CEMEF-UTN v1.1-6
High Voltage Tower Example

Marcos C. Ruggeri
`mruggeri@frh.utn.edu.ar`

Jefe de Proyecto CEMEF-UTN
Grupo de Simulación y Mecánica Computacional
Universidad Tecnológica Nacional
Facultad Regional Haedo

June 19, 2011

A How to get the program

The latest version of CEMEF-UTN can be downloaded for free at the ATOMS Portal Homepage, directly from the following link:

`http://atoms.scilab.org/`

This code runs under the Scilab platform and is compatible with Scilab 5.2 and later releases.

B Preliminaries

1. Once in the Scilab Console Window, change the current working directory to the location of the cemef-utn1.1-6 folder

   `File -> Change current directory...`

2. Load all .sci scripts into Scilab

   Note: Replace the cemef-utn path depending on the location of the cemef-utn1.1-6 folder containing the .sci scripts.

   `-->exec('<cemef-utn_path>/cemef-utn1.1-6/inpbar3d.sci', -1)`

   `-->exec('<cemef-utn_path>/cemef-utn1.1-6/stiffbar3d.sci', -1)`

   `-->exec('<cemef-utn_path>/cemef-utn1.1-6/assemb3d.sci', -1)`

   `-->exec('<cemef-utn_path>/cemef-utn1.1-6/preplottruss3d.sci', -1)`

   `-->exec('<cemef-utn_path>/cemef-utn1.1-6/SPDverif.sci', -1)`

   `-->exec('<cemef-utn_path>/cemef-utn1.1-6/bc.sci', -1)`

   `-->exec('<cemef-utn_path>/cemef-utn1.1-6/SPDsolve.sci', -1)`

   `-->exec('<cemef-utn_path>/cemef-utn1.1-6/stressbar3d.sci', -1)`

   `-->exec('<cemef-utn_path>/cemef-utn1.1-6/postplottruss3d.sci', -1)`

C Working with CEMEF-UTN routines

1. Import the .xls data into scilab using inpbar3d.sci

   ```
   -->head_comments('inpbar3d')

   -->[M,nod,elem,E,A,L,dircos,nn,ne] = inpbar3d('example_hvtower.xls');
   ```

2. Implement the 3D linear bar element formulation using stiffbar3d.sci

   ```
   -->head_comments('stiffbar3d')

   -->[kL,B,T,kG]=stiffbar3d(E,A,L,dircos,ne);
   ```

3. Assemble the stiffness matrix of the structure using assemb3d.sci

   Note: Depending on the element formulation, DOFn and DOFe might change. For the 3D linear bar element, there are 3 DOFs per node in the global coordinate system and thus 6 DOFs per element in the global coordinate system, since the element contains 2 nodes.

   ```
   -->head_comments('assemb3d')

   -->[tc,K0]=assemb3d(elem,3,6,kG,nn,ne);
   ```

4. Plot the structure in the undeformed position using preplottruss3d.sci

   ```
   -->head_comments('preplottruss3d')

   -->preplottruss3d(nod,elem,nn,ne,2)
   ```

5. Apply boundary conditions to the problem using bc.sci

   ```
   -->head_comments('bc')

   -->[K,P,cDOF,cDOFasc]=bc(M,K0);
   ```

6. Verify the stiffness matrix is not singular using SPDverif.sci

   ```
   -->head_comments('SPDverif')

   -->SPDverif(K0,cDOFasc,K);
   ```

7. Solve the matrix equations system using SPDsolve.sci

```
-->head_comments('SPDsolve')

-->[u0,u,P0]=SPDsolve(K0,K,P,cDOF,cDOFasc,nn);
```

8. Compute the stresses on each bar using stressbar3d.sci

```
-->head_comments('stressbar3d')

-->[ue,sigma,f]=stressbar3d(E,A,B,T,tc,u0,ne);

-->sigma
```

9. Plot the structure in the deformed position using postplottruss3d.sci

   Note: This function is intended to plot the structure on top of the previous plot. For this reason the nodes and element numbering are not recalled again, in the case that the first graphic was closed before.

```
-->head_comments('postplottruss3d')

-->postplottruss3d(nod,elem,nn,ne,2,sigma,u0,100,24)
```

