UPC, CIMNE, MASTER IN NUMERICAL METHODS

Computational Mechanics Tools
Course GiD Project.

Inocencio Castañar

January 15, 2017

Contents

1 Configuration Files **3**

1.1 Materials file.mat 3

1.2 Problems and intervals data file .prb 5

1.3 Conditions file .cnd 7

2 Template file .bas **9**

3 Execution file for the problem type.bat **11**

4 Data for dynamical analysis **12**

5 Units file .uni **14**

6 html help **15**

7 Examples **17**

7.1 Corbel with Nodal Force on the far right 17

7.2 exB 19

7.2.1 Static analysis 19

7.2.2 Static analysis 24

8 Problems found and solved **25**

1 Configuration Files

1.1 Materials file.mat

The materials file stores the physical properties of the material under study for the problem type. To know which properties are needed for the solver we can consult the user-manual which is located in the folder html, inside FRAME3DD.gid . There, we can see a table with different materials and their properties at ***7.5 Approximate Properties of Structural Materials:***

	Young's Modulus E N/mm^2	Shear Modulus G N/mm^2	Thermal Expansion Coefficient a /deg.C	Mass Density d T/mm^3	Modulus per Density E/d mm^2/s^2
Steel A36	200000	79300	11.7e-6	7.85e-9	2.55e13
Boron Fiber-Epoxy	106000	38000	30.0e-6	2.00e-9	5.30e13
Carbon Fiber-Epoxy	83000	30000	30.0e-6	1.54e-9	5.39e13
Aluminum 2024-T4	73100	28000	23.2e-6	2.78e-9	2.63e13
Aluminum 6061-T6	68900	26000	23.6e-6	2.70e-9	2.55e13
Kevlar Fiber-Epoxy	40000	50000	30.0e-6	1.40e-9	2.86e13
Glass Fiber-Epoxy	22000	80000	30.0e-6	1.97e-9	1.12e13
Magnesium AM1000A	44800	17500	25.2e-6	1.80e-9	2.49e13
Douglas Fir	12400	4600	30.0e-6	0.50e-9	2.48e13

Figure 1.1: Material Properties obtained at the user manual.

Moreover, it is needed cross-section properties. Putting them into this file, we allow the user to modify and create different materials with the same physical properties but changing the cross-section. Remember that not all the pieces of a frame must have the same cross-section. In particular, it is needed: Cross-sectional area of a prismatic frame element, shear areas in the local y-axis and z-axis of a prismatic frame element, torsional moment of inertia of a frame element, Moments of inertia for bending about the local y axis and z-axis, the roll angle of the frame element, section modulus in y-direction and z-direction and torsion shear constant.

As a example, it is shown the code for one material:

```
NUMBER: 1 MATERIAL: Steel_A36
QUESTION: YOUNG_(E)
VALUE: 200000
QUESTION: SHEAR_MODUL_(G)
VALUE: 79300
QUESTION: Thermal_Exp_Coeff_(a)
VALUE: 11.7e-6
QUESTION: Mass_density_(d)
VALUE: 7.85e-9
QUESTION: Modulus_per_density_(E/d)
VALUE: 2.55e13
QUESTION: Ax#UNITS#
VALUE: 0mm^2
```

```
HELP: Cross-sectional area of a prismatic frame element
QUESTION: Asy#UNITS#
VALUE: 0mm^2
HELP: Shear area in the local y-axis of a prismatic frame element
QUESTION: Asz#UNITS#
VALUE: 0mm^2
HELP: Shear area in the local z-axis of a prismatic frame element
QUESTION: Jx#UNITS#
VALUE: 0mm^4
HELP: Torsional moment of inertia of a frame element
QUESTION: Iy#UNITS#
VALUE: 0mm^4
HELP: Moment of inertia for bending about the local y axis
QUESTION: Iz#UNITS#
VALUE: 0mm^4
HELP: Moment of inertia for bending about the local z axis
QUESTION: p:
VALUE: 0
HELP: the roll angle of the frame element, in degrees
QUESTION: Sy#UNITS#
VALUE: 0mm^3
HELP: Section Modulus in y-direction
QUESTION: Sz#UNITS#
VALUE: 0mm^3
HELP: Section Modulus in z-direction
QUESTION: C#UNITS#
VALUE: 0mm^3
HELP: Torsion Shear Constant
END MATERIAL
```

In GiD, the information pertaining to the "FRAME3DD.mat" file is managed in the materials window, located in Data - -> Materials.

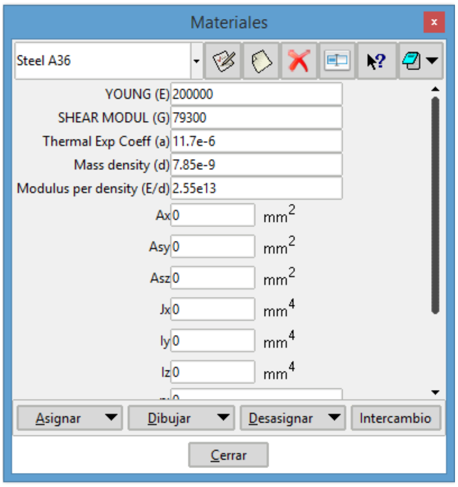


Figure 1.2: Materials window.

1.2 PROBLEMS AND INTERVALS DATA FILE .PRB

prb.file contain all the information about general problem and intervals data. The general problem data is all the information required for performing the analysis and it does not concern any particular geometrical entity. The properties put in this file will remain constant for the whole problem if they are inside the problem data part, whilst the properties which are introduced inside the interval data part can be changed for the different intervals that the user wants to use. In this case, all the data has been placed in the problem data part due to some technical problems which will be commented on the following section.

Taking a look to the user-manual, at ***9. Input Data Format*** we can see some properties which are fixed in the program:

```
shear        # 1=Do, 0=Don't include shear deformation effects
geom         # 1=Do, 0=Don't include geometric stiffness effects
exagg_static # exaggeration factor for static mesh deformations
scale        # zoom scale for 3D plotting
dx           # length of x-axis increment for frame element internal force data, mm
             # if dx is -1 then internal force calculations are skipped
```

Figure 1.3: Different properties to be introduced at the problem data. Static Analysis

```
nM      # number of desired dynamic modes

# if nM is set to 0 (zero) the remaining Input Data may be omitted

Mmethod # 1= Subspace-Jacobi iteration, 2= Stodola (matrix iteration) method
lump    # 0= consistent mass matrix, 1= lumped mass matrix
tol     # frequency convergence tolerance  approx 1e-4
shift   # frequency shift-factor for rigid body modes, make 0 for pos.def. [K]
exagg_modal  # exaggerate modal mesh deformations
```

Figure 1.4: Different properties to be introduced at the problem data. Dynamic Analysis

In addition, we can add gravitational acceleration for self-weight loading, so that the user can decide whether he want to include self-weight loading in the problem or not.

