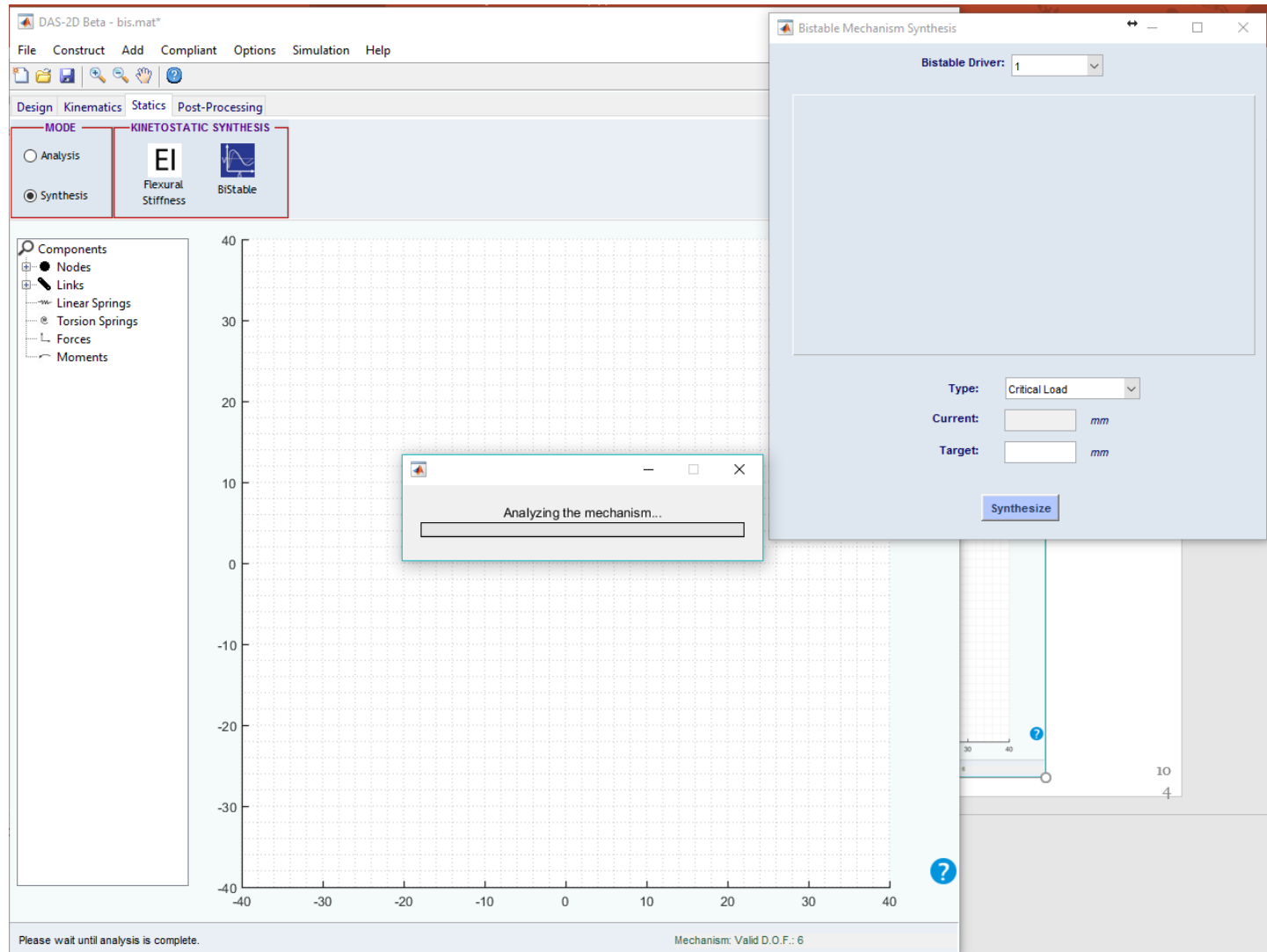
The resulting force is

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

Step 1. Create or load Ex.4

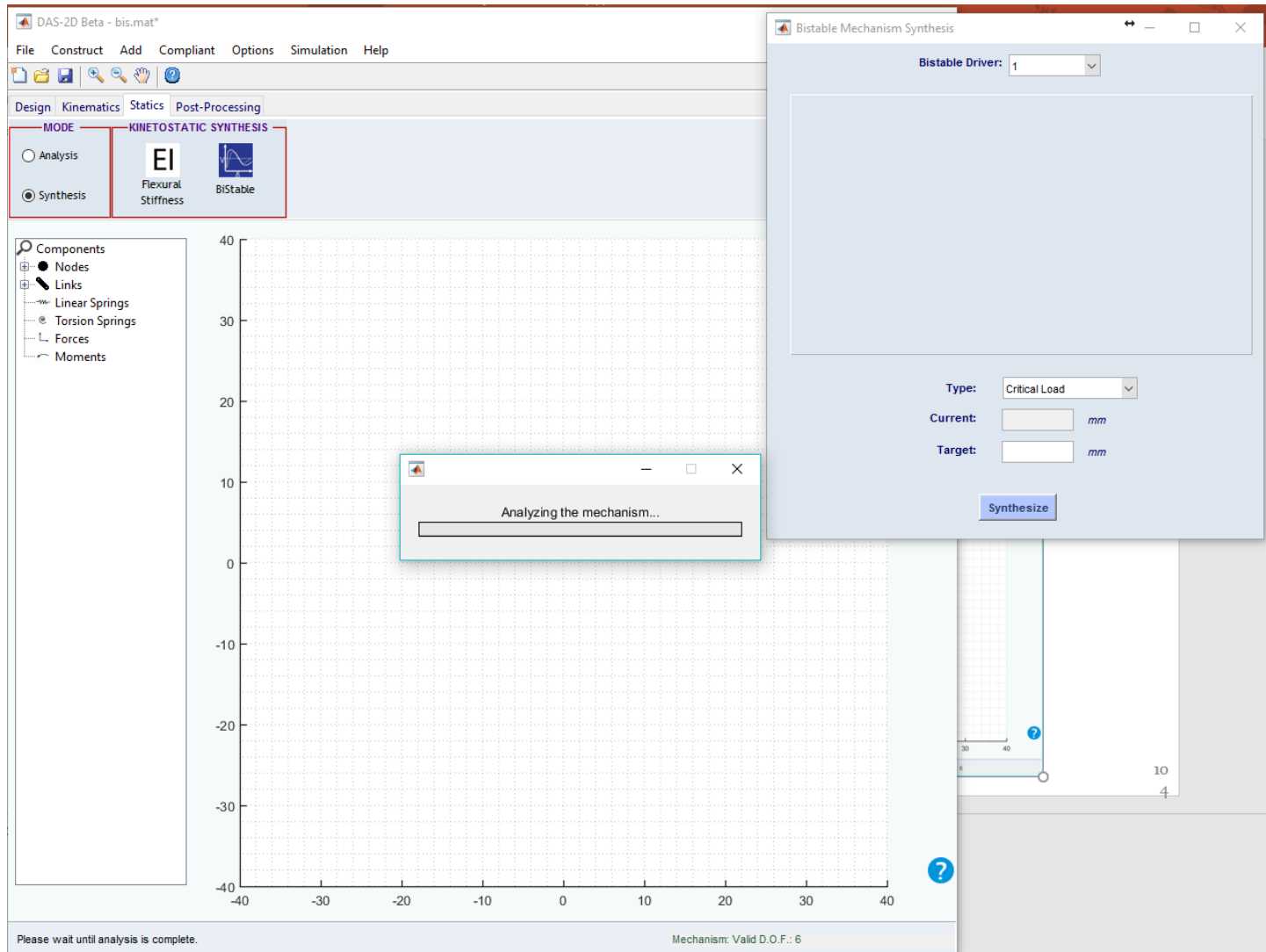


Step 2. Go to Statics-> Synthesis -> Bistable. The mechanism will be analyzed and two bistable drivers will be found (same driver CW and CCW)



The screenshot shows the DAS-2D Beta software interface. The main window is titled "DAS-2D Beta - bis.mat*" and has a menu bar with "File", "Construct", "Add", "Compliant", "Options", "Simulation", and "Help". Below the menu bar is a toolbar with various icons. The "Design" tab is active, and the "MODE" section is set to "Synthesis". The "KINETOSTATIC SYNTHESIS" section is also active, with "EI Flexural Stiffness" and "Bistable" options selected. A "Components" panel on the left lists "Nodes", "Links", "Linear Springs", "Torsion Springs", "Forces", and "Moments". The main workspace is a coordinate grid with axes ranging from -40 to 40. A "Bistable Mechanism Synthesis" dialog box is open on the right, with "Bistable Driver" set to 1, "Type" set to "Critical Load", and "Current" and "Target" fields set to 0 mm. A "Synthesize" button is visible. A smaller dialog box in the center says "Analyzing the mechanism...". The status bar at the bottom indicates "Mechanism: Valid D.O.F.: 6".

