STrussMatlab

Program for the Linear Static Analysis of Space (3D) Trusses with Matlab

v1.00, 4 January 2016

1. Introduction

STrussMatlab is a program for the linear static analysis of space (3D) 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 3D model is described in an input file (in text format).
- The program draws and shows the model in a simple figure (in 3D). The user can rotate the figure and view the 3D model in different angles.
- 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, Fz).
- The source code is well commented, since it is developed mainly for educational purposes.
- The results of the program have been verified by comparison with the results of ABAQUS (v 6.13.1) and SAP2000 (v 17.2.0) software (see "Verification files").

3. Matlab source code files

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

File name	Usage
1. STrussAssignElementProperties.m	Assigns properties for the n^{th} element
2. STrussDataInput.m	Reads model data from input file
3. STrussDataOutput.m	Saves analysis results to output file
4. STrussDrawModel.m	Draws the model on a figure
5. STrussElementLength.m	Returns the length of an element, given the coordinates of start and end nodes
6. STrussElementStiffness.m	Returns the local stiffness matrix [2x2] of a 3D truss element
7. STrussElementTransformation.m	Returns the transformation matrix [2x6] of a 3D truss element
8. STrussMatlab.m	Program for the Linear Static Analysis of Space (3D) Trusses 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
Truss3D-25.txt	Input file (example of a 3D truss model) which can be read and analyzed by the program
Truss3D-25_Output.txt	Corresponding output file, which is created by the program if the program is run for the input file "Truss3D-25.txt"
Version History.txt	Contains information about the successive versions of the program
Documentation.pdf	Documentation file (this file)

There is also a directory, named "**Verification files**". It contains the files needed for the verification of the program's results by comparison with the results of ABAQUS and SAP2000 software. For details, please see the section "Verification files" below.

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 example input file ("**Truss3D-25.txt"**).
- Open the main file "**STrussMatlab.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 with its file extension ("**Truss3D-25.txt"** for the example input file) and then hit "Enter".
- The program reads the model. A figure window appears, showing the Model in 3D.
- If the analysis has been carried out successfully, then the output file is generated in the same directory ("**Truss3D-25_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 25-bar space truss (included file "Truss3D-25.txt")

The test example is a 25-member space truss. The structure is depicted in Figures 1 and 2 below.

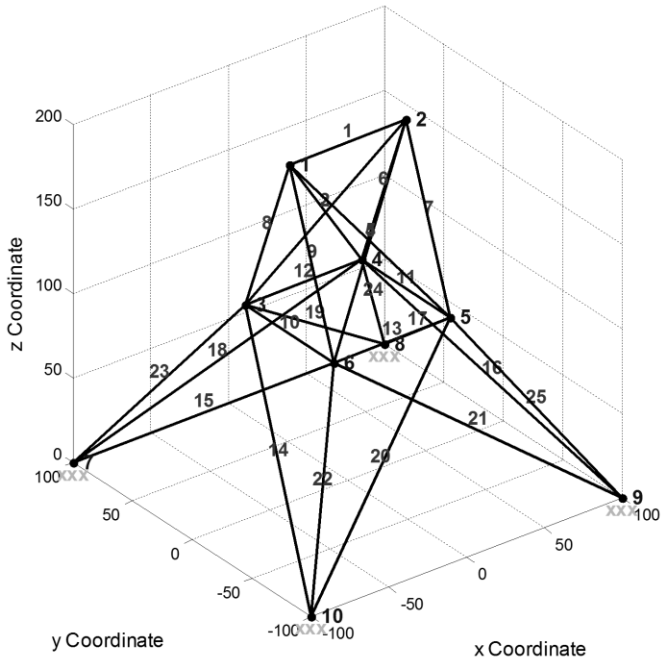

Figure 1. 25-bar space truss: 3D view of the truss model (coordinates in inches).