As a example, a part of the file is shown here:

```
PROBLEM DATA

QUESTION: gx:
VALUE: 0

QUESTION: gy:
VALUE: 0
```

```
QUESTION: gz:
VALUE: −9810
HELP: Include the gravity effects in directions x,y,z (mm/s^2)

QUESTION: Shear_deformation#CB#(1,0)
VALUE: 0
HELP: Include/Don't include shear deformation effects

QUESTION: Geometric_stiffness#CB#(1,0)
VALUE: 0
HELP: Include/Don't include geometric stiffness effects

QUESTION: Exaggeration_factor
VALUE: 5
HELP: Exaggeration factor for static mesh deformations

QUESTION: Plot_scaling
VALUE: 1
HELP: Zoom scale for 3D plotting

QUESTION: x−axis_increment_(internal_forces)
VALUE: −1
HELP: Length of x−axis increment for frame element internal force data (mm), if its
    value is −1 then internal force calculations are skipped
```

In GiD, the information in the "FRAME3DD.prb" file is managed in the materials window, which is located in Data - - > Problem Data.

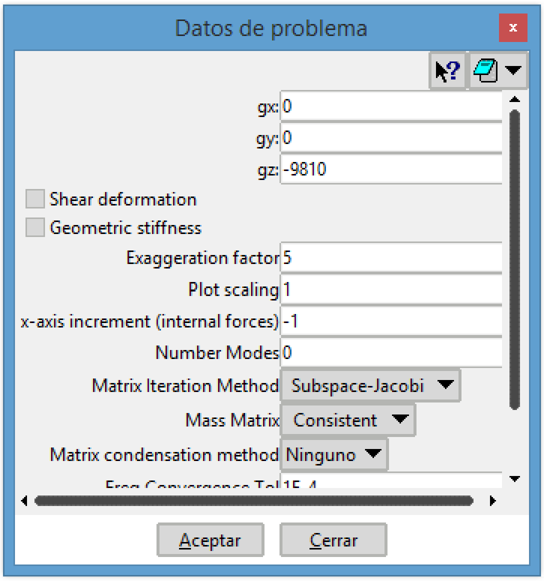


Figure 1.5: Problem Data window.

1.3 CONDITIONS FILE .CND

This file specifies the boundary and/or load conditions of the problem type in question. To know which kind of conditions the solver can use, we have to look at ***9. Input Data Format*** where the different loading and boundary cases are illustrated. The different conditions that have been created are:

- ***Fixed-Displacements*** which allow us to put if some displacement or rotation of any point are fixed.
- ***Prescribed-Displacements*** which allow us to put if some point has some initial displacement.
- ***Point-Load*** which allow us to put if there is some external nodal force or moment at some point.
- ***Distributed-Load*** which allow us to put if some distributed uniform load is applied at any line.
- ***Trapezoidally-Distributed-Load*** which allow us to put if some distributed trapezoidal load is applied at any line.
- ***Thermal-Load*** which allow us to put if some thermal load is applied at any line.

As it has been created the data for the dynamical analysis, we have to include the dynamic loads:

- ***Extra-Node-Inertia-Mass*** which allow us to assign extra mass or intertia in a node.
- ***Extra-Element-Mass*** which allow us to assign extra mass value to a frame element.
- ***Node-Condensation*** which allow us to put which nodes are condensed.

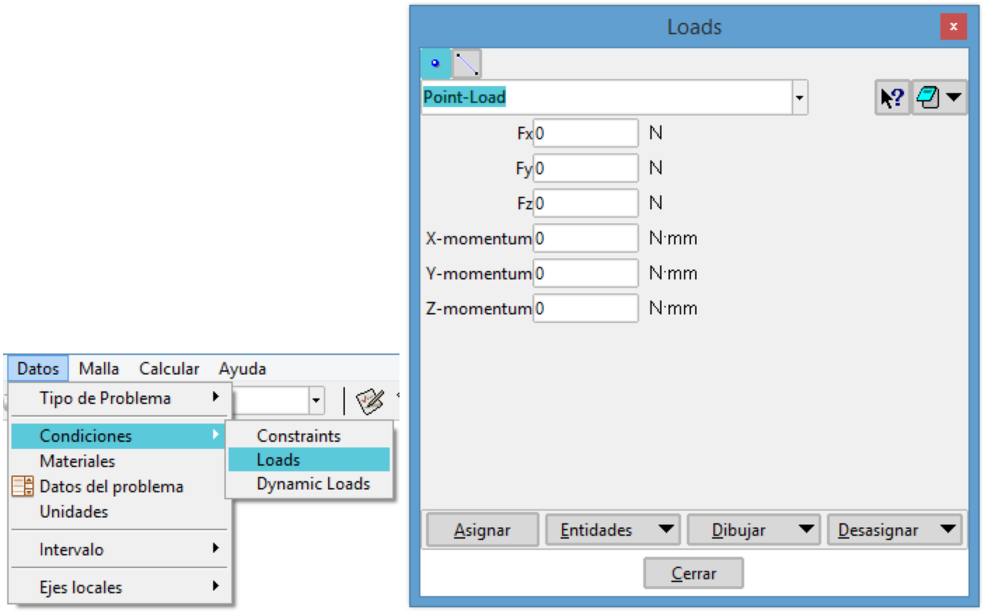
As a example, it is shown the box of the user-manual where we can extract the needed data of Distributed-Load and its format in the file:

```
nU                    # number of uniformly-distributed element loads (local)
#.elmnt  X-load   Y-load   Z-load   uniform member loads in member coordinates
#         N/mm     N/mm     N/mm
 EL[1]    Ux[1]    Uy[1]    Uz[1]
    :        :        :        :
 EL[nU]   Ux[nU]   Uy[nU]   Uz[nU]
```

Figure 1.6: Distributed-Load table.

```
CONDITION: Distributed-Load
CONDTYPE: over lines
CONDMESHTYPE: over elements
QUESTION: X-load#UNITS#
VALUE: 0N/mm
HELP: Concentrated point load in the local X-direction
QUESTION: Y-load#UNITS#
VALUE: 0N/mm
HELP: Concentrated point load in the local Y-direction
QUESTION: Z-load#UNITS#
VALUE: 0N/mm
HELP: Concentrated point load in the local Z-direction
END CONDITION
```

In GiD, the information in the "FRAME3DD.cnd" file is managed in the conditions window, which is found in Data - - > Conditions.