D   Scilab Console Window

```
-------------------------------------------
               scilab-5.3.2

       Consortium Scilab (DIGITEO)
       Copyright (c) 1989-2011 (INRIA)
       Copyright (c) 1989-2007 (ENPC)
-------------------------------------------


Startup execution:
  loading initial environment

-->

/usr/scilab-5.3.2

-->

/home/mcr/cemef-utn1.1-6
```

```
-->exec('/home/mcr/cemef-utn1.1-6/inpbar3d.sci',-1);

-->exec('/home/mcr/cemef-utn1.1-6/stiffbar3d.sci',-1);

-->exec('/home/mcr/cemef-utn1.1-6/assemb3d.sci',-1);

-->exec('/home/mcr/cemef-utn1.1-6/preplottruss3d.sci',-1);

-->exec('/home/mcr/cemef-utn1.1-6/bc.sci',-1);

-->exec('/home/mcr/cemef-utn1.1-6/SPDverif.sci',-1);

-->exec('/home/mcr/cemef-utn1.1-6/SPDsolve.sci',-1);

-->exec('/home/mcr/cemef-utn1.1-6/stressbar3d.sci',-1);

-->exec('/home/mcr/cemef-utn1.1-6/postplottruss3d.sci',-1);

-->dir
 ans  =

BSDlicense.txt bc.sci              preplottruss3d.sci
SPDsolve.sci    example_hvtower.xls stiffbar3d.sci
SPDverif.sci    inpbar3d.sci        stressbar3d.sci
assemb3d.sci    postplottruss3d.sci

-->[M,nod,elem,E,A,L,dircos,nn,ne]=inpbar3d('example_hvtower.xls');
Warning : redefining function: file .
Use funcprot(0) to avoid this message


 ---> PROBLEM SIZE:

!   - NUMBER OF NODES:   10  !

!   - NUMBER OF ELEMENTS:   25  !


-->[kL,B,T,kG]=stiffbar3d(E,A,L,dircos,ne);

-->[tc,K0]=assemb3d(elem,3,6,kG,nn,ne);
```

```
-->preplottruss3d(nod,elem,nn,ne,2)

 WARNING: Due to your configuration limitations, Scilab
     switched in a mode where mixing uicontrols and gra
     phics is not available. Type "help usecanvas" for
     more information.

-->[K,P,cDOF,cDOFasc]=bc(M,K0);

-->SPDverif(K0,cDOFasc,K);

 ACTIVE STIFFNESS MATRIX OF STRUCTURE IS NOT SINGULAR. S
     PD VERIFICATION COMPLETED SUCCESSFULLY.

-->[u0,u,P0]=SPDsolve(K0,K,P,cDOF,cDOFasc,nn);

 ACTIVE STIFFNESS MATRIX OF STRUCTURE IS NOT SINGULAR. S
     PD VERIFICATION COMPLETED SUCCESSFULLY.

-->[ue,sigma,f]=stressbar3d(E,A,B,T,tc,u0,ne);

-->sigma
 sigma  =

  - 840.79948
  - 2402.8037
    6291.7339
  - 9145.5709
  - 439.59776
    2466.2086
  - 5748.2451
    2675.4828
  - 5538.6956
  - 641.70214
  - 736.33232
    2189.4085
  - 5293.5537
    2522.477
  - 4186.2732
    2225.3412
  - 4483.8499
    1887.3184
    1736.1682
```
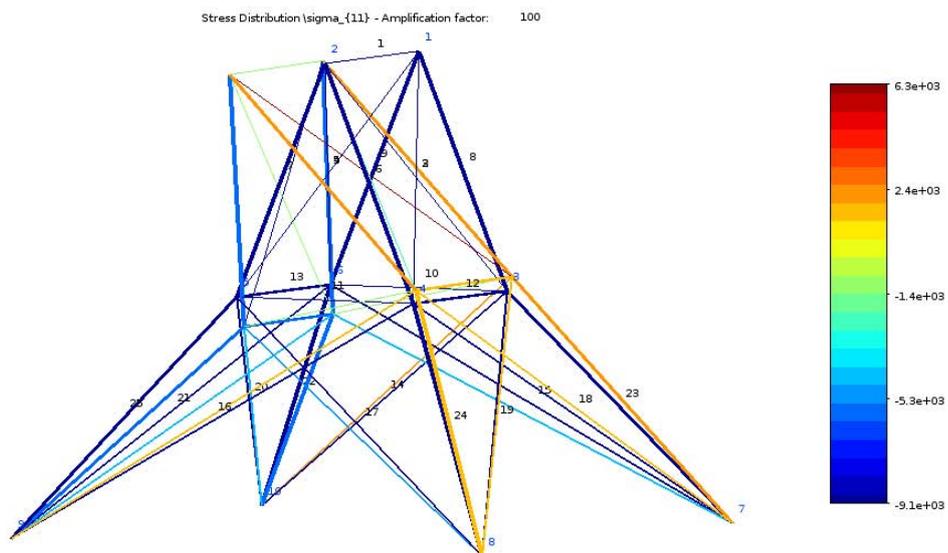
```
  -  4046.7731
  -  4288.0976
  -  5188.9955
     2664.4886
     2166.6428
  -  5706.9842
```

```
-->postplottruss3d(nod,elem,nn,ne,2,sigma,u0,100,24)
```

Stress Distribution \sigma_{11} - Amplification factor: 100

Flow Diagram of CEMEF-UTN v1.1-6

Implementation of 3D Linear Bar Element (bar3d)