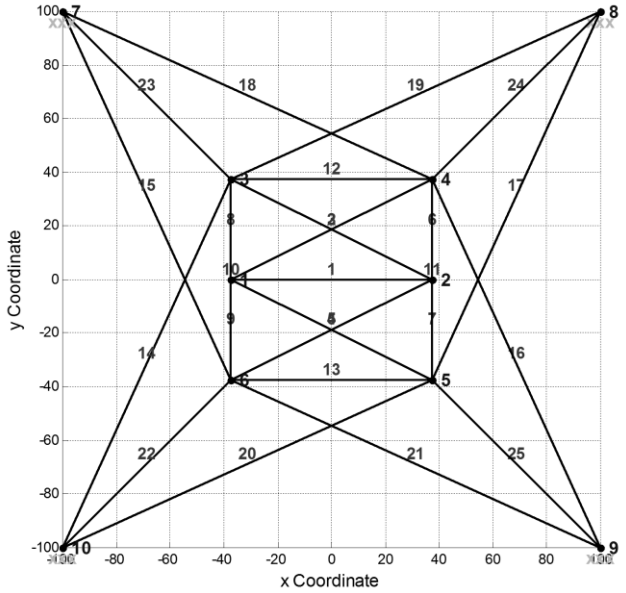

Figure 2. 25-bar space truss: Top view of the truss model (coordinates in inches).

The “xxx” label for nodes 7, 8, 9, 10 denotes restrictions that apply to all three DOFs of the node (pinned constraint). The Nodes coordinates are shown in Table 1.

Table 1. 25-bar space truss: Nodes Coordinates.

Node	*x* (in)	*y* (in)	*z* (in)
1	-37.5	0	200
2	37.5	0	200
3	-37.5	37.5	100
4	37.5	37.5	100
5	37.5	-37.5	100
6	-37.5	-37.5	100
7	-100	100	0
8	100	100	0
9	100	-100	0
10	-100	-100	0

The Modulus of Elasticity of the material is ***E*=10000 ksi**. The structural members are divided into 8 groups as shown in the table below.

Table 2. 25-bar space truss: Member groups and section areas.

Group	Member ID	Section area (in²)
1	1	0.01
2	2-5	2.043
3	6-9	3.00239
4	10,11	0.01
5	12,13	0.01
6	14-17	0.68337
7	18-21	1.62296
8	22-25	2.67194

The nodal loads are presented in the table below.

Table 3. 25-bar space truss: Nodal Loads.