(a) Condition Menu (b) Condition Window

Figure 1.7: Condition window

2 TEMPLATE FILE .BAS

This file will define the format of the .dat text file created by GiD. It will store the geometric and physical data of the problem. The .dat file will be the input to the calculating module. To create it we can consult the GiD Customization Help, very useful to complete this work.

To know the structure of the input file requested by the solver, we can take a look to ***9. Input Data Format*** and reproduce our data to obtain the same input file.

```
# node data ...

nN                          # number of nodes
#.node        X-coord  Y-coord  Z-coord  radius
#                 mm       mm       mm       mm
  N[1]            x[1]     y[1]     z[1]     rj[1]
    :               :        :        :        :
  N[nN]           x[nN]    y[nN]    z[nN]    rj[nN]


# reaction data ...

nR                             # number of nodes with reactions
#.node    X      Y      Z       XX      YY      ZZ       0:free, 1:fixed
  N[1]    Rx[1]  Ry[1]  Rz[1]   Rxx[1]  Ryy[1]  Rzz[1]
    :       :      :      :       :       :       :
  N[nR]   Rx[nR] Ry[nR] Rz[nR]  Rxx[nR] Ryy[nR] Rzz[nR]
```

Figure 2.1: Distributed-Load table.

And this would be the needed code to be written in the file .bas to obtain the previous input data:

```
# node data ...

*npoin                    # number of nodes

#.node        X-coord  Y-coord  Z-coord  radius
*set elems(all)
*loop nodes
*format "%5i%14.5e%14.5e%14.5e"
*NodesNum *NodesCoord(1) *NodesCoord(2) *NodesCoord(3)      0
*end nodes


# reaction data ...
*set Cond Fixed-Displacement *Nodes
*set var FD=CondNumEntities
*set Cond Prescribed-Displacement *Nodes
*set var PD=CondNumEntities
*operation(FD+PD)                                # number of nodes with reactions

#.node     X       Y       Z       XX       YY       ZZ       0:free, 1:fixed
*set Cond Fixed-Displacement *Nodes
*loop Nodes *OnlyInCond
```

```
*format "%5i%5i%5i%5i%5i%5i%5i"
*NodesNum *Cond(X-Displacement) *Cond(Y-Displacement) *Cond(Z-Displacement) *Cond(
    X-Rotation) *Cond(Y-Rotation) *Cond(Z-Rotation)
*end Nodes

*set Cond Prescribed-Displacement *Nodes
*loop Nodes *OnlyInCond
*for(i=1;i<=CondNumFields;i=i+1)
        *if(Cond(i,real)==0)
        *set var n_Imp_Disp(*i)=0
        *else
        *set var n_Imp_Disp(*i)=1
        *end
*end for
*format "%5i%5i%5i%5i%5i%5i%5i"
*NodesNum *n_Imp_Disp(1) *n_Imp_Disp(2) *n_Imp_Disp(3) *n_Imp_Disp(4) *n_Imp_Disp(5)
    *n_Imp_Disp(6)
*end nodes
```

3 EXECUTION FILE FOR THE PROBLEM TYPE.BAT

This file connects the data file(s) (.dat) to the calculating module. To know how to create this file we can consult cmas2d tutorial which explains it to us. Moreover, we can use the following table which is located at ***11. Command-line options***:

```
-----------------------------------------------------------------------
 -i   InFile   the  input data file name --- described in the manual
 -o  OutFile   the output data file name
 -h            print this help message and exit
 -v            display program version, website, brief help info and exit
 -a            display program version, website, and exit
 -c            data check only - the output data reviews the input data
 -q            suppress screen output except for warning messages
 -w            write stiffness and mass matrices to files named Ks Kd Md
 -x            suppress writing of 't' or 'c' for sign of axial forces
 -s  On|Off    On: include shear deformation or Off: neglect ...
 -g  On|Off    On: include geometric stiffness or Off: neglect ...
 -e  value     static deformation exaggeration factor for gnuplot output
 -z            force X-Y-Z plotting
 -l  On|Off    On: lumped mass matrix or Off: consistent mass matrix
 -f  value     modal frequency shift for unrestrained structures
 -m   J|S      modal analysis method: J=Jacobi-Subspace or S=Stodola
 -t  value     convergence tolerance for modal analysis
 -p  value     pan rate for mode shape animation
 -r  value     matrix condensation method: 0, 1, 2, or 3
-----------------------------------------------------------------------
```

Figure 3.1: Command-line options.

The execution file end up as:

```
del %2\%1.post.res
del %2\%1.out

rem OutputFile: $2/$1.out

%3\Frame3DD.exe -i %2\%1.dat -o %2\%1.out -x
```

As a compulsory work, we are requested to launch the calculations and show our console output. First of all we delete the previous results files to avoid confusion. Later on with the option rem OutputFile we can demand the program to show us the output while processing.

4 Data for dynamical analysis

As an optional things, we are requested to create the dynamical analysis data. To do that we have to introduce more code in different files:

- **Problem Data file**: As we have shown at Figure 1.4 there are some properties which can remain constant for the whole problem and are involved in the dynamic analysis.
- **Conditions file**: As we have shown at Section 1.3 Conditions File .cnd the dynamic loads must be included there by following the same steps than for the shown example.
- **Template file**: By following ***9. Input Data Format*** in the user-manual we can localize the desired input data:

```
# dynamic analysis data ...

nM      # number of desired dynamic modes

# if nM is set to 0 (zero) the remaining Input Data may be omitted

Mmethod # 1= Subspace-Jacobi iteration, 2= Stodola (matrix iteration) method
lump    # 0= consistent mass matrix, 1= lumped mass matrix
tol     # frequency convergence tolerance  approx 1e-4
shift   # frequency shift-factor for rigid body modes, make 0 for pos.def. [K]
exagg_modal  # exaggerate modal mesh deformations


# extra node inertia data ...
nI      # number of nodes with extra node mass or rotatory inertia
#.node  mass      XX-inertia   YY-inertia   ZZ-inertia
#       tonne     tonne.mm^2   tonne.mm^2   tonne.mm^2
 N[1]   EMs[1]    EMx[1]       EMy[1]       EMz[1]
   :      :          :            :            :     # (global coordinates)
 N[nI]  EMs[nI]   EMx[nI]      EMy[nI]      EMz[nI]


# extra frame element mass data ...
nX      # number of frame elements with extra mass
#.elmnt   extra mass
#         tonne
 EL[1]    EMs[1]
    :        :
 EL[nX]   EMs[nE]
```

Figure 4.1: Part of the input file for dynamic analysis.

