**PTrussMatlab**

**Program for the Linear Static Analysis of Plane Trusses with Matlab**

v1.00, 1 January 2016



## 1. Introduction

PTrussMatlab is a program for the linear static analysis of plane trusses developed by Vagelis Plevris (vplevris@gmail.com) and George Papazafeiropoulos (gpapazafeiropoulos@yahoo.gr). It is written in MATLAB programming language and is available as source code distributed under a BSD-style license (see License.txt).

## 2. Main features and characteristics

The main features and characteristics of the program are the following:

- The model is described in an input file (in text format).
- The program draws and shows the model in a simple figure.
- It calculates the Node Displacements, Element Forces, Stresses and Strains and the Constraint Reactions.
- The results of the analysis are written in an output file (in text format).
- It supports nodal forces, given in the global coordinate system (Fx, Fy).
- The source code is well commented, since it is developed mainly for educational purposes.

## 3. Matlab Source code files

The source files package consists of 8 Matlab (.m) files, listed below in alphabetical order:

| File name | Usage |
|---|---|
| 1. PTrussAssignElementProperties.m | Assigns properties for the n element |
| 2. PTrussDataInput.m | Reads model data from input file |
| 3. PTrussDataOutput.m | Saves analysis results to output file |
| 4. PTrussDrawModel.m | Draws the model on a figure |
| 5. PTrussElementLength.m | Returns the length of an element, given the coordinates of start |
| 6. PTrussElementStiffness.m | Returns the local stiffness matrix [2x2] of a 2D truss element |
| 7. PTrussElementTransformation.m | Returns the transformation matrix [2x4] of a 2D truss element |
| **8. PTrussMatlab.m** | Program for the Linear Static Analysis of Plane Trusss with Matlab (**main file**) |

## 4. Additional files

Apart from the above source code files, the package contains also some other files listed below:

| File name | Usage |
|---|---|
| License.txt | Contains the licensing information |
| Truss10.txt | Input file (example of a 2D truss model) which can be read and analyzed by the program |
| Truss10_Output.txt | Corresponding output file, which is created by the program if the program is run for the input file "Truss10.txt" |
| Version History.txt | Contains information about the successive versions of the program |
| Documentation.pdf | Documentation file (this file) |

## 5. Instructions for running the program

Follow the instructions below to run and use the program:

- Prepare a model input file, as shown in the instructions, or use the existing "**Truss10.txt**" example input file.
- Open the main file "**PTrussMatlab.m**" within Matlab and run it (Press F5).
- The Matlab Command Window asks for the name of the input file. Note that the input file has to be located at the same directory as the source files. Type the file name and then hit "Enter".
- The program reads the model. A figure window appears, showing the Model.
- If the analysis has been carried out successfully, then the output file is generated in the same directory ("**Truss10 Output.txt**" for the example input file).
- Open the output file with a text editor and see the analysis results.

## 6. Example: Analysis of a 10-bar plane truss (included file "Truss10.txt")

This is a standard benchmark 10 bar plane truss shown in the figure below with the following structural characteristics: Modulus of Elasticity $E$**=10000ksi**, material weight $\rho$**=0.1lb/in$^3$**, length $L$**=360in**, load $P$**=100kip**.

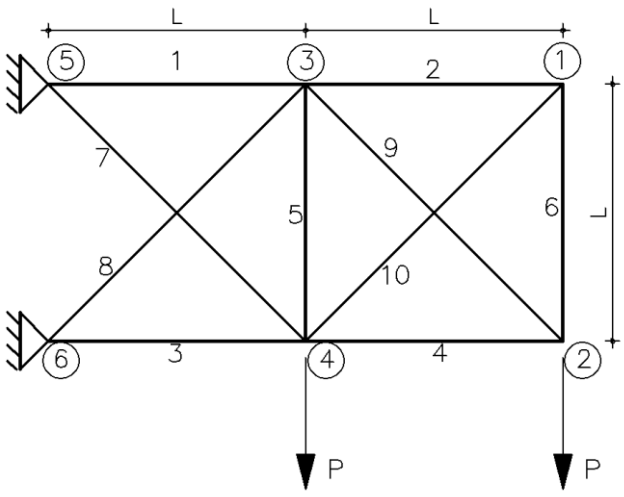


**Figure 1. 10 bar plane truss: The truss model.**

The structural members are divided into 10 groups as shown in the table below.

