# TRUSS ANALYSIS AND VISUALIZATION

Programmed by
Hussam Dandashli
POBox 3035
W. Lafayette, IN 47906
e-mail: dandashl@mn.ecn.purdue.edu

Truss Analysis and Visualization is a program for the force analysis of planar trusses and the visualizations of the results.
The program allows the user to :

* Create a new truss design or use an existing truss stored in memory.
* Alter the geometry of an existing truss design through moving the joint locations or by the addition/removal of truss members, truss supports or truss joints.
* Alter the loading on an existing truss.
* Produce free body diagrams of the entire truss or for selected pins.
* Produce a color graphical display of the load carried by the individual members with the color of the member denoting tension/compression and the magnitude of the load.

A sample truss design is shown below in Fig. 1 as it would appear on the screen prior to the force analysis.
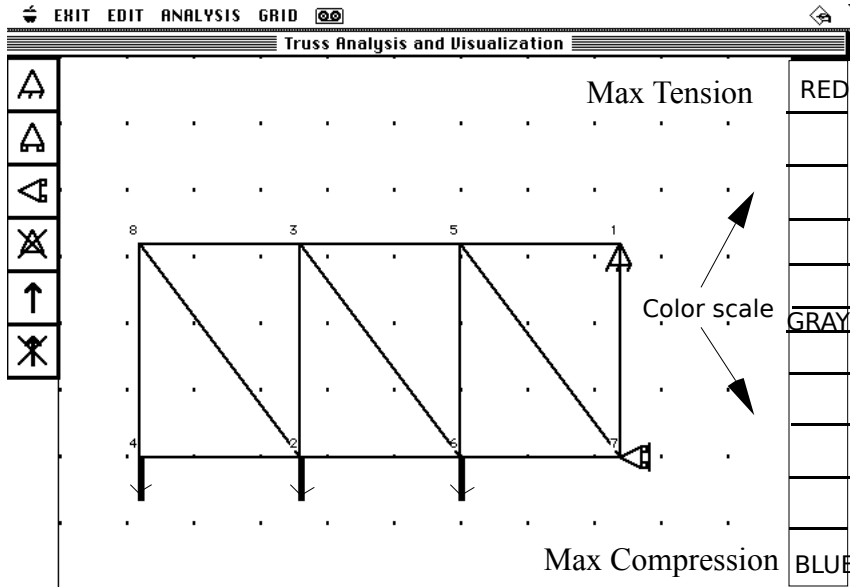
**Fig. 1** Screen view of TRUSS program.

The right column has the color scale for the members following the force analysis: Red colors indicate tension, and Blue colors indicate compression, and gray indicates no load. The left column has the Supports options and the Applied Forces option which can be called by clicking in the appropriate box after selecting the receiving node.

**USAGE OF TRUSS ANALYSIS:**

To initialize the program double click on the application named TRUSS.

**A. Creating a truss:**

This task is very simple:

> **1.** Choose "***Create Truss*** " from "***Edit*** " Menu.  This will produce a triangle on the screen which can be moved and altered by dragging the nodes.
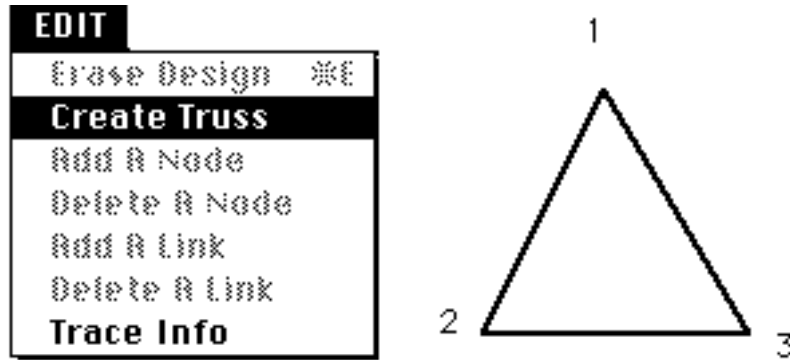
**Fig. 2**. Creating a TRUSS .
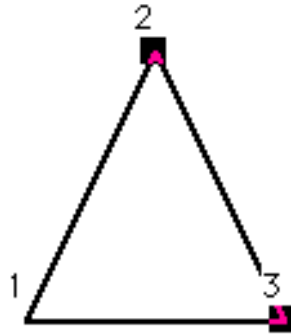
**2.** Select any two nodes by clicking on them.