As example, it is shown the code to produce the previous part of the input data:

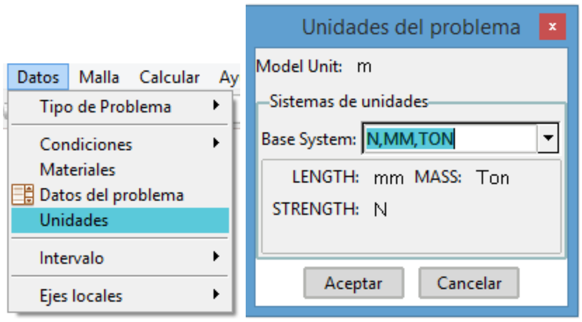
```
# dynamic analysis data

*GenData(Number_Modes)          # number of desired dynamic modes

*if(strcmp(GenData(Matrix_Iteration_Method),"Subspace-Jacobi")==0)
1       # 1=Subspace-Jacobi iteration, 2=Stodola (matrix iteration) method
*elseif(strcmp(GenData(Matrix_Iteration_Method),"Stodola"))
2       # 1=Subspace-Jacobi iteration, 2=Stodola (matrix iteration) method
*endif
```

```
*if(strcmp(GenData(Mass_Matrix),"Consistent")==0)
0       # 0=consistent mass matrix, 1=lumped mass matrix
*elseif(strcmp(GenData(Mass_Matrix),"Lumped"))
1       # 0=consistent mass matrix, 1=lumped mass matrix
*endif

*GenData(Freq_Convergence_Tol)    # frequency convergence tolerance aprox 1e-4
*GenData(Freq_Shift_Factor)       # frequency shift-factor for rigid body modes, make 0
    for pos.def. [K]
*GenData(6)                       # exaggerate modal mesh deformations

# extra node inertia data...

*Set Cond  Extra-Node-Inertia-Mass *nodes
*set var nI(int)=CondNumEntities(int)
*nI         #number of nodes with extra mass or rotatory inertia

#.node    mass      XX-inertia     YY-inertia      ZZ-inertia
#         tonne     tonne.mm^2     tonne.mm^2      tonne.mm^2
*Set Cond  Extra-Node-Inertia-Mass *nodes
*if(CondNumEntities(int)>0)
*loop nodes *OnlyInCond
*NodesNum  *cond(1) *cond(2)     *cond(3)        *cond(4)
*end nodes
*endif

# extra frame element mass data...

*Set Cond  Extra-Element-Mass *elems
*set var nX(int)=CondNumEntities(int)
*nX         #number of frame elements with extra mass
```

5 UNITS FILE .UNI

As an optional work we are requested to add GiD units to the data fields. To understand how to do it we can consult GiD Customization manual, where everything necessary is there.

It is needed to create a new file .uni which contains the different relations among units to let the user use the different options. Moreover we have to add the beggining units to our properties.Some pictures are shown from the results obtained:



(a) Units Menu (b) Units Window

Figure 5.1: Units window

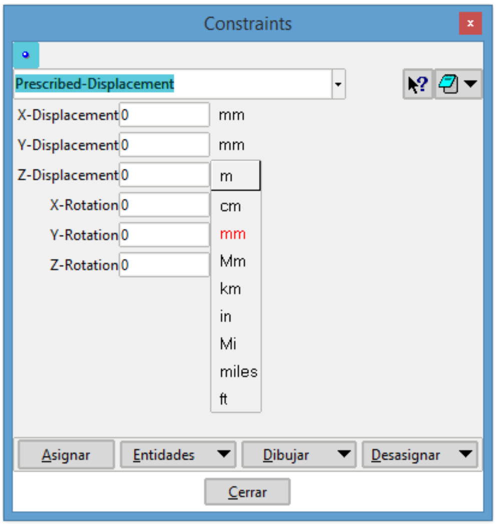


Figure 5.2: Possibility to choose different units for the case of Prescribed-Displacement.

6 HTML HELP

As an optional work, we are requested to show the html help with GiD viewer and add an entry in the corresponding GiD menu. By consulting de GiD Customization we can find an example which explains exactly what we have to do. The only thing needed here is to write two lines in the file .tcl :

```
# Adition of html help in the menu help
GiDMenu::InsertOption "Help" [list "html_help_FRAME3DD"] 0 PREPOST {HelpWindow "
    CUSTOM_HELP" "problemtypes/FRAME3DD.gid/html/user-manual.html"} "" "" insert _
GiDMenu::UpdateMenus
```

where first line creates an option inside GiD help menu and search the folder where the html file is placed. The second line updates the menus per each operation in order not to lose this change. It is obtained the following menu and help:

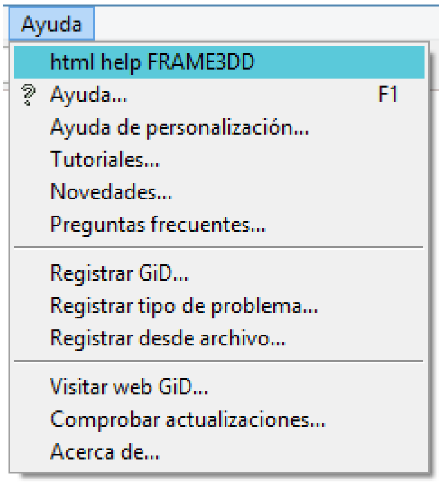


Figure 6.1: Help Menu.