Node	F_x (kip)	F_y (kip)	F_z (kip)
1	1	10	-5
2	0	10	-5
3	0.5	0	0
6	0.5	0	0

6.1 Input file: "Truss3D-25.txt"

The input file corresponding to the above model ("Truss3D-25.txt") is as follows, where explanations are given in red for each line:

```
1  **SFrameMatlab 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  8 Number of sections
9  1    0.01 Section ID and Area (for lines: 9-16)
10  2   2.043
11  3   3.00239
12  4   0.01
13  5   0.01
14  6   0.68337
15  7   1.62296
16  8   2.67194
17  Blank line
18  *Nodes (Node_ID, Xcoordinate, Ycoordinate, Zcoordinate) Definition of nodes
19  10 Number of nodes
20  1   -37.5   0       200 Node ID, x-coordinate, y-coordinate and z-coordinate (for lines: 20-29)
21  2   37.5    0       200
22  3   -37.5   37.5    100
23  4   37.5    37.5    100
24  5   37.5    -37.5   100
25  6   -37.5   -37.5   100
26  7   -100    100     0
27  8   100     100     0
28  9   100     -100    0
29  10  -100    -100    0
30  Blank line
31  *Elements (Element ID, Start Node i, End Node j, Material Group, Section Group)
32  25 Number of Elements                                          Definition of elements
33  1 1 2 1 1 Element ID, Start Node, End Node, Material ID, Section ID (for lines: 33-57)
34  2 4 1 1 2
35  3 2 3 1 2
36  4 2 6 1 2
37  5 1 5 1 2
38  6 2 4 1 3
39  7 2 5 1 3
40  8 1 3 1 3
41  9 1 6 1 3
42  10 3 6 1 4
43  11 5 4 1 4
44  12 3 4 1 5
45  13 5 6 1 5
46  14 3 10 1 6
47  15 7 6 1 6
48  16 9 4 1 6
49  17 8 5 1 6
50  18 7 4 1 7
51  19 3 8 1 7
52  20 10 5 1 7
53  21 9 6 1 7
54  22 10 6 1 8
55  23 3 7 1 8
56  24 8 4 1 8
57  25 9 5 1 8
58  Blank line
59  *Constraints (Node_ID, x_constr, y_constr, z_constr) Definition of Constraints (Supports)
60  4 Number of Constrained Nodes
61  7   1 1 1 Node ID, x-Constraint y-Constraint z-Constraint (for lines: 61-64)
62  8   1 1 1
63  9   1 1 1
64  10  1 1 1
65  Blank line
66  *Nodal_Loads (Node_ID, x_load, y_load, z_load) Definition of Nodal Loads
67  4 Number of Nodes with Loads on
68  1   1       10      -5 Node ID, Fx, Fy, Fz (for lines: 68-71)
69  2   0       10      -5
70  3   0.5      0       0
71  6   0.5      0       0
```

6.2 Graphical representation of the 3D 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, as shown in the table below.

Table 4. Constraint cases in PTrussMatlab.

Constraint case (Dx, Dy, Dz DOFs)	Constrained DOFs	Free DOFs
xxx	Dx, Dy, Dz	-
xxo	Dx, Dy	Dz
xox	Dx, Dz	Dy
oxx	Dy, Dz	Dx
xoo	Dx	Dy, Dz
oxo	Dy	Dx, Dz
oox	Dz	Dx, Dy
-	-	Dx, Dy, Dz

For each of the three DOFs (Dx, Dy, Dz), "**x**" means "fixed" (not allowed to move) and "**o**" means "free" (free to move).