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



The screenshot displays the DAS-2D Beta software interface during the synthesis of a bistable four-bar mechanism. The main window is titled "DAS-2D Beta - bis.mat*" and features a menu bar (File, Construct, Add, Compliant, Options, Simulation, Help) and a toolbar. The "Design" tab is active, and the "MODE" section shows "Synthesis" selected. Under "KINETOSTATIC SYNTHESIS", the "Bistable" option is chosen. A "Bistable Mechanism Synthesis" dialog box is open on the right, with "Bistable Driver: 1" selected in a dropdown menu. The "Type" is set to "Critical Load", and the "Current" and "Target" fields are empty, both with units of "mm". A "Synthesize" button is visible at the bottom of the dialog. A small "Analyzing the mechanism..." dialog box is also present in the center. The background shows a coordinate grid with axes ranging from -40 to 40. A partial mechanism diagram is visible in the bottom right corner. The status bar at the bottom indicates "Please wait until analysis is complete." and "Mechanism: Valid D.O.F.: 6".

DAS-2D Beta - bis.mat*

File Construct Add Compliant Options Simulation Help

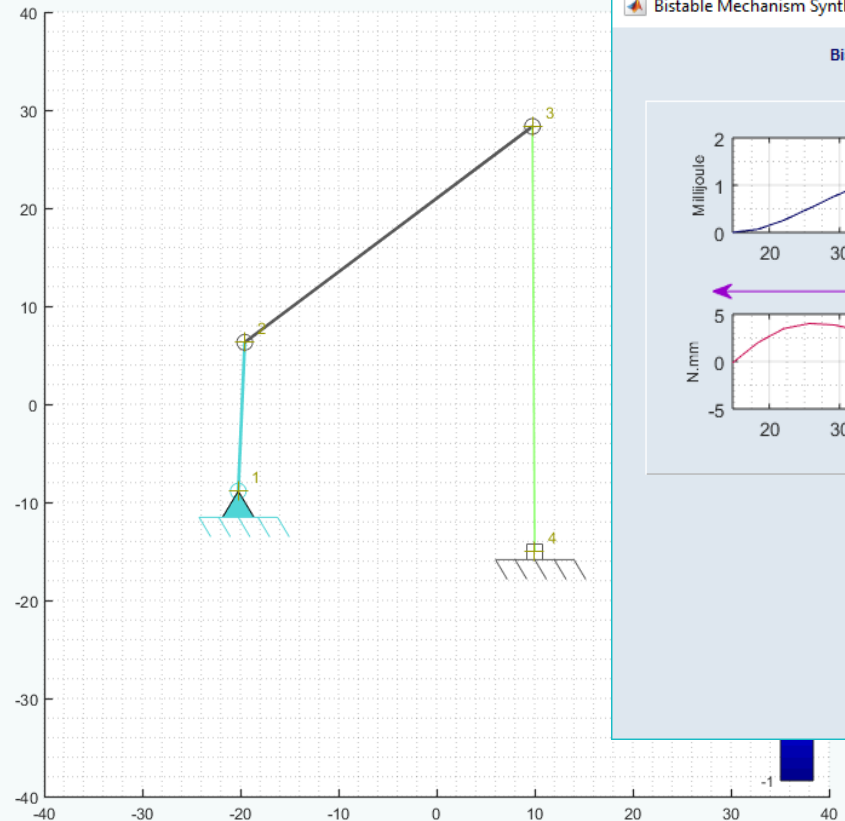
Design Kinematics Statics Post-Processing

MODE KINETOSTATIC SYNTHESIS

Analysis EI Flexural Stiffness BiStable

Synthesis

- Components
 - Nodes
 - Links
 - Linear Springs
 - Torsion Springs
 - Forces
 - Moments

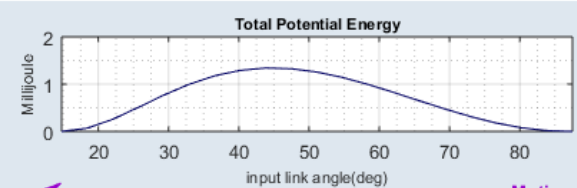


The mechanism has 2 drivers that will result in bistable position.