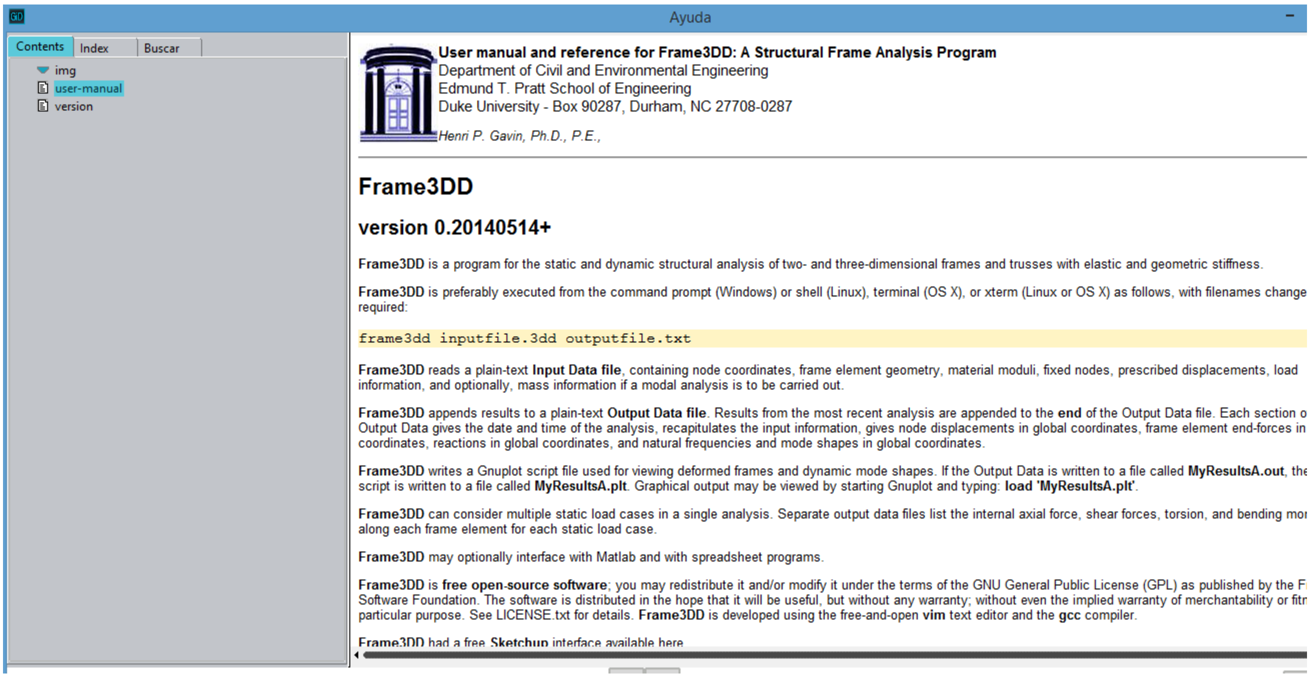


Figure 6.2: html help FRAME3DD.

7 EXAMPLES

7.1 CORBEL WITH NODAL FORCE ON THE FAR RIGHT

As a first example, it is wanted to solved a basic exercise to see if the output works properly. It is considered a frame with clamped support on the far left, which means displacement and rotations zero and with a nodal force applied on the far right.

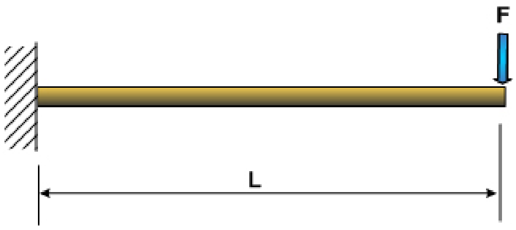


Figure 7.1: Corbel with clamped support condition on the far left and nodal force on the far right.

Let's assume $L = 1m$ and $F = -100N$. The needed material properties will be $E = 200.000MPa$ and $I = 492mm^4$ Neglecting self-weight load, the analytical result of the problem is:

$$y_{max} = \frac{L^3}{3EI} F = -338,75mm \tag{7.1}$$

Now introducing the example on GiD, it is obtained the following output result, in Calculate - - > View process info...

L O A D   C A S E   1   O F   1 ...

N O D E   D I S P L A C E M E N T S   (global)

Node	X-dsp	Y-dsp	Z-dsp	X-rot	Y-rot	Z-rot
1	0.0	-338.753388	0.0	0.0	0.0	-0.508130
2	0.0	-313.368056	0.0	0.0	0.0	-0.506860
3	0.0	-288.109756	0.0	0.0	0.0	-0.503049
4	0.0	-263.105522	0.0	0.0	0.0	-0.496697
5	0.0	-238.482385	0.0	0.0	0.0	-0.487805
6	0.0	-214.367378	0.0	0.0	0.0	-0.476372
7	0.0	-190.887534	0.0	0.0	0.0	-0.462398
8	0.0	-168.169885	0.0	0.0	0.0	-0.445884
9	0.0	-146.341463	0.0	0.0	0.0	-0.426829
10	0.0	-125.529302	0.0	0.0	0.0	-0.405234
11	0.0	-105.860434	0.0	0.0	0.0	-0.381098
12	0.0	-87.461890	0.0	0.0	0.0	-0.354421
13	0.0	-70.460705	0.0	0.0	0.0	-0.325203
14	0.0	-54.983909	0.0	0.0	0.0	-0.293445
15	0.0	-41.158537	0.0	0.0	0.0	-0.259146
16	0.0	-29.111619	0.0	0.0	0.0	-0.222307
17	0.0	-18.970190	0.0	0.0	0.0	-0.182927
18	0.0	-10.861280	0.0	0.0	0.0	-0.141006
19	0.0	-4.911924	0.0	0.0	0.0	-0.096545
20	0.0	-1.249153	0.0	0.0	0.0	-0.049543

Figure 7.2: Output File.

We can see that for the node 1 which is the one which is located on the far right, the displacement is $y = -338,75mm$. It means that our problem works properly.

Some pictures are shown:

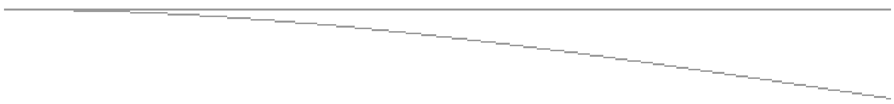


Figure 7.3: Original and Deformed geometry.

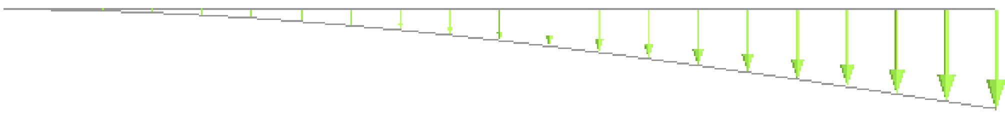


Figure 7.4: Y Displacement vectors.

7.2 EXB

7.2.1 STATIC ANALYSIS

Now we want to study the example B which is included in the FRAME3DD package. We can obtain the Input file and the outfile and compare it with our ones.