**Table 1. 10 bar plane truss: Section area for each member.**

| Member ID | Section Area ($in^2$) |
|---|---|
| 1 | 30.9810 |
| 2 | 0.1000 |
| 3 | 23.1714 |
| 4 | 15.6935 |
| 5 | 0.1000 |
| 6 | 0.5848 |
| 7 | 7.4298 |
| 8 | 20.6310 |
| 9 | 21.3287 |
| 10 | 0.1000 |

### 6.1 Input file: Truss10.txt

The input file corresponding to the above model (Truss10.txt) is as follows, where explanations are given in red for each line:

```
1    *PTrussMatlab input file Beginning of input file
2    Blank line
3    *Materials (Material_ID, E) Definition of materials
4    1 Number of materials
5    1  1e4 Material ID and modulus of elasticity
6    Blank line
7    *Sections (Section_ID, A) Definition of sections
8    10 Number of sections
9    1  30.9810 Section ID and Area (for lines: 9-18)
10   2  0.1
11   3  23.1714
12   4  15.6935
13   5  0.1
14   6  0.5848
15   7  7.4298
16   8  20.6310
17   9  21.3287
18   10 0.1
19   Blank line
20   *Nodes (Node_ID, Xcoordinate, Ycoordinate) Definition of nodes
21   6 Number of nodes
22   1  720    360 Node ID, x-coordinate and y-coordinate (for lines: 22-27)
23   2  720    0
24   3  360    360
25   4  360    0
26   5  0      360
27   6  0      0
28   Blank line
29   *Elements (Element ID, Start Node i, End Node j, Material Group, Section Group)
Definition of elements
30   10 Number of Elements
31   1 5 3 1 1 Element ID, start node, end node, material ID, section ID (for lines: 31-40)
32   2 3 1 1 2
33   3 6 4 1 3
34   4 4 2 1 4
35   5 3 4 1 5
36   6 1 2 1 6
37   7 5 4 1 7
```

```
8  6  3  1  8
9  3  2  1  9
10  4  1  1  10
Blank line
*Constraints (Node_ID, x_constr, y_constr) Definition of Constraints (Supports)
2 Number of Constrained Nodes
5  1      1 Node ID, x-Constraint y-Constraint (for lines: 44-45)
6  1      1
Blank line
*Nodal_Loads (Node_ID, x_load, y_load) Definition of Nodal Loads
2 Number of Nodes with Loads on
2  0      -100 Node ID, Fx, Fy (for lines: 49-50)
4  0      -100
```

### 6.2 Graphical representation of the model in Matlab

The program draws the model in a simple figure where the following are shown:

- Nodes with their ID
- Elements with their ID
- Constraints, in the form xx (hinge support) or ox/xo (free translation along x or y global axis, respectively), where for each DOF **x** means "fixed" and **o** means "free to move"