**Fig. 3**. Selecting a member by selecting its nodes.

**3.** Choose "***Add a Node*** " from "***Edit*** " Menu.   This will add a node and connect it to the two selected nodes.
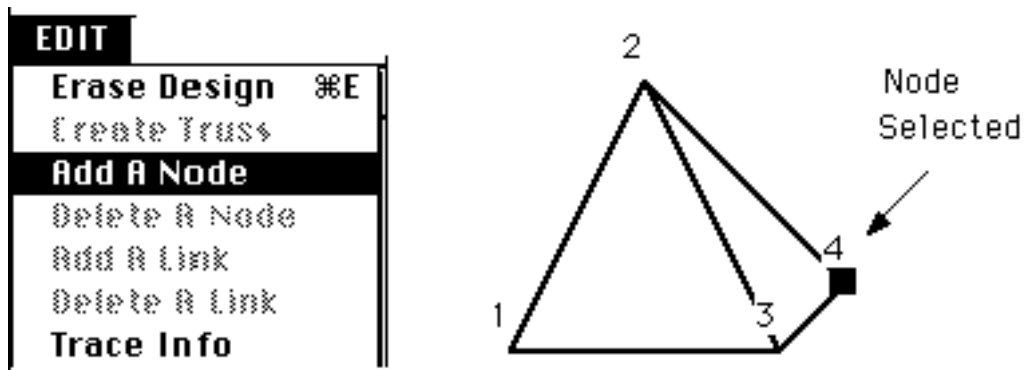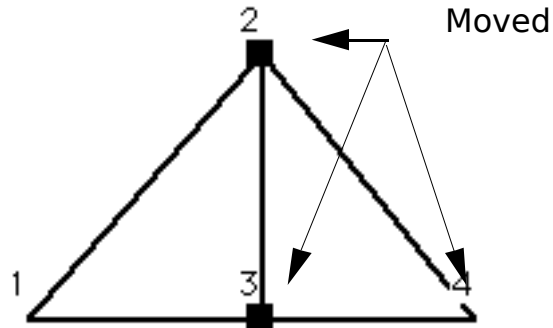


**Fig. 4.** The node added is selected automatically.

**4.** The newly created node can then be moved by dragging (with the mouse button down) it to the desired location.



**Fig. 5.** Moving the nodes to obtain the desired structure.

**5.** Subsequent addition of new members is accomplished by repetition of steps 3 and 4 above.

**6.** A support can be added at any node by first clicking on that node followed by a click on one of the three types of supports shown at the top of the left column of Fig. 1 ( pinned, horizontal roller, and vertical roller respectively).