All the needed properties, loads, displacements... can be read thanks to the input file:

```
Example B: a pyramid-shaped frame --- static and dynamic analysis (N,mm,ton)

5                               # number of nodes
#.node  x       y       z       r
#       mm      mm      mm      mm

1       0.0     0.0     1000    0.0
2       -1200   -900    0.0     0.0
3        1200   -900    0.0     0.0
4        1200    900    0.0     0.0
5       -1200    900    0.0     0.0

4                               # number of nodes with reactions
#.n     x  y  z xx yy zz          1=fixed, 0=free

  2     1  1  1  1  1  1
  3     1  1  1  1  1  1
  4     1  1  1  1  1  1
  5     1  1  1  1  1  1

4                               # number of frame elements
#.e n1 n2 Ax    Asy     Asz     Jxx     Iyy     Izz     E       G     roll density
#   .  .  mm^2  mm^2    mm^2    mm^4    mm^4    mm^4    MPa     MPa   deg  T/mm^3

1 2 1   36.0    20.0    20.0    1000    492     492     200000  79300  0 7.85e-9
2 1 3   36.0    20.0    20.0    1000    492     492     200000  79300  0 7.85e-9
3 1 4   36.0    20.0    20.0    1000    492     492     200000  79300  0 7.85e-9
4 5 1   36.0    20.0    20.0    1000    492     492     200000  79300  0 7.85e-9


1               # 1: include shear deformation
1               # 1: include geometric stiffness
10.0            # exaggerate static mesh deformations
2.5             # zoom scale for 3D plotting
20.0            # x-axis increment for internal forces, mm
                # if dx is -1 then internal force calculations are skipped.

3                               # number of static load cases
                                # Begin Static Load Case 1 of 3

# gravitational acceleration for self-weight loading (global)
#.gX            gY              gZ
```

```
#.mm/s^2          mm/s^2            mm/s^2
  0               0                 -9806.33

1                                   # number of loaded nodes
#.e       Fx        Fy       Fz       Mxx       Myy       Mzz
#         N         N        N        N.mm      N.mm      N.mm
 1       100      -200     -100      0.0       0.0       0.0
0                                   # number of uniform loads
0                                   # number of trapezoidal loads
0                                   # number of internal concentrated loads
0                                   # number of temperature loads
0                                   # number of nodes with prescribed displacements
                                    # End   Static Load Case 1 of 3

                                    # Begin Static Load Case 2 of 3

# gravitational acceleration for self-weight loading (global)
#.gX              gY                gZ
#.mm/s^2          mm/s^2            mm/s^2
  0               0                 -9806.33

0                                   # number of loaded nodes
2                                   # number of uniform loads
#.e    Ux    Uy    Uz
#     N/mm N/mm N/mm
  2    0    0.1    0
  1    0    0     0.1
2                                   # number of trapezoidally distributed loads
#.e      x1        x2        w1       w2
#        mm        mm        N/mm     N/mm
  3      20        80        0.01     0.05      # location and loading - local x-axis
          0         0         0        0        # location and loading - local y-axis
         80       830       -0.05     0.07      # location and loading - local z-axis

  4       0         0         0        0        # location and loading - local x-axis
         68       330        0.05     0.00      # location and loading - local y-axis
         80       830       -0.05     0.07      # location and loading - local z-axis

0                                   # number of internal concentrated loads
1                                   # number of temperature loads
#.e   alpha    hy    hz    Ty+   Ty-   Tz+   Tz-
#     /degC    mm    mm    degC  degC  degC  degC
1     12e-6    10    10    20    10    10    -10
0                                   # number of nodes with prescribed displacements
                                    # End   Static Load Case 2 of 3

                                    # Begin Static Load Case 3 of 3

# gravitational acceleration for self-weight loading (global)
#.gX              gY                gZ
#.mm/s^2          mm/s^2            mm/s^2
  0               0                 -9806.33

0                                   # number of loaded nodes
```

```
0                               # number of uniform loads
0                               # number of trapezoidal loads
2                               # number of internal concentrated loads
#.e     Px     Py      Pz     x
#       N      N       N      mm
  1     0      100    -900   600
  2     0     -200     200   800
0                               # number of temperature loads
0                               # number of nodes with prescribed displacements
                                # End   Static Load Case 3 of 3


6                               # number of desired dynamic modes of vibration
1                               # 1: subspace Jacobi     2: Stodola
0                               # 0: consistent mass ... 1: lumped mass matrix
1e-9                            # mode shape tolerance
0.0                             # shift value ... for unrestrained structures
10.0                            # exaggerate modal mesh deformations

# nodes and concentrated mass and inertia
1                               # number of nodes with extra inertia
#.n       Mass     Ixx         Iyy         Izz
#         ton      ton.mm^2    ton.mm^2    ton.mm^2
1         0.1      0           0           0

0                               # frame elements with extra mass

6                               # number of modes to animate, nA
 1  2  3  4 5 6                 # list of modes to animate - omit if nA == 0
2                               # pan rate during animation

# End of input data file for example B
```

The geometry yields as:

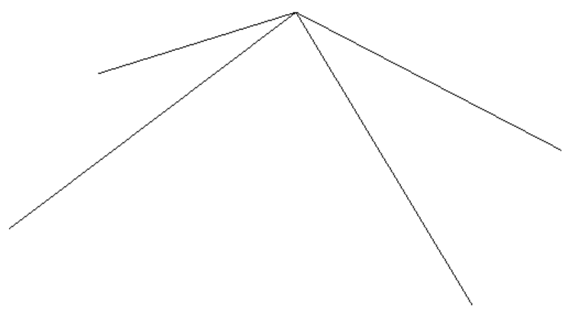
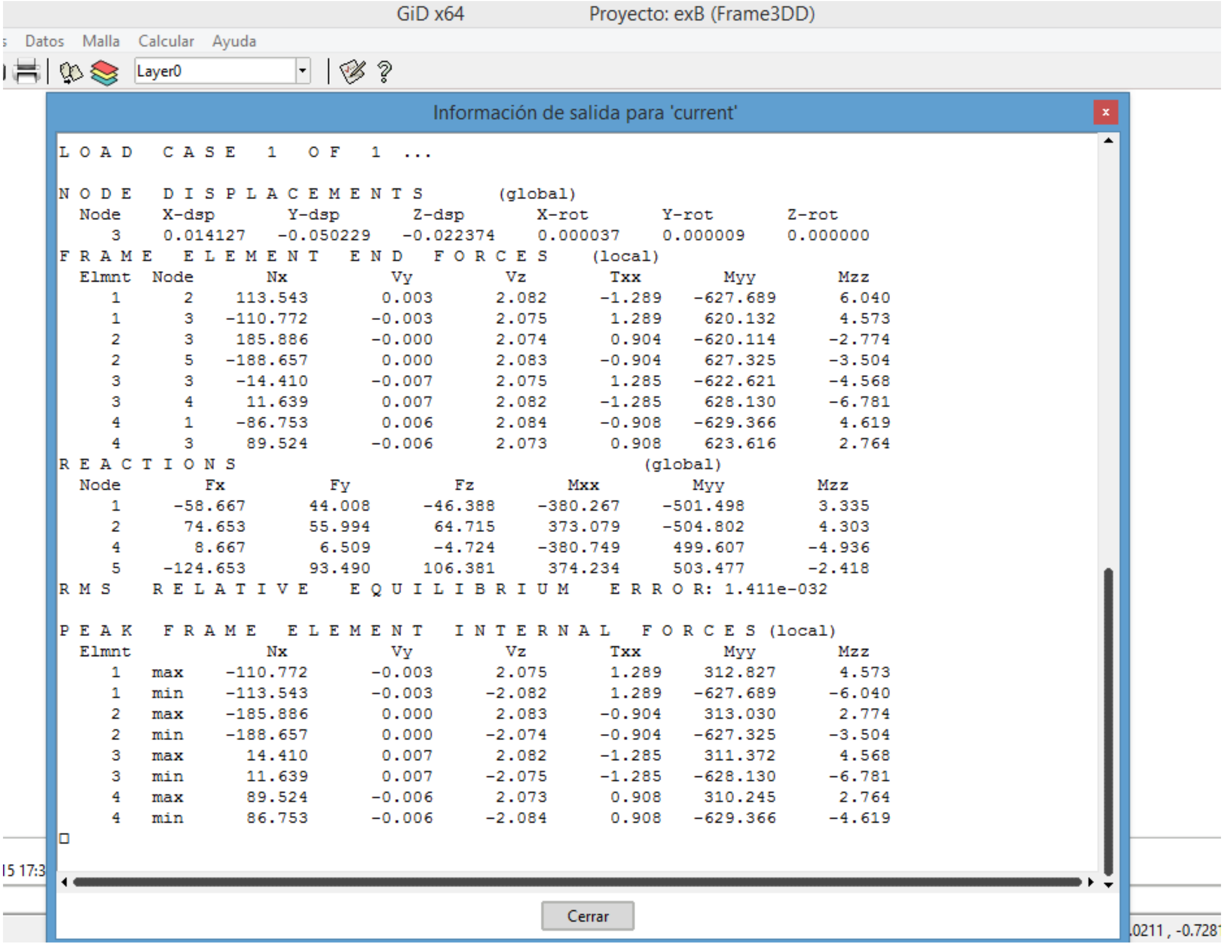