Mechanism: Valid D.O.F.: 6

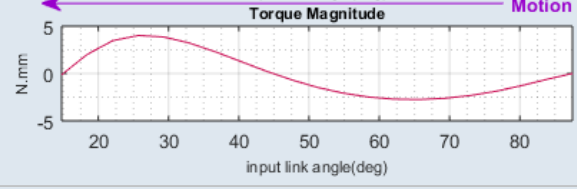
Bistable Mechanism Synthesis

Bistable Driver: 1



Total Potential Energy

Motion



Torque Magnitude

Type: Critical Load

Current: Critical Load

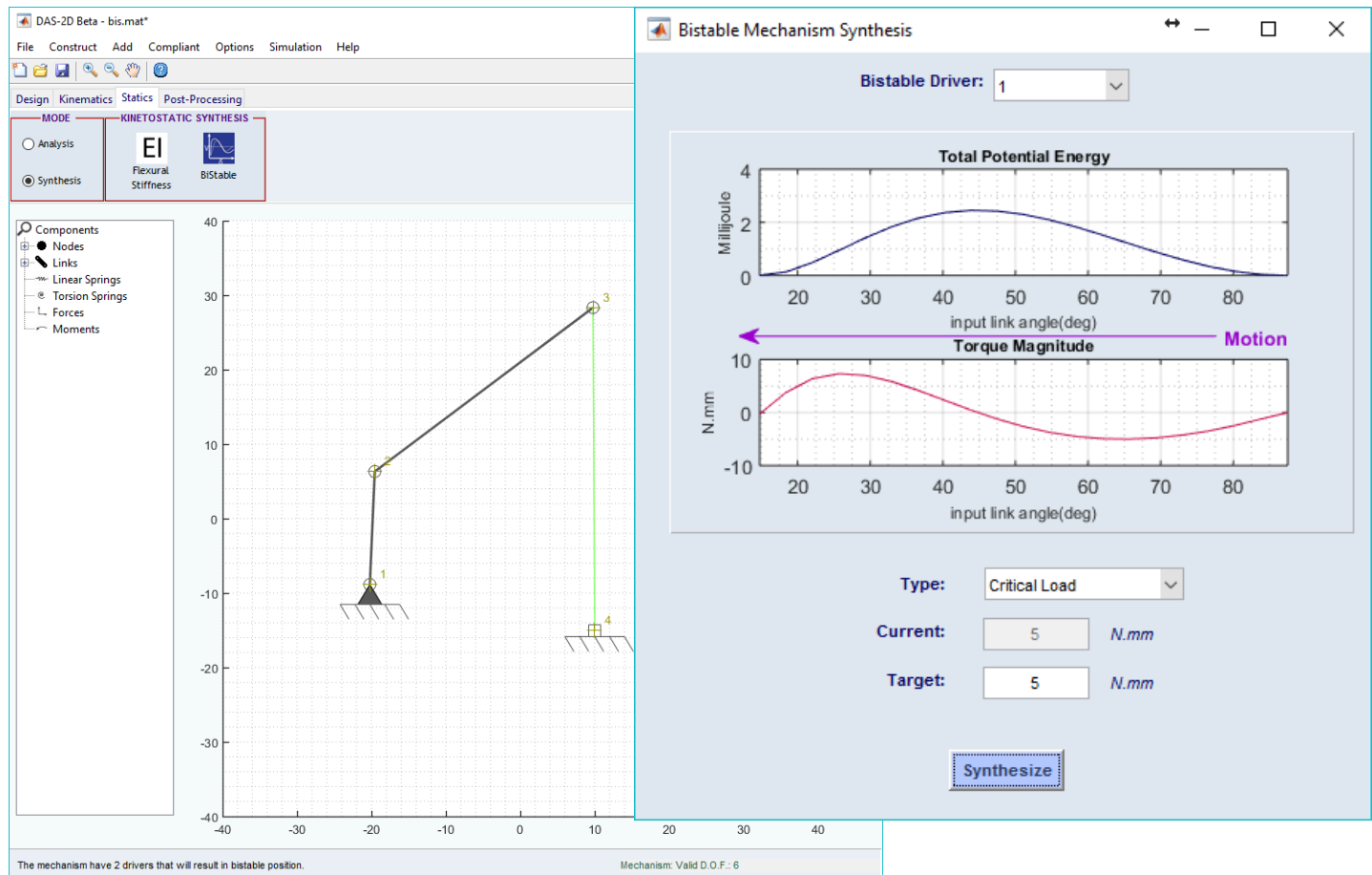
Target: Unstable Position

Stable Position

27.333 14.111

Synthesize

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.



The screenshot displays the DAS-2D Beta software interface for Bistable Mechanism Synthesis. The main window shows a 2D plot of a four-bar mechanism with nodes 1, 2, 3, and 4. Node 1 is a revolute joint, node 2 is a revolute joint, node 3 is a revolute joint, and node 4 is a prismatic joint. The plot shows the mechanism in a bistable position.

The right-hand panel, titled "Bistable Mechanism Synthesis", displays the following information:

- Bistable Driver:** 1
- Total Potential Energy:** A graph showing energy (Millijoules) versus input link angle (deg). The energy curve is a smooth, symmetric curve peaking at approximately 2.5 Millijoules at 45 degrees.
- Torque Magnitude:** A graph showing torque (N.mm) versus input link angle (deg). The torque curve is a smooth, symmetric curve peaking at approximately 8 N.mm at 25 degrees and reaching a minimum of approximately -5 N.mm at 65 degrees. A purple arrow labeled "Motion" points from right to left, indicating the direction of motion.
- Type:** Critical Load
- Current:** 5 N.mm
- Target:** 5 N.mm
- Synthesize** button

At the bottom of the software window, a status bar indicates: "The mechanism have 2 drivers that will result in bistable position." and "Mechanism: Valid D.O.F.: 6".