**7.** Applying a nodal force is accomplished by first clicking on the appropriate node followed by a click on the force option in the left column (i.e., the "arrow" in the fifth box from the top). This will bring a dialog box on to the screen prompting the user for a force magnitude followed by a force direction (in degrees and measured from a right-oriented horizontal line). Type the desired numbers and hit **return** when done. Note that a positive force points away from the node, while a negative one will point towards the node.

```
Enter Force Value  200
```
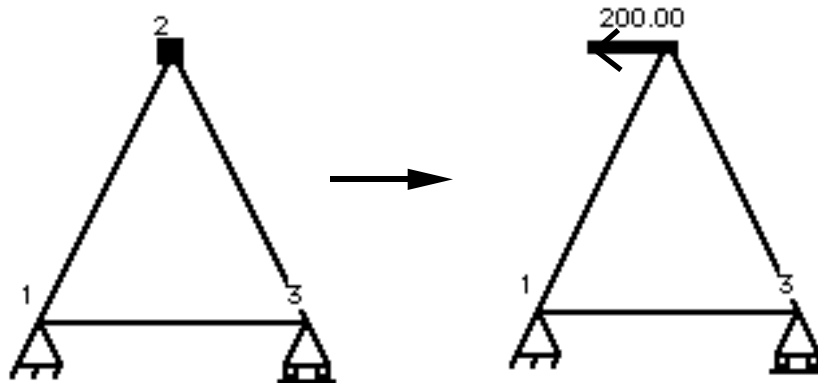
```
Enter Force Angle  180
```



**Fig. 6.** Loading

the truss.

## B. Alteration of the truss/loading:

**1.** Any node can be moved by first clicking on the node and dragging it to a new location.

**2**. Any member may be removed by first clicking on the nodes of that member and selecting "***Delete A Link*** " option in the "***EDIT*** " menu.

**3.** A member may be added by first clicking on two nodes for which a new member is desired followed by selecting "***Add A Link*** " option from the "***EDIT*** " menu.

**4.** Any force (support) can be removed by first clicking on the node on which the force (support) is at followed by a click on the remove force (support) option on the left Design Box shown as a crossed arrow ( crossed support).

**5.** The grid can be of help while creating or altering the truss and the "***Align Truss*** " option in the "***GRID*** " menu will correct any minor misalignments in the links.
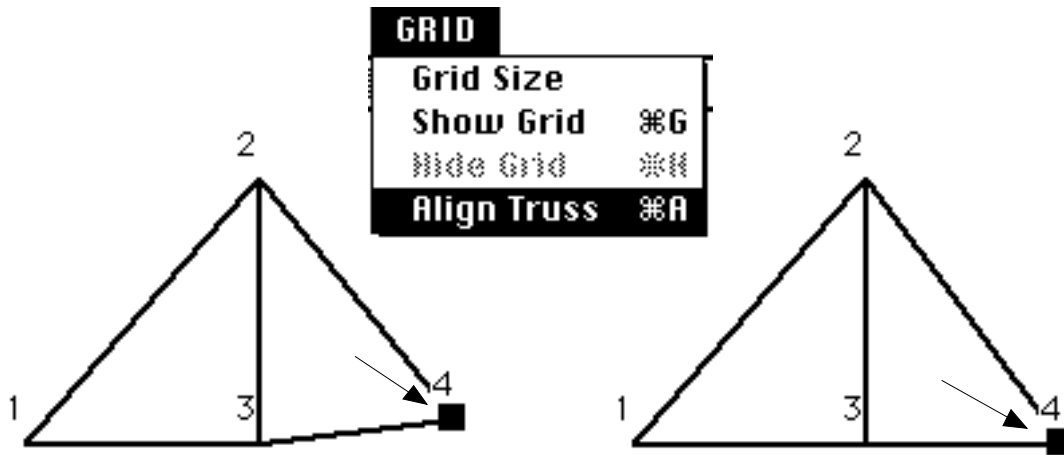
Fig.     7.

Aligning the truss in case of minor misalignments.

**6.** Dragging a node too close to a member will add the node to that member creating two connected members. Also, moving a node close to another node will make them one node.