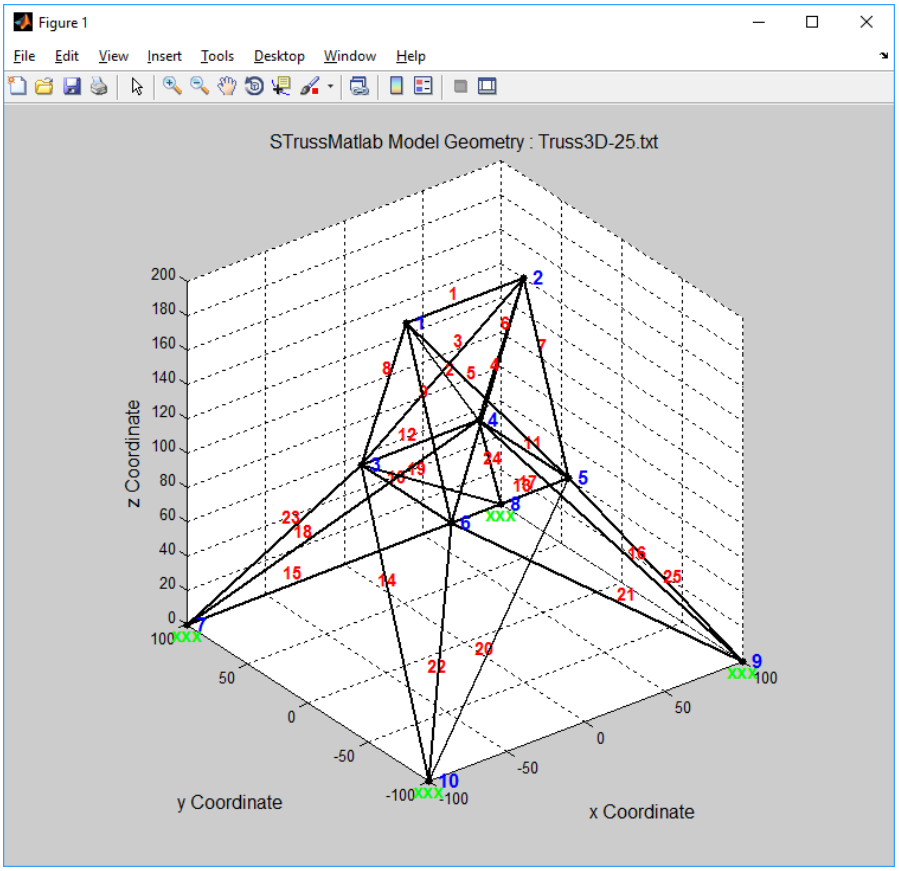

Figure 3. 25-bar space truss: The truss model drawn in STrussMatlab figure.

6.3 Output file: "Truss3D-25_Output.txt"

The output file with the results of the analysis of the above model "**Truss3D-25.txt**" is shown below, where explanations are given in red for each line:

```
 1  *STrussMatlab output file Beginning of output file
 2  Blank line
 3  *Node Displacements (global system)
 4  Blank line
 5   ID   x-displacement   y-displacement   z-displacement Table header
 6    1       5.6960E-03       3.5000E-01      -2.2985E-02 Node ID, Dx, Dy, Dz (for lines: 6-15)
 7    2       3.3616E-02       3.5000E-01      -3.2792E-02
 8    3       1.9355E-02      -3.8354E-02      -1.2356E-01
 9    4      -1.2259E-02      -3.9940E-02      -1.3125E-01
10    5       4.2191E-03      -2.4729E-02       8.0049E-02
11    6       2.8771E-03      -2.6315E-02       8.7746E-02
12    7       0.0000E+00       0.0000E+00       0.0000E+00
13    8       0.0000E+00       0.0000E+00       0.0000E+00
14    9       0.0000E+00       0.0000E+00       0.0000E+00
15   10       0.0000E+00       0.0000E+00       0.0000E+00
16  Blank line
17  *Element Forces, Stresses and Strains
18  Blank line
19   ID    Axial Force   Axial Stress     Axial Strain Table header
20    1         0.0372         3.7226       3.7226E-04 El. ID, axial force, stress and strain (for lines: 20-44)
21    2        -6.1690        -3.0196      -3.0196E-04
22    3        -5.2989        -2.5937      -2.5937E-04
23    4         5.2342         2.5620       2.5620E-04
24    5         4.3641         2.1361       2.1361E-04
25    6       -12.5735        -4.1878      -4.1878E-04
26    7         7.2865         2.4269       2.4269E-04
27    8       -11.8615        -3.9507      -3.9507E-04
28    9         7.9985         2.6641       2.6641E-04
29   10        -0.0161        -1.6052      -1.6052E-04
30   11        -0.0203        -2.0281      -2.0281E-04
31   12        -0.0422        -4.2152      -4.2152E-04
32   13         0.0018         0.1789       1.7892E-05
33   14        -3.4196        -5.0040      -5.0040E-04
34   15         2.6185         3.8317       3.8317E-04
35   16        -3.7177        -5.4402      -5.4402E-04
36   17         2.3204         3.3955       3.3955E-04
37   18        -6.0910        -3.7530      -3.7530E-04
38   19        -6.2419        -3.8460      -3.8460E-04
39   20         3.4819         2.1454       2.1454E-04
40   21         3.3309         2.0524       2.0524E-04
41   22        10.9649         4.1037       4.1037E-04
42   23       -13.1234        -4.9116      -4.9116E-04
43   24       -14.7946        -5.5370      -5.5370E-04
44   25         9.2938         3.4783       3.4783E-04
45  Blank line
46  *Constraint Reactions (global system)
47  Blank line
48   ID  DOF       Reaction Table header
49    7    1         9.8657 Node ID, DOF ID, Support Reaction (for lines: 49-60)
50    7    2        -6.2596
51    7    3        11.7500
52    8    1       -10.8657
53    8    2        -7.3205
54    8    3        13.2500
55    9    1         5.5979
56    9    2        -2.6795
57    9    3        -6.7500
58   10    1        -6.5979
59   10    2        -3.7404
60   10    3        -8.2500
```