Figure 7.5: exB Geometry.

The Output file could be obtained in Calculate - - > View process info...




```
L O A D   C A S E   1   O F   1  ...

N O D E   D I S P L A C E M E N T S       (global)
  Node    X-dsp       Y-dsp       Z-dsp       X-rot       Y-rot       Z-rot
     3    0.014127   -0.050229   -0.022374    0.000037    0.000009    0.000000
F R A M E   E L E M E N T   E N D   F O R C E S     (local)
  Elmnt  Node       Nx          Vy         Vz        Txx        Myy        Mzz
     1      2    113.543      0.003      2.082     -1.289   -627.689      6.040
     1      3   -110.772     -0.003      2.075      1.289    620.132      4.573
     2      3    185.886     -0.000      2.074      0.904   -620.114     -2.774
     2      5   -188.657      0.000      2.083     -0.904    627.325     -3.504
     3      3    -14.410     -0.007      2.075      1.285   -622.621     -4.568
     3      4     11.639      0.007      2.082     -1.285    628.130     -6.781
     4      1    -86.753      0.006      2.084     -0.908   -629.366      4.619
     4      3     89.524     -0.006      2.073      0.908    623.616      2.764
R E A C T I O N S                                        (global)
  Node        Fx          Fy          Fz         Mxx         Myy         Mzz
     1     -58.667      44.008     -46.388    -380.267    -501.498       3.335
     2      74.653      55.994      64.715     373.079    -504.802       4.303
     4       8.667       6.509      -4.724    -380.749     499.607      -4.936
     5    -124.653      93.490     106.381     374.234     503.477      -2.418
R M S    R E L A T I V E    E Q U I L I B R I U M    E R R O R: 1.411e-032

P E A K   F R A M E   E L E M E N T   I N T E R N A L   F O R C E S (local)
  Elmnt             Nx          Vy         Vz        Txx        Myy        Mzz
     1   max   -110.772     -0.003      2.075      1.289    312.827      4.573
     1   min   -113.543     -0.003     -2.082      1.289   -627.689     -6.040
     2   max   -185.886      0.000      2.083     -0.904    313.030      2.774
     2   min   -188.657      0.000     -2.074     -0.904   -627.325     -3.504
     3   max     14.410      0.007      2.082     -1.285    311.372      4.568
     3   min     11.639      0.007     -2.075     -1.285   -628.130     -6.781
     4   max     89.524     -0.006      2.073      0.908    310.245      2.764
     4   min     86.753     -0.006     -2.084      0.908   -629.366     -4.619
```

Figure 7.6: exB Output File.