7. The entire truss design can be removed by choosing " *Erase Design* " from the " *EDIT* " menu.

## C. Analyzing a truss:

Once the desired truss has been created the analysis step follows with several options:

**1**-Static Truss Solution:  Choose "*Solve Truss* " Option.  This will cause the truss members to be drawn in different intensities of **RED**(*tension*) and **BLUE** (*compression)* so that the forces in the members can be viewed relative to the color scale on the right of the screen.
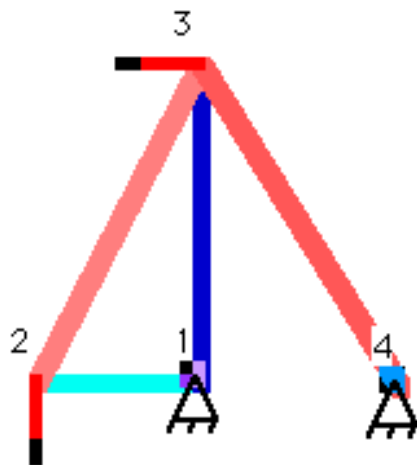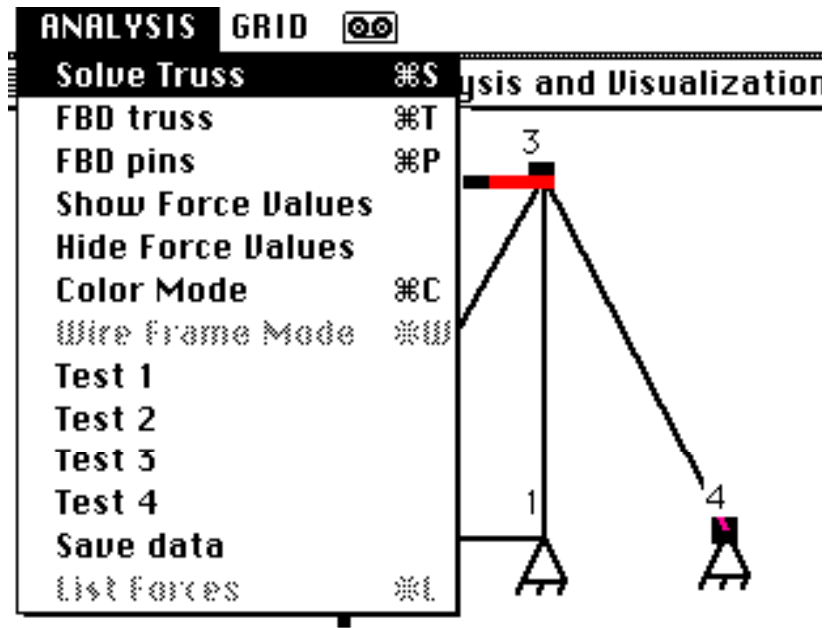
**Fig. 8.** The truss after analysis, each member is drawn with the color that indicates its force value.