7. Verification files

The analysis results of the program have been verified with two well-known commercial Finite Element Analysis programs:

- **ABAQUS (v 6.13)**
- **SAP2000 (v 17.2.0)**

The package contains the input and output files of the two programs for the 25-bar 3D truss example (PTrussMatlab file "**Truss3D-25.txt**"). The files are listed below:

File name	Usage
ABAQUS-Truss3D-25.inp	**ABAQUS** Input file for the 25-bar 3D truss example to be run with ABAQUS software (ABAQUS/Command)
ABAQUS-Truss3D-25.dat	**ABAQUS** Output file created if the input file ABAQUS-Truss3D-25.inp is run with ABAQUS software
SAP2000-Truss3D-25.sdb	**SAP2000** input file for the 25-bar 3D truss example to be run with SAP2000 software
SAP2000-Truss3D-25 RESULTS.xlsx	**Excel** file with the results of SAP2000 software for the 25-bar 3D truss example (generated with Export > Tables from SAP2000)

By comparing the STrussMatlab output file ("**Truss3D-25_Output.txt**") with the output files of ABAQUS and SAP2000, one can easily see that the results for the truss example practically coincide.

7.1 ABAQUS Modeling remarks

The following have been applied in ABAQUS for the analysis of the test example:

- **Element type**: 2-node linear displacement 3D stress/displacement truss element (T3D2) were used with the option: *ELEMENT, TYPE=T3D2
- **Sections**: For all sections the *SOLID SECTION, MATERIAL=…, ELSET=… option was used, where the ELSET parameter corresponds to the element set containing the elements of a specified section group, and the MATERIAL parameter corresponds to the material definition.
- **Constraints**: For all constraints the *BOUNDARY option was used, where the degrees of freedom 1 to 3 were fixed.
- **Analysis**: The options *STEP and *STATIC were used to perform static analysis.
- **Nodal loads**: For all nodal loads the *CLOAD option was used
- **Output**: The options *EL PRINT and *NODE PRINT were used to print element and node variable output respectively to the data file.

7.2 SAP2000 Modeling remarks

The following settings have been applied in SAP2000 for the analysis of the test example:

- **Active Degrees of Freedom for the model**: UX=Yes , UY=Yes, UZ=Yes, RX=No, RY=No, RZ=No (Analyze > Analysis Options):

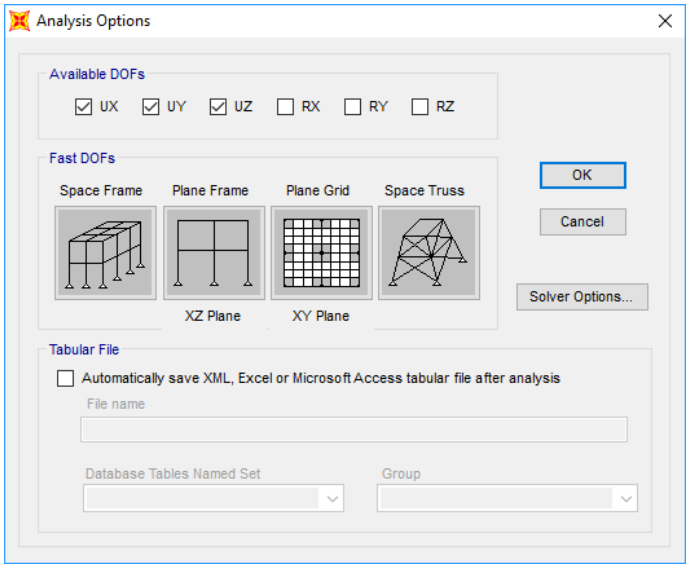

Figure 4. 25-bar space truss: Analysis Options in SAP2000.

