Using Mastan - Trusses

Annette Tardiff, 2003
Sanjay Arwade, 2008
Introduction

Mastan is an engineering analysis tool used to analyze how different structures will react under specific loading conditions.

Because Mastan uses the Matlab engine to do its calculations, it can do a wide range of analyses that are cumbersome or impossible on paper.
Outline

- Structure Definition
- Example problem: Truss
- Miscellaneous options
Before you begin, you must know:

- **Geometry:** nodes and elements
- **Fixities/boundary conditions** (physical restraints on the structure)
- **Sectional Properties**
  - $A$: cross-sectional area of each beam
- **Material Properties**
  - $E$: modulus of elasticity
Definition: Nodes and Elements

Elements (also called members) are the actual truss elements composing your structure.

Nodes are the joints between elements. Nodes must be placed first, and then connected by elements in Mastan.
Definition: Connections

The kind of joints you are using is crucial to your structure.

Use pins as joints for a truss. The character of the joints is specified at analysis time, after building the model.
Definition: Fixities

These are the boundary or support conditions of the structure, for example, pins and rollers
Definition: Section Properties
(Properties dependent upon shape rather than material)

- A: cross-sectional area of beam
- For a truss, A is the only section property required. All the others may be left blank
Material Properties:
(Properties dependent on material)

- **E**: Young’s modulus, or modulus of elasticity: $E$ is a measure of how much you have to pull on something to make it stretch a certain amount.

- For our analyses only the Young’s modulus is required.
Working with Mastan

In order to analyze a structure, you must input it into the program. The relevant components are:

- **Nodes**
- **Elements**
- **Connections**
- **Sections** (Define and Attach)
- **Materials** (Define and Attach)
- **Fixities**
- **Loads** (Only point forces for trusses)
Example Problem: Truss

Objectives:
- Deflected shape diagram
- Axial Force diagram
- Node displacements
- Reactions
- Element results
- Units: KN mm Gpa
Example Problem: Truss (cont’d)

This truss, is made of two kinds of elements.
All vertical members are wide flange W203X60 beams (A=7550mm$^2$) made of Structural Steel (E=200GPa).
All other members are wide flange W203X36 beams (A=4570mm$^2$) made of annealed stainless steel (E=190GPa)
Adding Nodes (rectangular frame)

Go to Geometry/Define Frame and enter the number and width of the bays (4 bays @ 4000mm), the number and height of the stories (1 story at 4000mm), and (for 3D frames) the number and depth of the frames (1) in each box respectively. Hit “Apply.”
Mastan will work with any consistent set of units. We are using kN-mm-MPa.
Adding Nodes (manual layouts)

If you are building anything other than a rectangular frame, you will need to enter each node manually. Go to Geometry/Define Node, and enter the \((x,y,z)\) coordinates of each node in the boxes at the bottom of the screen, pressing “Apply” after each one.
Coordinates shown for second node added (node 12)
Adding Elements

Go to Geometry/Define Element. For each element, click on each of its end nodes to select them, and press “Apply.”
Add enough elements to connect all nodes as shown, including this last one.
Subdividing Elements

To add the smaller triangles we can subdivide the existing elements to create new nodes at their midpoints. This option is found under the Geometry menu.

As an added note, the Geometry menu gives the options to move or duplicate nodes and reorient elements, should the need arise.
After hitting “Apply” for one set, selecting the next.
Fill in with elements, as shown
Notes on Material and Sectional Property Sets

- Note that elements do not become deselected after you attach the property set to them – you must deselect them yourself.
- If you attach a second section or material property set to an element, it will replace the first.
- Properties/“Remove Section” or “Remove Material” deletes the section or material definition, rather than giving you the option to unattach it from one or more elements.
- As you attach a section or material property set to an element, that element goes from being represented by a dotted line to a dashed one, and then to a solid line once both property sets are attached.
Defining Sectional Properties

Go to Properties/Define Section, and enter your value of A in the boxes at the bottom of the screen, then click “Apply.”

If your structure will be made up of parts with different cross sections, continue to define sections for each one (2 in this case).
Defined the first section, hit “Apply” then define the second section.
Applying Sectional Properties to Individual Elements

- Go to Properties/Attach Section
- Select all of the elements that have the first cross section
- Make sure the proper section is selected in the boxes at the bottom of the screen
- Hit “Apply”
- Continue in this manner until all of your elements have sectional property sets attached
Attach Section 1 to vertical members after attaching Section 2 to all others
Defining Material Properties

Go to Properties /Define Material, and enter your value of E in the box at the bottom of the screen.