**2-** A free body diagram of the whole truss can be viewed by selecting "***FBD Truss***" option. This will produce a wire frame truss and replace the supports with the appropriate reaction forces drawn in intensities of red with a green arrow tip showing the direction of the reaction force.
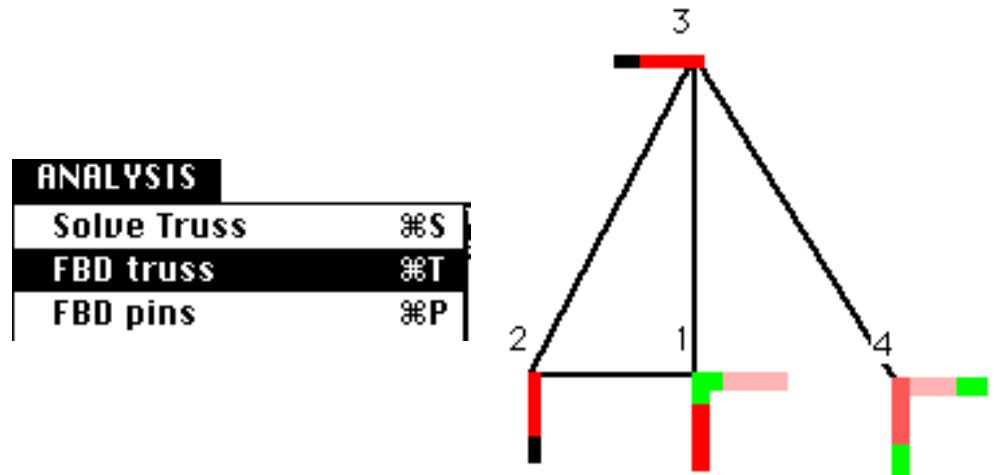
**Fig. 9**. Free body diagram of the truss showing ground forces at the supports.
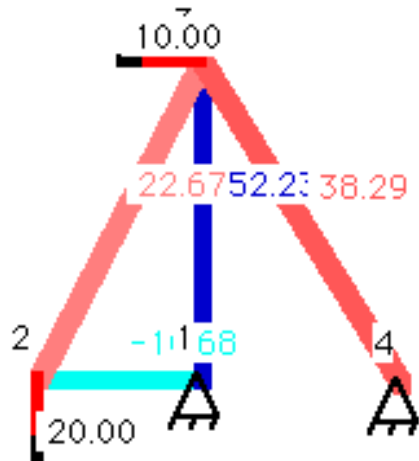
**3-** A free body diagram of a pin starts by selecting a node and choosing "***FBD Pin***". Colored arrows with a green tip will simulate the member forces exerted at the selected node.
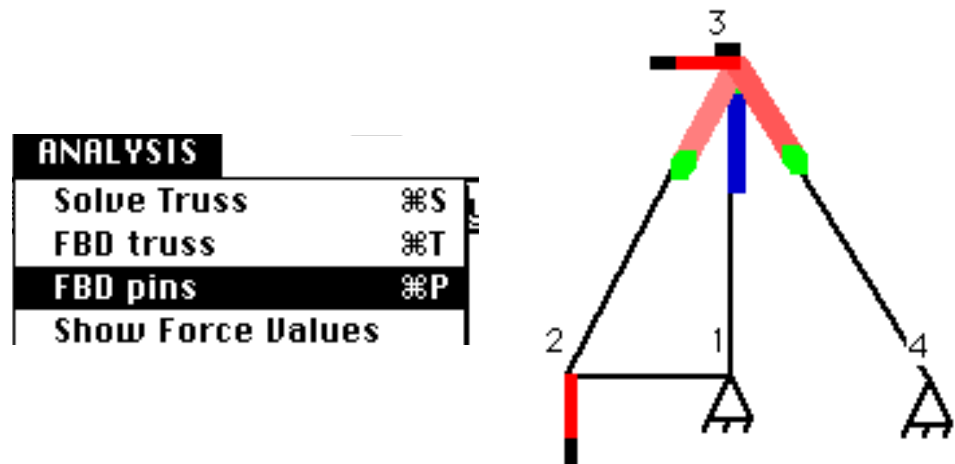


**Fig. 10.** The free body diagram of a selected node.

Force values can be called to appear on the screen or disappear by choosing either "***Show Force...***" or "***Hide Force...***".

**Fig. 11.** Analyzed truss with numerical data shown on the screen.

Member forces can be printed in a separate window if "***List Forces***" was chosen while in Color or in Free body diagram modes, a window will appear having all the member forces and ground reaction forces listed on it.
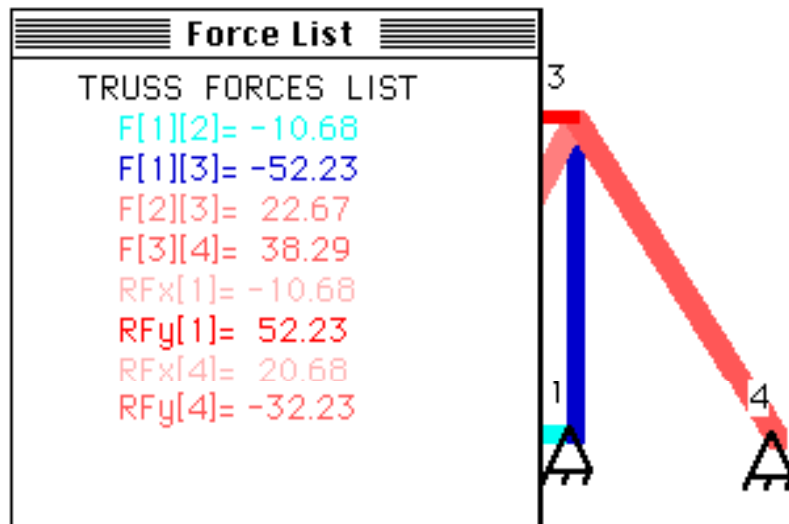