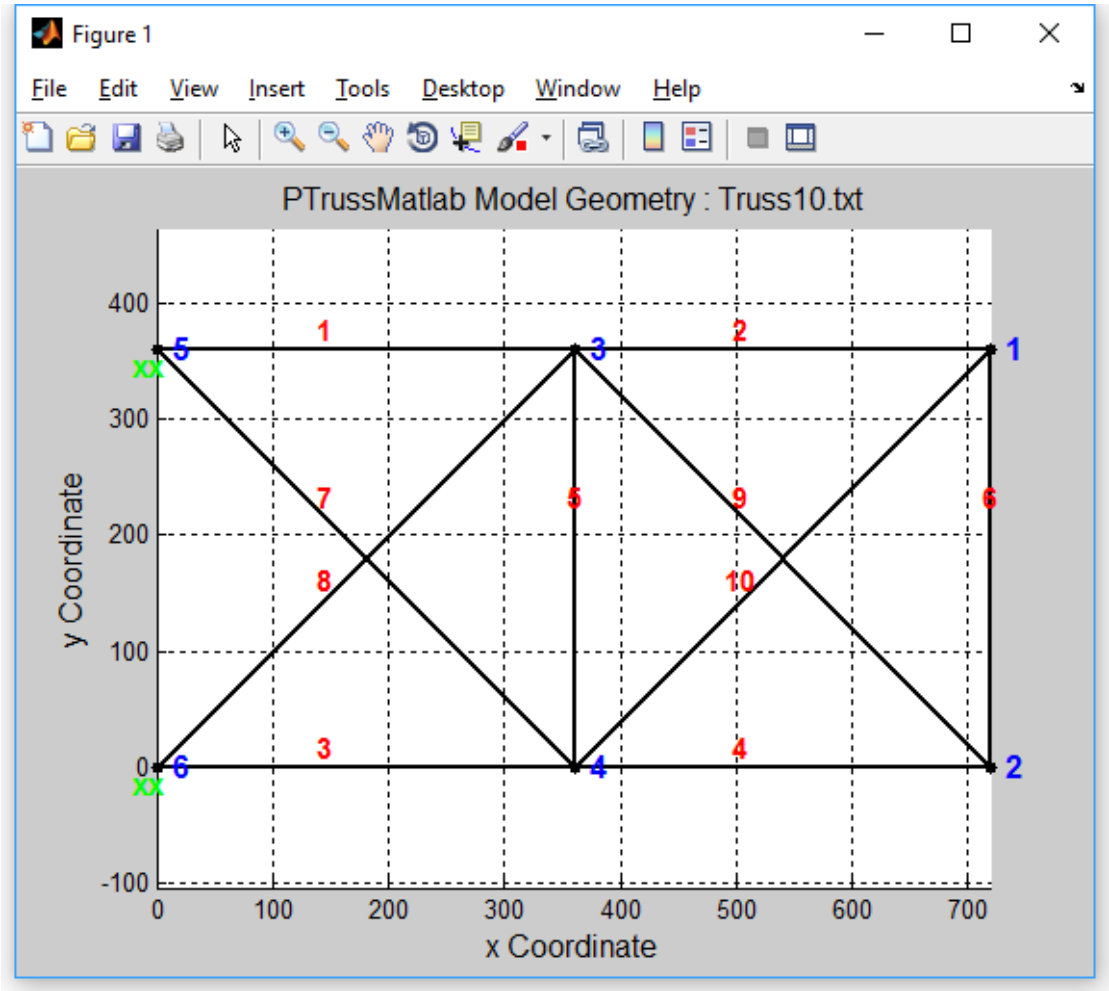


**Figure 2. 10 bar plane truss: The truss model drawn in PTrussMatlab figure.**

### 6.3 Output file: Truss10_Output.txt

The output file with the results of the analysis of the above model "Truss10.txt" is as follows, where explanations are given in red for each line:

```
1   *PTrussMatlab output file Beginning of output file
2   Blank line
3   *Node Displacements (global system)
4   Blank line
5     ID   x-displacement   y-displacement Table header
6      1       1.8761E-01      -2.0000E+00 Node ID, x-displacement, y-displacement (for lines: 6-11)
7      2      -5.3634E-01      -1.9918E+00
8      3       2.3546E-01      -7.4198E-01
9      4      -3.0664E-01      -1.6411E+00
10     5       0.0000E+00       0.0000E+00
11     6       0.0000E+00       0.0000E+00
12  Blank line
13  *Element Forces, Stresses and Strains
14  Blank line
15   ID    Axial Force   Axial Stress     Axial Strain Table header
16     1       202.6303         6.5405       6.5405E-04 Element ID, axial force, stress and strain (for lines: 16-25)
17     2        -0.1329        -1.3290      -1.3290E-04
18     3      -197.3697        -8.5178      -8.5178E-04
19     4      -100.1329        -6.3805      -6.3805E-04
20     5         2.4974        24.9745       2.4974E-03
21     6        -0.1329        -0.2273      -2.2726E-05
22     7       137.7015        18.5337       1.8534E-03
23     8      -145.1412        -7.0351      -7.0351E-04
24     9       141.6093         6.6394       6.6394E-04
25    10         0.1879         1.8795       1.8795E-04
26  Blank line
27  *Constraint Reactions (global system)
28  Blank line
29   ID  DOF       Reaction Table header
30     5    1      -300.0000 Node ID, DOF ID, Support Reaction (for lines: 30-33)
31     5    2        97.3697
32     6    1       300.0000
33     6    2       102.6303
```