which are the same than in the exB.out implemented by the webside

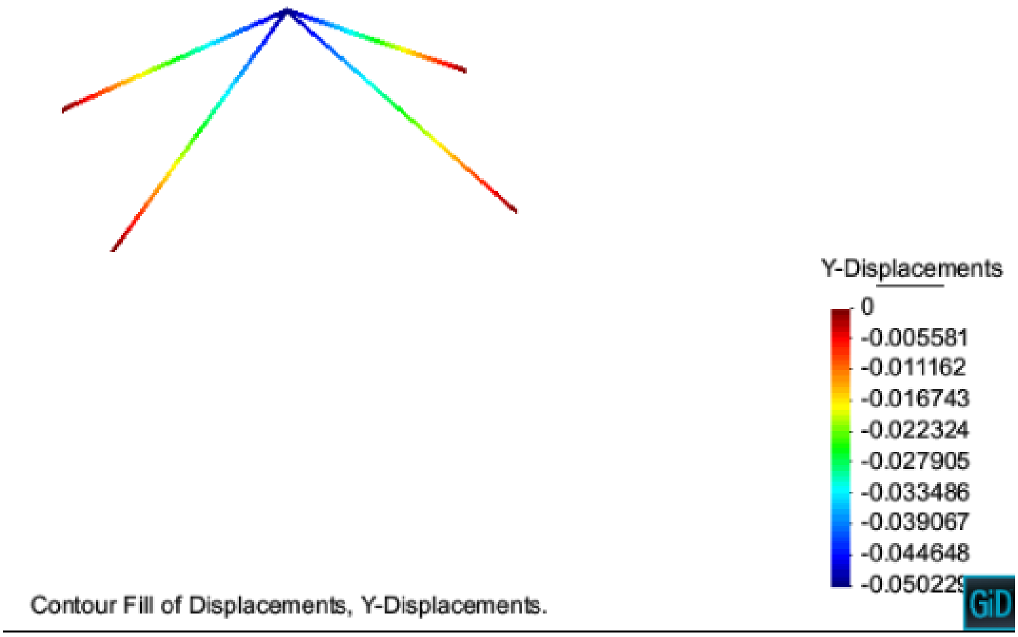
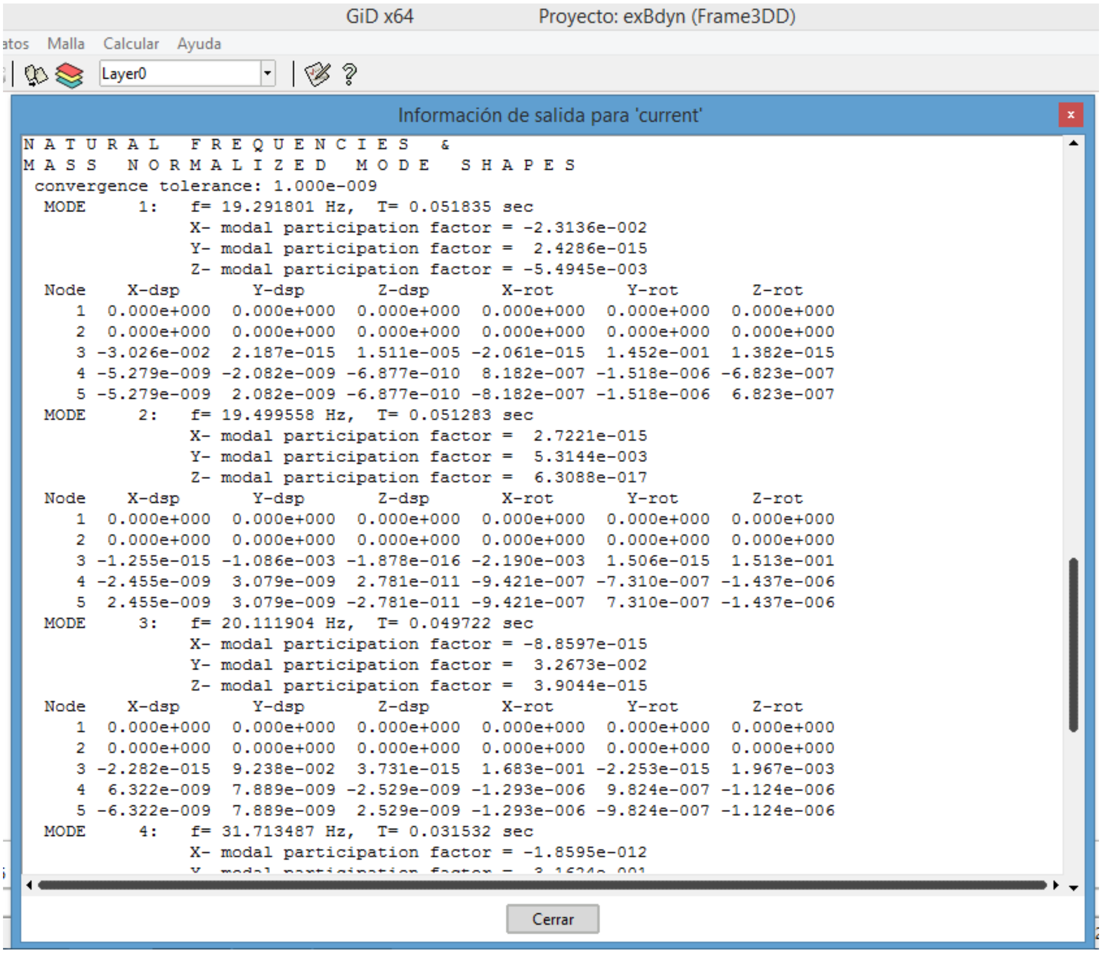


Figure 7.7: exB CountourFill Y Displacement.

7.2.2 STATIC ANALYSIS

In addition, it has been studied the dynamic analysis:




```
N A T U R A L   F R E Q U E N C I E S   &
M A S S   N O R M A L I Z E D   M O D E   S H A P E S
 convergence tolerance: 1.000e-009
  MODE     1:   f= 19.291801 Hz,  T= 0.051835 sec
              X- modal participation factor = -2.3136e-002
              Y- modal participation factor =  2.4286e-015
              Z- modal participation factor = -5.4945e-003
  Node    X-dsp       Y-dsp       Z-dsp       X-rot       Y-rot       Z-rot
     1  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000
     2  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000
     3 -3.026e-002  2.187e-015  1.511e-005 -2.061e-015  1.452e-001  1.382e-015
     4 -5.279e-009 -2.082e-009 -6.877e-010  8.182e-007 -1.518e-006 -6.823e-007
     5 -5.279e-009  2.082e-009 -6.877e-010 -8.182e-007 -1.518e-006  6.823e-007
  MODE     2:   f= 19.499558 Hz,  T= 0.051283 sec
              X- modal participation factor =  2.7221e-015
              Y- modal participation factor =  5.3144e-003
              Z- modal participation factor =  6.3088e-017
  Node    X-dsp       Y-dsp       Z-dsp       X-rot       Y-rot       Z-rot
     1  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000
     2  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000
     3 -1.255e-015 -1.086e-003 -1.878e-016 -2.190e-003  1.506e-015  1.513e-001
     4 -2.455e-009  3.079e-009  2.781e-011 -9.421e-007 -7.310e-007 -1.437e-006
     5  2.455e-009  3.079e-009 -2.781e-011 -9.421e-007  7.310e-007 -1.437e-006
  MODE     3:   f= 20.111904 Hz,  T= 0.049722 sec
              X- modal participation factor = -8.8597e-015
              Y- modal participation factor =  3.2673e-002
              Z- modal participation factor =  3.9044e-015
  Node    X-dsp       Y-dsp       Z-dsp       X-rot       Y-rot       Z-rot
     1  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000
     2  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000  0.000e+000
     3 -2.282e-015  9.238e-002  3.731e-015  1.683e-001 -2.253e-015  1.967e-003
     4  6.322e-009  7.889e-009 -2.529e-009 -1.293e-006  9.824e-007 -1.124e-006
     5 -6.322e-009  7.889e-009  2.529e-009 -1.293e-006 -9.824e-007 -1.124e-006
  MODE     4:   f= 31.713487 Hz,  T= 0.031532 sec
              X- modal participation factor = -1.8595e-012
```

Figure 7.8: exB dynamic analysis Output File.

As we can see comapring with their outputfile the first modes are a little bit different but from the fourth one on they are identic. It means that there are something missing in the dynamic data.

8 Problems found and solved

While doing this project I have found a lot of problems. As a beginer in programing, my first contact with this project was horrible. I found it extremely difficult and hardly found the way to create the materials files.

With the passage of time, I found two fundamental pillars: GiD Customization which allows me to understand exactly what was the purpose of each file and the way to create them and FRAME3DD user-manual which allows me to learn how solver works and what kind of variables and parameters were needed.

Once I knew all this, my problem was how to learn Tcl/Tk. By asking for help to my colleagues and consulting different webpages I was able to obtain the different files.

While contrasting some examples, I saw that the dynamic analysis was a little bit different from the outputs. I think that this problem may be caused because there is something wrong in my dynamic analysis data, but I was no able to find it.

While doing an example, I saw that if I put more than one interval, I did not obtained any result. I think that this should be because there is something missing in my .bas file but I was no able to find it.

As a conclusion, I think that this problem was very difficult for me, due to my level of programming, but hard-working leads me to obtain this final result. I am very proud of my final project.