**Fig. 12.** Another way of obtaining numerical results.

If the truss is over-constrained or under-constrained, a dialog box will appear with the appropriate warning and the program will return to the wire-frame mode.
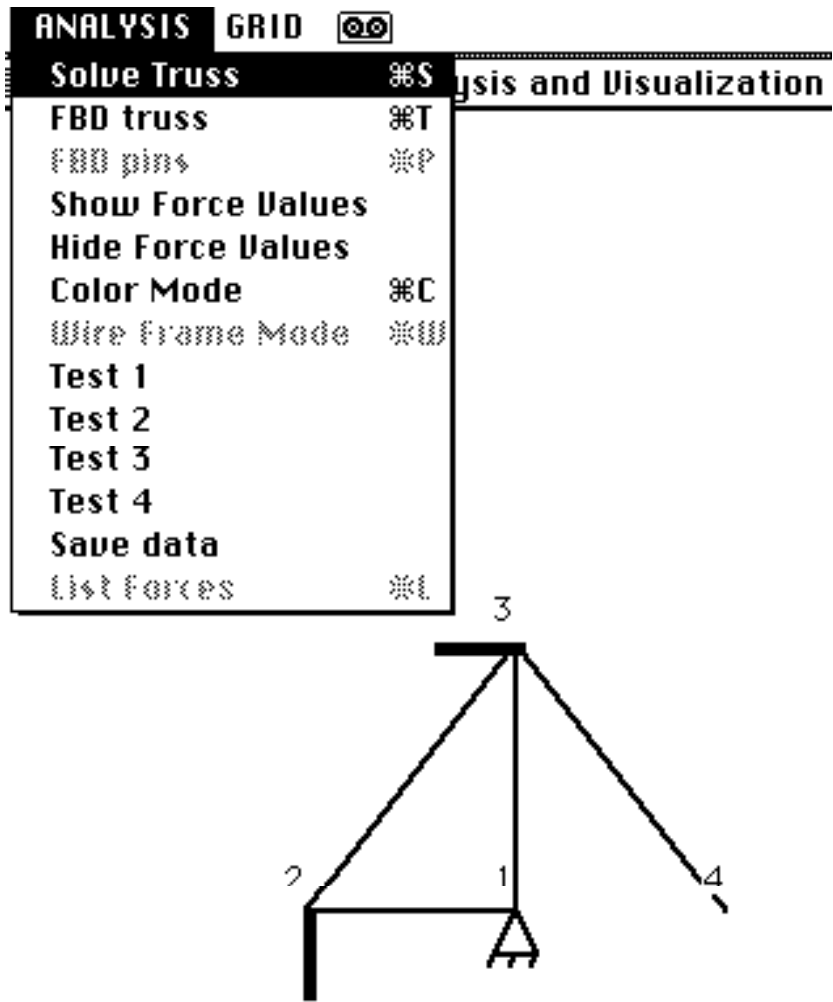
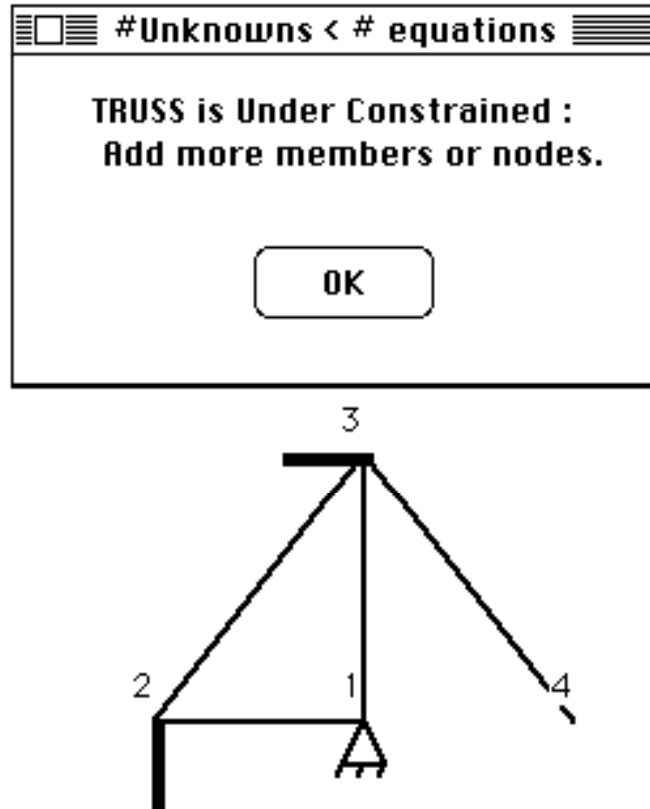**Fig. 13.** An improper truss missing a support (indeterminate)