The Poisson’s ratio $\nu$ and the yield stress $F_y$ need not be entered.

Hit “Apply” and continue in this manner until you have defined all of the materials that you will be using (in this case 2).
Hit “Apply” to define Material 2, after defining Material 1.
Applying Material Properties to Individual Elements

- Go to Properties/Attach Material
- Select all of the elements that will be made from that material
- Make sure the proper section is selected in the boxes at the bottom of the screen
- Hit “Apply.”
- Continue in this manner until all of your elements have material property sets attached.
Attach Material 2 to vertical members after attaching Material 1 to all others.
Defining Conditions

*Next, define the conditions under which the structure will be placed, including:*

- **Fixities/boundary conditions/support conditions**
- **Point Loads**
Defining Fixities

- Go to Conditions/Fixities.
- Select one of the nodes that will be attached to the ground.
- Check the boxes that correspond to the degrees of freedom that will be restrained.

For example, if your structure is simply supported (meaning one end is held in place with a pin joint, the other using a roller along the x-axis), you will check x and y for the first joint, but only y motion for the second joint.

- Hit “Apply.”
The structure is simply supported (pin on left, roller on right)
Applying Point Loads and Moments

- Under the Conditions menu, select Define Forces
- Select the node to which you will apply the point force
- Enter the x, and y components of the force into the boxes at the bottom of the screen.
- Hit “Apply.”
Applying several point loads
Analysis

Mastan is capable of many different types of analyses.

The data resultant from these analyses can be displayed as:
- Diagrams, or
- Numeric results
Definition: Elastic and Inelastic Behavior

Elastic behavior: When a loaded object is stretched and then the load is removed, the object returns to its original size and shape, much like an elastic band would.

Inelastic behavior: When the load permanently deforms the object, even after it is removed.

We will do elastic analysis
1\textsuperscript{st} and 2\textsuperscript{nd} Order Elastic Analyses

1\textsuperscript{st} order analysis assumes that the displacements are small compared to the size of the structure.

2\textsuperscript{nd} order analysis allows for large displacements.

We will do 1\textsuperscript{st} order analysis.
Analysis Execution

- Under Analysis, choose 1\textsuperscript{st} order elastic
- Choose in the selection box at the bottom of the screen planar truss (x-y)
- Hit Apply
- You should get a message Success Analysis Complete
1st Order Elastic Analysis of Planar Truss (x-y)
Diagrams

Diagrams can be made by selecting the desired diagram from under the Results/Diagrams menu from:

- Deflected shape
- Axial force
Deflected shape, 1st Order Elastic Truss analysis
Axial Force, 1st Order Elastic Truss analysis
Numeric Results

Pure numeric results can be found by selecting the node or element for which the results are needed, after choosing from the Results menu:

- Node Displacements
- Node Reactions
- Element Results
Viewing displacements at top center node
Viewing reactions at far left node
Miscellaneous Options

*Other options can be found under:*

- The **View** menu
- **Geometry/Information/**
  (Node or Element)
View Options

- Pan/Zoom
- Rotate
- Zoom Box
- Center
- Fit
- Defined Views
- Labels
- Display Settings
View/"Pan/Zoom"

This option moves the display window or adjusts the size of the object on the display using a row of well labeled buttons at the bottom of the screen.
Rotate

This option rotates the angle of the display window in three dimensions using another row of well-labeled buttons at the bottom of the screen.
Zoom Box, Center, and Fit

- Zoom Box – select a box with the curser which will become the center of the new view
- Center – select the center point of the new view
- Fit – automatically re-zoom and center to fit the whole structure (with diagrams if applicable) on the screen
Defined Views

The predefined views are quite handy when dealing with 3-dimensional structures. The available ones are:

- Front (x-y)
- Side (y-z)
- Top (x-z)
- Isometric (x-y-z)
Labels

In this menu, you may turn on or off the labeling of all of the various parts of structures and diagrams.

This is useful when you would like to know more information, or when your display is getting cluttered.
Display Settings

Using the same menu system as before (the 2-3 bottom lines of the screen), you can set the defaults for:

- Font size, for menus or figure labels
- Pan amount
- Zoom factor
- Rotate amount
- Symbol
Geometry/Information

You can view the position, orientation, and loading condition of any node or element by selecting it after selecting the appropriate option from under the Geometry/Information menu.
Viewing info for selected element
Good Luck!

I hope this tutorial has helped you learn to use Mastan.