- **Element releases**: For all elements (Assign > Frame releases):
  - Releases End-I: T, M2, M3
  - Releases End-J: M2, M3

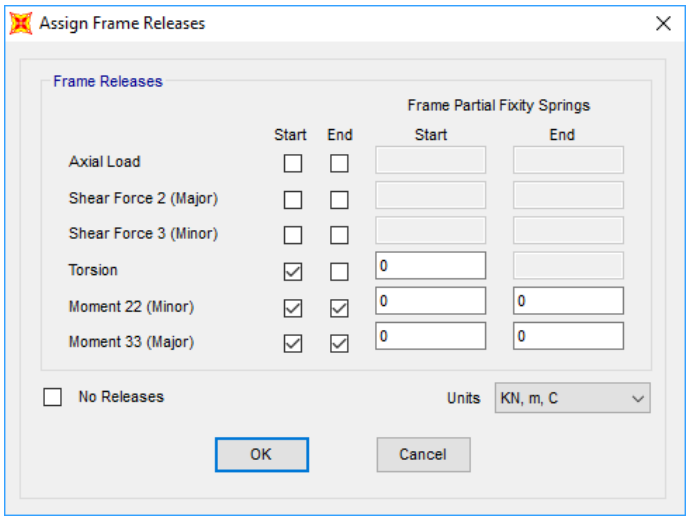

Figure 5. 25-bar space truss: Frame Element Releases in SAP2000.

- **Section modeling**: All sections have been modeled as rectangular sections with the following properties:
  - Depth (t3) = Given Area for the specific Group (from Table 2)
  - Width (t2) = 1 (Unity, so that t3·t2= Given Area)

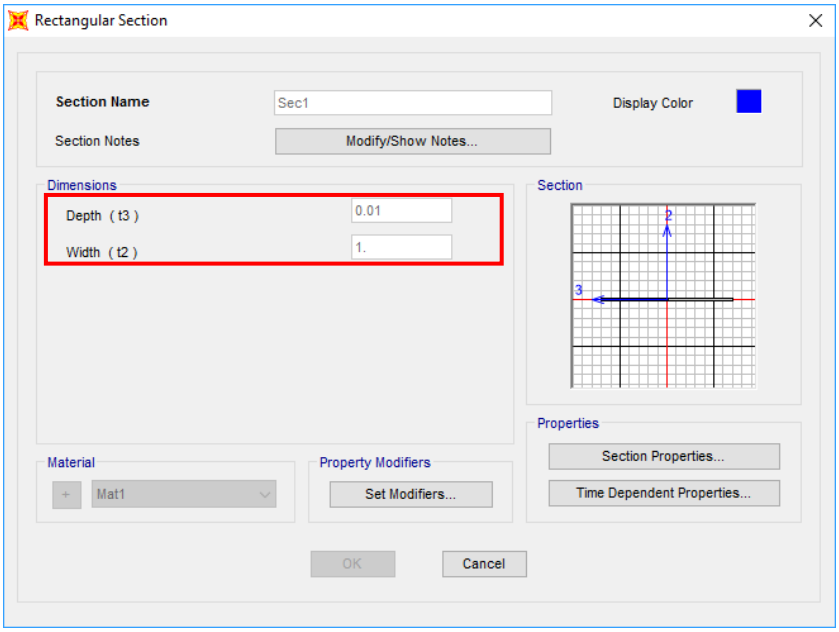

Figure 6. 25-bar space truss: Rectangular Section properties (for Group 1) in SAP2000.