**Fig. 14.** Dialog box indicating the problem.

## D. MENU OPTIONS

**Apple**
> **About Truss...**: Display credits and logo.
> **Help...**: Simple on-line help explaining menu functions.
> **Why the Beep...**: Explains why the computer beeps (or might beep).

**EXIT**
> **Save Truss...**: Save the current truss and parameters as an input file.
> **Open Truss...**: Load the truss saved by the previous menu option.
> **Invert Screen**: Change background color between black and white.
> **Copy Window**: Copy application window to the clipboard.
> **Print Window**: Print current window and member force values.
> **Quit**: Exit the program.

**EDIT**

      **Erase Design**: Erase the screen and initialize the program.

      **Create Truss**: Create a triangle which is the basis of the truss.

      **Add A Node:** When a member is selected (by selecting its nodes) another node is added, selected, and connected to that member.

      **Delete A Node:** Erase the node and all the members connected to it.

      **Add A Link:** When two nodes are selected and they are not connected, they will be joined.

      **Delete A Link:** Erase the member joining two selected nodes.

      **List Parameters:** Display information about the design in console.

      **Member Color Width:** Adjust thickness of members for easier viewing.

**ANALYSIS**

      **Solve Truss:** Produce a colored truss with the appropriate color indicating the force in each member.

      **FBD of Truss:** Produce a free body diagram of the whole truss showing the reactions at the supports.

      **FBD of a Pin:** Show the forces at a selected node.

      **Show/Hide Force Values:** Prints/hides numerical information on the screen like force magnitudes.

      **Mode:** View truss forces in colors / wire-frame mode.

      **Truss1 -> 4:** Pre-loaded truss examples.

      **Write Analysis Steps:** Will save solution steps from the analysis of  the truss (Matrix solution by GAUSS elimination method) to an external TEXT file called "Trace File"  which can be viewed using a word processor.

**GRID**

      **Grid Size:** Specify a desires grid spacing.

      **Show Grid:** Draws a grid in the design area.

      **Hide Grid:** Hides the grid.

      **Align Truss:** In case there were any minor crookedness in the member, it will be aligned.

**Window**

      Bring a window to the front (Truss window, Force values, or console).

## DESIGN BUTTONS

Add a pin support at the selected node

Add a x-roller support at the selected node

Add a y-roller support at the selected node

Remove a support from a selected node

Add a force to a selected node

Remove a force from a selected node

Toggle color / wire-frame modes

Translate truss left-right depending on the arrow selected

Translate truss up-down depending on the arrow selected